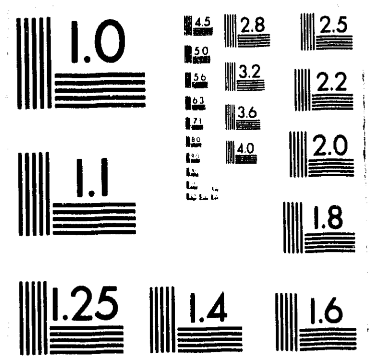

1.0
4.5
5.0
5.6
6.3
7.1
8.0
2.8
3.2
3.6
4.0
2.5
2.2
2.0
1.1
1.8
1.25
1.4
1.6

DOE/MC/24193-3503
(DE94000035)

FORCE2 - A Multidimensional Flow Program for Gas Solids Flow User's Guide

Topical Report

S.W. Burge

May 1991

Work Performed Under Contract No.: DE-AC21-89MC24193
W-31109-ENG-38

For
U.S. Department of Energy
Office of Fossil Energy
Morgantown Energy Technology Center
Morgantown, West Virginia

By
Argonne National Laboratory
Energy Systems Division
Argonne, Illinois

DOE/MC/24193-3503
(DE94000035)
Distribution Category UC-103

FORCE2 - A Multidimensional Flow Program for Gas Solids Flow User's Guide

Topical Report

S.W. Burge

Work Performed Under Contract No.: DE-AC21-89MC24193

For
U.S. Department of Energy
Office of Fossil Energy
Morgantown Energy Technology Center
P.O. Box 880
Morgantown, West Virginia 26507-0880

By
Argonne National Laboratory
Energy Systems Division
9700 South Cass Avenue
Argonne, Illinois 60439-4815

May 1991

TABLE OF CONTENTS

TABLE OF CONTENTS (cont'd)

TABLE OF CONTENTS (cont'd)

LIST OF FIGURES

LIST OF FIGURES

LIST OF TABLES

1.0 INTRODUCTION

This report describes the FORCE2 flow program input, output, and the graphical post-processor. The manual describes the steps for creating the model, executing the programs and processing the results into graphical form.

The FORCE2 post-processor was developed as an interactive program written in FORTRAN-77. It uses the Graphical Kernel System (GKS) graphics standard recently adopted by International Organization for Standardization, ISO, and American National Standards Institute, ANSI, and, therefore, can be used with many terminals. The post-processor was written with Calcomp subroutine calls and is compatible with Tektronix terminals and Calcomp and Nicolet pen plotters.

B&W has been developing the FORCE2 code as a general-purpose tool for flow analysis of B&W equipment. The version of FORCE2 described in this manual was developed under the sponsorship of ASEA-Babcock as part of their participation in the joint R&D venture, "Erosion of FBC Heat Transfer Tubes," and is applicable to the analyses of bubbling fluid beds. This manual is the principal documentation for program usage and is segmented into several sections to facilitate usage. In Section 2.0 the program is described, including assumptions, capabilities, limitations and uses, program status and location, related programs and program hardware and software requirements. Section 3.0 is a quick user's reference guide for preparing input, executing FORCE2, and using the post-processor. Section 4.0 is a detailed description of the FORCE2 input. In Section 5.0, FORCE2 output is summarized. Section 6.0 contains a sample application, and Section 7.0 is a detailed reference guide.

2.0 PROGRAM DESCRIPTION

FORCE2 (Flow in ORthogonal Coordinates Eulerian, 2-Phase) is a multidimensional flow code that considers flow of two continuous phases described in an Eulerian reference frame. For the version described in this manual, one of the phases is solid particles.

2.1 PROGRAM APPLICATION

Assumptions

- Two- or three-dimensional Cartesian coordinate systems
- Two continuous phases described by an Eulerian formulation; one of the phases is solid particles
- Compressible Navier-Stokes equation for the transient model
- Incompressible Navier-Stokes equation for the steady-state model
- Macrocontrol volume formulation with scalar nodes centered in the control volumes
- Laminar flow
- Upwind finite differencing used to discretize the convection terms and central differencing for the diffusion terms in the transient formulation
- Hybrid upwind finite differencing used to discretize the convection and diffusion terms in the steady formulation

Capabilities

- Transient or steady-state simulation

- Two- or three-dimensional parabolic or elliptical flows

- Distributed resistance modeling to simulate obstructions such as porous plates and tube bundles in the flow field

- General geometry specification using either complete blockage of control volumes (cells) or blockage of control volume surfaces

- Flow areas and volumes modified with surface permeabilities and volume porosities to account for obstructions in the flow field

- The implicit multifield solution method (Rivard, 1977) for transient simulations

- A modified SIMPLER solution algorithm (Patankar 1980, Schnipke 1986) for the steady simulation.

- Simultaneous solution of the pressure field with the modified SIMPLER algorithm

Limitations

- Incompressible flow for the steady simulation

- Newtonian fluids

- Two phases, one of which is solid particles

- Forced convection or diffusion-dominated flows. The solution method favors pressure-driven flows in contrast to buoyancy-driven flows.

2.2 STATUS

FORCE2 can be used to simulate a wide range of fluid beds, from laboratory to plant scale. Limited (due to budgetary constraints) validation of the program with data collected on laboratory-size beds has been performed.

2.3 HARDWARE/SOFTWARE REQUIREMENTS

FORCE2 can be run in its present form on any VAX-11 series virtual storage super-minicomputer or Sun 4 workstation without any modification of the source code. FORCE2 has been developed on the ARC VAX-11/785 and Sun 4 using standard FORTRAN 77. It should port to other systems without modification.

A feature of FORCE2 which aids program portability is the use of secondary storage and adjustable internal storage buffers. Currently, the internal buffers, which are one-dimensional arrays, are very large making use of the more efficient virtual memory. In this case, large arrays permit FORCE2 to be used for most 3-D problems without any FORCE2 secondary storage (disk) input-output (I/O) transfers. The I/O is handled by the architecture of the VAX virtual memory. On mainframe computers without virtual memory, the FORCE2 internal arrays can be reduced to fit within the available memory. The secondary storage I/O is then processed by FORCE2 for large 3-D problems.

To run the post-processor, a license for the GKS graphics package must be purchased. Since GKS has been adopted by ANSI as a graphics standard, it can be purchased from many different vendors.

The post-processor is written with familiar Calcomp subroutine calls and is compatible with Calcomp and Nicolet pen plotters. For previewing plots on Tektronix terminals, a PLOT 10 emulator is used. The emulator is a library of Calcomp subroutines which are linked with the post-processor. The emulator converts the Calcomp calls into graphics commands which can be interpreted by a Tektronix terminal. The post-processor conforms to the ANSI FORTRAN 77 standard and can, therefore, be readily ported to a variety of computer systems.

3.0 USING THE PROGRAM

3.1 OVERVIEW

Key steps in developing predictions with FORCE2 are illustrated in Figure 3-1. A FORCE2 data file, describing the problem geometry and operating conditions, is prepared and input to the program. FORCE2 then computes the flow field solution. During the solution, several output files are developed that describe intermediate results and that can be used to continue the solution. These files include i) a restart file that can be used to continue the simulation, ii) post-processor files that are used with the post-processor to graphically display FORCE2 predictions, and iii) printer output. During a simulation, many of these files may be developed (for example, at various times during a transient solution) depending on the input specification. During the simulation, the post-processor may be used to graphically display the predicted flow field. For example, contours of void fraction, velocities, and pressures can be displayed for all or just a part of the flow field. The post-processor reads files that are written by FORCE2 and converts the flow field predictions into graphical output.

3.2 INPUT PREPARATION

The FORCE2 input file contains a description of the problem geometry, material properties, operating conditions, and solution options. The input is free form, uses keywords to identify data values and is organized into sections, denoted paragraphs, to facilitate input preparation. The input is described in detail in Section 4.0. This section should be followed in developing an input data set.

3.2.1 Error Messages

Error messages are displayed when the input data file contains inconsistent or unrecognizable data. The error message is displayed along with the listing of the input data (FORCE2 printer output on logical Unit #6) and will appear below the erroneous input data. The severity of the error is indicated by the error level and results in the following:

Error Level	Action
1	None. A warning message only.
2	None. A noncritical error; execution continues.
3	Corrective action taken. A critical error; may not produce a usable run. But corrective action has been taken and execution continues.
4	Execution stops. A fatal error; run cannot continue.

It is recommended that all errors be corrected before performing extensive analyses.

3.3 EXECUTING FORCE2

3.3.1 Running FORCE2

A number of files are read or created by FORCE2 during a run. Because an analysis may consist of many runs and will make use of files that are created at the end of each run, the user should understand the FORCE2 I/O file structure. In particular, to efficiently use the program and to avoid I/O errors, the user should keep track of all the files created during the analysis.

Files that are used or created by FORCE2 include:

- Input data file
- Input restart file
- Output restart file
- FORCE2 output listing
- Runtime file
- FORCE2 log file
- Post-processor files
- Hydrodynamic data file for use with the ANL erosion model

The input data file, described in detail in Section 4.0, is the primary input file for FORCE2. It defines the problem geometry, operating conditions, and program operation. It is an ASCII file.

The restart files are used to continue a FORCE2 solution that was intentionally stopped to review progress. The results can be output via the post-processor before continuing. The FORCE2 program is restarted using the input restart file. The output restart file stores the data at the end of a FORCE2 run and becomes the input restart file for continuing the FORCE2 run. The restart file is a binary file.

The FORCE2 output listing (output for printing) contains the node-by-node tabular printout of the results. It is highly detailed output and may be excessively large unless the user limits the output with FORCE2 commands. The output listing is an ASCII file.

The runtime file is an ASCII file that can be used to stop the program before the time or iteration number specified in the input file. This file is described in detail in Section 4.0.

The log file, an ASCII file, can be used to monitor the solution, particularly during a transient simulation. The frequency at which the log file is written is controlled by a parameter on the runtime file. The log file is described in detail in Section 5.0

Two binary files are written for the post-processor. They contain predicted hydrodynamic operating conditions and are written periodically during a transient or steady simulation. The frequency at which data is written to these files is controlled by input (in the FORCE2 input data file) to the program. One of the post-processor files contains operating conditions (velocities, voids, etc.) throughout the flow field at the selected times, timesteps, or iterations. The other contains specific operating conditions at various places in the flow field at prescribed times or timesteps. The conditions and locations are defined in the FORCE2 input data file. This latter file is written only during a transient run of the program.

Hydrodynamic data (velocities, voids, pressures, etc.) is also periodically written to a file for subsequent use with the ANL erosion code (EROSION). The frequency of the written output is controlled through input. This data is stored as a binary file.

Typical command procedures for executing FORCE2 on a Sun workstation and a VAX computer are given in Appendix A. These procedures make the required file assignments and can be adopted for the user's system.

3.3.2 Restarts

Restart input data files generally have a limited number of input paragraphs compared to the initial input data file. Typically, a RESTART statement, a flow paragraph, and a control paragraph are required. The FLOW paragraph is used to change the solution method. The CONTROL paragraph contains the number of iterations (steady mode) or timesteps (transient mode), the under-relaxation parameters, and print control statements.

The number of iterations (steady mode) or timesteps (transient mode) is problem dependent. The solution is commonly developed by using the restart feature. Typical steps in developing a solution include:

Step 1: Make initial run and save output restart file.

Step 2: Modify input data for restart run (i.e., specify RESTART SAVE in restart paragraph).

Step 3: Rename output restart file as input restart file.

Step 4: Specify number of iterations or timesteps to be performed in next run.

Step 5: Run FORCE2 and save the newly-created output restart file. Review predictions with post-processor or examine FORCE2 output listing to determine if the analysis has been completed.

Step 6: Repeat steps 3, 4, and 5 until the desired solution has been achieved.

3.3.3 Guidelines

The guidelines for using FORCE2 the first time are as follows:

- o Review the sample problem outlined in Section 6.0
- o Acquire detailed geometrical and flow data (initial and boundary conditions) for the particular application
- o Perform any geometrical or boundary condition calculations for the problem
- o Develop the FORCE2 input data file
- o Run one iteration (steady mode) or one timestep (transient mode) with FORCE2, make a restart file and check the output
- o Perform as many successive restarts as necessary to achieve a solution, as defined below
- o Initialize the pressure field based on minimum fluidization (an option in the INITIALIZATION paragraph). The gas and solids velocity fields should be initialized to approximately satisfy continuity. One-dimensional velocity fields are adequate. This approach appears to give reasonable starting conditions.

If performing a steady simulation, the recommended guidelines are also:

- o Examine the flow field, temperature, etc., after each restart to insure that the solution is converging
- o Monitor residuals of the conservation equations. Convergence is obtained after the residuals have decreased by 2 or 3 orders of magnitude

- o Adjust under-relaxation parameters (Appendix A) to control convergence. The under-relaxation values are somewhat problem dependent, and experience will help in choosing the values. Use the defaults as a starting point. If the residuals are decreasing monotonically, leave the under-relaxation parameters alone.

- o Use the post-processor to observe changes in the flow field as the problem converges. Solutions that reach reasonably low residuals and are not changing appreciably can be stopped. Reducing the residuals to values comparable with machine roundoff is unnecessary and wastes computer time.

If performing a transient simulation, the recommended guidelines are also:

- o Monitor the mass residual and solution convergence. The log file lists the maximum mass residual along with the number of non-converged nodes if they exist. Experience indicates that reasonable solutions can be achieved, although the mass residuals at a few nodes may exceed the convergence criteria at a few timesteps. If the solution does not converge over several successive timesteps, it will probably diverge. This situation can usually be corrected by rerunning the simulation with a tighter solution tolerance or by reducing problem timestep. Experience indicates that the first symptom of a diverging solution is a predicted void fraction that is greater than 1.0 (a physically unrealistic value). If this occurs, execution stops and warning messages are written to the log file and FORCE2 output listing.

- o Apply fluidizing boundary conditions in steps. To develop reliable transient hydrodynamic conditions, experience indicates that the gas in-flow boundary condition should be applied in steps a follows:

  i) Run 0.0 - 1.0 second at the minimum fluidizing condition

  ii) Run 1.0 - 1.5 seconds with the gas in-flow increasing to its final operating value

iii) Run 1.5 - 2.0 seconds with gas in-flow at final operating value

iv) Beyond 2.0 seconds, hydrodynamics are "fully developed" and not influenced by the initial conditions

It should be noted that these are just "rules of thumb". The analyst should note that, in most simulations, the predicted hydrodynamics at the early times will be influenced by initial conditions.

3.4 POST-PROCESSOR

3.4.1 Overview

The post-processing program is a Fortran program that creates graphical output for a variety of devices (graphics terminals, plotting devices, etc.) using files written by FORCE2. The output device is selected when the processor is run.

Basic plots that can be generated using the post-processor are:

- o Geometry including control volumes
- o Contour plots
- o Profile plots
- o Velocity vector plots
- o Streamlines in 2-D applications
- o Variable vs time plots in transient applications.

PROGRAM CONVENTIONS

used by the post-processor are:

: Following the colon a user response is expected

( ) Suggested or possible user responses are shown inside the parentheses

[ ] Default responses are shown in brackets and are normally shown with the prompt. The default is chosen by pressing the <RETURN> key (abbreviated as <CR> for carriage return).

At any time during the input phase the user can interrupt the program, cancelling the previous input by hitting a <Ctrl> Z (holding the Control key while pressing the Z key). Usually the user interrupt function returns you to the main plot selection menu.

3.4.2 Running the Post-Processor

The method of running the post-processor will depend on user preferences and computer system. The approach described here is used by B&W to run on a Sun 4/260 super-workstation. It is not the only way of executing the program. However, it is recommended that the B&W approach be adopted and modified for the user's particular computer system.

The post-processor is best run using an interactive command procedure. The procedure sets up the files to be read and/or written by the processor and then runs the post-processor program. The command procedure (shell script) listed in Table A-4 of Appendix A is used by B&W.

Graphical output that is used to make hardcopy plots can be developed in one of the following ways:

Interactive Sequence

The post-processor is used to interactively display problem geometry, FORCE2 predictions, etc., on a graphics terminal. In this mode, an interactive graphics terminal (such as a Tektronix 4014) is selected as the GKS device in the GKS menu. Each time the post-processor is run in this interactive mode, a session file is created that contains a list of all user entries. After the user has displayed the desired plots at his terminal, the appropriate GKS plotting device and output file name are added to the associated session file. This modified session file is then used to run the post-processor. This final step generates a graphics file that can produce hardcopy plots on the chosen GKS device.

Non-Interactive Sequence

The post-processor is run from a non-graphics terminal. In this mode, a plotting device is selected in the GKS menu. The post-processor creates a graphic file that will produce hardcopy plots on the chosen device. The only difference between this mode and the Interactive Sequence, described above, is that the plots are not displayed on the terminal (the post-processor output device is a plotter).

3.4.3 Input Phase

The program uses data files for unit conversion, boundaries of the geometry, and FORCE2 predictions. The program saves the user's input in a session file for later use. With B&W's approach, the command procedure establishes default file names for the post-processor and then runs the processor. This initial step is not absolutely necessary because the post-processor also prompts the user for the FORCE2 file names.

GKS MENU

As noted earlier, the post-processor is a Fortran program that creates graphical output for a variety of devices using files written by FORCE2. The type of device is listed in the GKS menu. The request, with available options, is shown below.

```
---- GKS DEVICE SELECTION ----
<Enter> User defined workstation
   2501 4014 TEKTRONIX Terminal
   3100 4107 TEKTRONIX Terminal
  10300 CGM Metafile, Clear Text
   1103 HP 7475 plotter (size A)
   1325 HPLaserJet, Portrait  (150 dpi)
   1328 HPLaserJet, Landscape (150 dpi)
   1900 PostScript, Portrait  (8.5x11.0)
   1901 PostScript, Landscape (8.5x11.0)
   5300 Suntools device, grey

  Enter device type:
```

The post-processor has been optimized to work with the 4014 terminal (2501) and the postscript, landscape printer (1901). Other devices may function but do not necessarily present the graphical output accurately. A list of additional user-defined, devices supported by the post-processor can be found in the Grafpak's GKS drivers' documentation.

Non-interactive output is normally written to a file and then printed after the post-processor session is complete. The filename is requested after a non-interactive device type is entered. If the filename does not have an extension, then an extension is provided by the post-processor. The extensions provided are dependent upon the device type and are listed below.

Type	Ext.	Type	Ext.	Type	Ext.
10300	cgm	1325	hp0	1900	ps0
1103	hp6	1328	hp1	1901	ps1

UNITS CONVERSION

requests the name of a file containing the list of variable names and units conversion coefficients.

Enter name of unit conversion file [CONVUNIT.DAT]:

If a file with the name CONVUNIT.DAT exists, depress the <RETURN> key. Aside from the FORCE2 variable names in the file CONVUNIT.DAT, two other names are defined:

GEOM - geometry units used to define X1, X2, X3 locations*
STRM - stream line

The user must respond to the command:

Update unit conversion data (Y or N) [N]?:

This command will change the units conversion coefficients, the unit name or may add a new unit name to the conversion file. This may be done while running the post processor.

SESSION FILE

A session file is created each time the user interactively runs the post-processor and contains a list of all user entries. This file can be used to rerun a previously-defined set of plots by assigning it the logical name PP_SESSION and rerunning the post-processor. It should be noted that if a session file exists with the name PP_ SESSION, the post-processor will use it. To develop hard copies of the plots, the session file is modified by i) changing the GKS output device (from a graphics terminal to a hardcopy plotting device) on the first line of the file and ii) adding the output file name after the GKS device selection. Session files that were developed during interactive operation of the post-processor (GKS output device 2501, a 4014 Tektronix terminal) and subsequently

---

*When this prompt is answered yes with a Y, the user is prompted for the required information.

modified to produce a file for printing on the postscript, landscape printer (GKS output device 1901, PostScript, Landscape) are shown below. The post-processor was run with the modified session file to create the postscript file, p33x14.ps1.

INITIAL SESSION FILE

```
2501


p33x14.ppf
Standard Problem
p33x14.ppt


Plot Title #1
Plot Title #2
g



.2

e
```

MODIFIED SESSION FILE

```
1901
p33x14


p33x14.ppf
Standard Problem
p33x14.ppt


Plot Title #1
Plot Title #2
g



.2

e
```

FORCE2 POST-PROCESSOR FILES

Two files containing the hydrodynamic predictions for the post-processor are written by FORCE2 (written to Fortran units 69 and 70, see Table 4-1). Up to three sets of files can be accessed by the program simultaneously. The program prompts the user for the name of the file containing the field variables.

Enter filename of FORCE2 data [PPF]:

The program also prompts the user for the title for the data file and prints it on the graphics output.

Enter data file identification (16 chars.):

The program also prompts the user for the name of the file containing the time data (variable vs time).

Enter FORCE2 time data filename [<Enter> = none]:

If a time data file has not been created, enter a carriage return and execution will continue.

BOUNDARY DATA

The boundaries and other key features of the geometry can be defined in a geometry data file. It contains the selected views and should be named or assigned the logical name, PP_BOUND1. Development of this file is described in Section 7.3.3.

LOGO AND TITLES

Each plot contains a 2-line title and a logo placed in the lower right corner (three lines of text consisting of the analysis name, company or division name and current date). The user is prompted to enter the company name (25 characters maximum) and the two title lines as follows:

```
Enter Division Name [BABCOCK & WILCOX] :
Enter plot title #1 (40 chars) :
                 #2 (40 chars) :
```

3.4.4 Main Menu

The main plotting menu allows selection of the field data to be plotted and the desired plot type.

Main Plotting Menu

```
=============================
E  End (Terminate) Program
P  Profiles
N  New Data Set
G  Geometry
C  Contours
V  Velocity Vectors
T  Variable vs Time
S  Streamlines
U  Update Units
Enter Selection:
```

The streamline option will appear only for 2-D problems.

3.4.4.1 General Input. The view, case, range and scale selections for plots are discussed first. A description for each plot type (profile, geometry, contours, etc.) listed in the main plotting menu follows.

VIEW

applies to all but axisymmetric plots. The user must select the view containing the information to be plotted. A view is defined by its axis labels (i.e., X1-X2) as shown below.

```
Select view
   1.  View of X2-X3 Plane
   2.  View of X1-X3 Plane
   3.  View of X1-X2 Plane
Enter selection [3] :
```

CASE

selects one or more field variable data files to plot. This menu is only displayed when more than one restart file is read.

```
Restart Cases
   1.  CASE 1
   2.  CASE 2
Enter Selections (1, 2) [1, 2] :
```

RANGE

The range input specifies the plane normal to the view direction and the range of data. For each restart case that is entered, the user chooses the plane number as shown below.

```
View of X1-X2 Plane
Select Plane Locations on X3 Axis (20 max)
   X3 Axis Nodes   5, Length   0.00 to  0.10
Enter Node Number(s) :
```

The user is also prompted for the range of data that applies to each of the axes in the selected view.

```
Enter grid (node) range
   Select node range for X1 AXIS (1, -12) [ 1,-12]:
   Select node range for X2 AXIS (1,  -6) [ 1, -6]:
```

SCALE

is required for all plots except profile plots. The program will prompt the user as follows:

```
Geometry data for       X1 AXIS   and  X2 AXIS
   Nodes          =         12              6
   Length (min)   =       0.00           0.00
          (max)   =       1.00           0.10
   Default scale  =       8.75          62.50
Enter Scale Factor (inches of paper/unit of length)
   X1 AXIS [   8.75]:
   X2 AXIS [  62.50]:
```

The actual size of the plot depends on the scale factors and geometry entered. Default values are for a plot on an 8-1/2 x 11 paper or on the graphics screen.

3.4.4.2 Profiles. Profiles are selected with the response, P, and permit creation of x-y plots for selected FORCE2 results. An example profile plot is shown in Figure 7-7. The options for profile plots are selected from the options menu below. Plot legends and/or symbols can be eliminated. One or more curves can be drawn on a plot. Additional curves on a plot can represent different variables, cases, planes, or elevations,[1] and multiple plots can be made.

The last option creates a file containing profile plot data.

Profile plot options
1. **Plot legend [YES]**
2. **Plot symbols [YES]**
3. **Multiple plots [NONE]**
4. **User size [NO]**
5. **File output [NO]**

Enter option or <RETURN> to end :

ABSCISSA SELECTIONS

enable the user to choose the abscissa, x, for the selected view.

Enter Abscissa for X axis
1. **X1 AXIS**
2. **X2 AXIS**

Select Axis [1] :

The abscissa is selected prior to the grid (node) range solution. The grid (node) range selection defines:

[1]Note: In the selected view, elevation indicates entries of the node location along the ordinate axis.

1) the range of the abscissa and 2) the location(s) along the ordinate for which dependent variables will be plotted.

Enter grid (node) range
   Select node range for X1 AXIS (1, -12) [1, -12]
   Select node range for X2 AXIS (1, -6) [1, -6]

PROFILE VARIABLE SELECTION

names the dependent variable data to be plotted. A list of the variable names can be displayed by typing a <RETURN>. Variable name entries must be in capital letters.

Enter Data Names, (10 Max) <CR> for list :

PROFILE PLOT LEGEND

allows the user to determine if the profile legend is to be displayed on the plot.

PROFILE PLOT SYMBOLS

can be turned-on or -off with this option.

MULTIPLE PROFILE PLOTS

are created from the multiple plot menu shown below.

Multiple Plot Menu
1. Case
2. Plane
3. Elevation
4. Variable
5. None

Enter option [3] :

The profile option produces a single plot containing a line for each case, plane, elevation and variable selected. Multiple plots can be made for each entry of the selected item: case, plane, elevation or variable. For example, if multiple data names are entered and option 4 is selected, separate plots will be created for each variable input: the same X and Y scale range is used for all plots.

PROFILE SIZE SELECTION

permits control of the length of the X and Y axis, and the Y axis range and title. When option 4 ,USER SIZE, is selected, the user is asked to enter the lengths, ranges and a title as shown below.

```
User input axis specifications
   Enter X-Axis Length [  7.50] :
   Enter Y-Axis Length [  5.00] :
   Enter Y-Axis Minimum [  0.000E+00] :
   Enter Y-Axis Maximum [    1.09     ] :
   Enter Y-Axis Title [METERS/SEC]:
```

Default values have been set for a plot on 8-1/2 x 11 paper or the graphics screen.

PROFILE DATA FILE OUTPUT writes a data file containing the values used to create the profile data. Section 7.3.2 describes a unit assignment that is required to obtain profile plots.

Once post-processing of the profile information is complete, the profile plot is displayed on the terminal or written to the ZETA plot file. Entering a <RETURN> will display the main menu.

<u>3.4.4.3 Geometry</u>. Geometry is selected with the response, G, and produces a plot of the modeled geometry created by the FORCE2 program. An example geometry plot is illustrated in Figure 7-9. Control volumes, blocked cells and faces and the geometric boundary are plotted. The geometry options menu allows the user to determine the appearance of the plot.

```
Geometry options
   1.  Boundary          [YES]
   2.  Axis Labels       [YES]
   3.  Plot Labels       [YES]
   4.  Grid lines        [YES]
   5.  Node centers      [YES]
   6.  Blocked cells     [NO ]
   7.  Blocked faces     [NO ]
   8.  Invert X axis     [NO ]
Enter option or <RETURN> to end :
```

BOUNDARY

controls plotting the geometric boundary. The file containing the boundary description was created by the pre-processor and is read by the post-processor. The file contains boundaries for only one view.

AXIS LABELS

permits plotting the axis labels.

PLOT LABELS

defines the title text on the plot.

GRID LINES

controls plotting the grid lines on the plot.

NODE CENTER

permits plotting the node center markers with a plus sign at the center of each control volume.

BLOCKED CELLS

controls the marking of blocked cells and other unused control volumes defined in FORCE.

BLOCKED FACES

defines the marking of the individual control volume faces which are defined as blocked in the FORCE program.

INVERT AXIS

specifies the view direction of the plot. A normal view of an X-Y plot displays the data viewed along the Z axis toward increasing Z values. A reverse view of the same plot displays the data viewed toward decreasing Z values.

Once the the geometry information has been chosen the geometry plot is displayed on the terminal or written to the ZETA plot file. Depressing the <RETURN> will display the main menu.

<u>3.4.4.4 Contour Plots</u>. Contours are selected with the response, C, and produce a contour plot of the named data calculated by FORCE2 (see Figure 7-11). User input consists of selecting the variable(s) to plot, the appearance of the plot and the contour line spacing.

```
Contour options
1.  Boundary                  [YES]
2.  Axis labels               [YES]
3.  Plot labels               [YES]
4.  Contour labels            [NO ]
5.  Contour lines    :
          Straight or Curved  [STR]
6.  Contour spacing  :
          Value or Number     [VAL]
7.  Invert X axis             [NO ]
Enter option or <RETURN> to end :
```

BOUNDARY

controls plotting the geometric boundary. The file containing the boundary description was created by the pre-processor and is read by the post-processor. The file contains boundaries for only one view.

AXIS LABELS

permits plotting the axis labels.

PLOT LABELS

defines the title text on the plot.

CONTOUR LABELS

controls markers that are plotted to identify selected contours. When contours markers are selected, a YES response, additional data is required to define how these markers will be placed on the graph.

AXIS If the response for CONTOUR LABELS is Yes, the user must identify the direction to mark the contours. For example, if the default is chosen

```
Enter Axis for Contour Marking
   1. X1 AXIS
   2. X2 AXIS
Select Axis [1] :
```

then markers will be placed on contour lines crossing the node centerline parallel to the X1 axis.

RANGE When the response for CONTOUR LABELS is Yes, the user is prompted for the range of node centerlines parallel to the selected axis. For example:

Enter grid (node) range for contour markers
Select node range for X1 AXIS (1, -34) [1, -34]:

places markers along each grid line from 1 to 34.

NUMBER When the response for CONTOUR LABELS is Yes, the user must select the number of contour lines between marked contour lines. For example the default

Enter the number of lines between marked contour
lines [0]:

specifies that every contour line crossing the node centerline be marked, zero lines skipped.

CONTOUR LINES

plots all contours within a control volume using values known at the corners. Either straight lines or interpolation functions between the faces of the control volume are used to plot curves. Experience indicates that straight line segments usually produce better plots.

CONTOUR SPACING

specifies the method of selecting contour spacing: a value for contour spacing or the number of lines on the plot. The program also prompts for the minimum and the maximum contour value to plot.

VALUE defines the contour spacing. Contours will correspond to multiples of this value.

Contour Spacing
Contour data range - .4 to 1.0
Enter contour spacing [.1] :

The contour data range displayed is the total range of contour variable data for all cases and ranges specified.

NUMBER selects the number of contour lines on the plot.

Enter the number of lines per plot [10] : 6

MINIMUM MAXIMUM selects the allowable range of data over which contour lines will be plotted. For example,

Enter minimum contour value [.4] :
Enter maximum contour value [1.0] : .9

INVERT AXIS

specifies the view direction of the plot. A normal view of an X-Y plot displays the data viewed along the Z axis toward increasing Z values. A reverse view of the same plot displays the data viewed toward decreasing Z values.

VARIABLE DATA

prompts the user to enter the name of the variable data to be plotted. A list of the variable names is displayed by typing a <RETURN>. If more than one variable name is entered, a separate contour plot is created for each entry.

Enter Data Name (<CR> for list): VFRG

Note - The variable names must be in capital letters.

Once the variable data has been chosen the contour plot will be displayed on the terminal or written to the plot file. Entering a <RETURN> will display the main menu.

3.4.4.5 Vector Plots. Vector plots are selected with the response, V, and produce a two-dimensional plot for the direction and magnitude of the velocity or mass flux. The components of velocity or mass flux, stored at faces, are interpolated to the node center. Figure 7-12 shows an example of a vector plot.

```
Vector options
   1.  Boundary                   [YES]
   2.  Axis labels                [YES]
   3.  Plot labels                [YES]
   4.  Vector label               [YES]
   5.  Vector    :
          Velocity or Mass Flux   [VCT]
   6.  Invert X axis              [NO]
   7.  Gas or solids              [GAS]
Enter option or <RETURN> to end :
```

BOUNDARY

controls plotting the geometric boundary. The file containing the boundary description was created by the pre-processor and is read by the post-processor. The file contains boundaries for only one view.

AXIS LABELS

permits plotting the axis labels.

PLOT LABELS

defines the title text on the plot.

VECTOR LABEL

controls plotting a unit vector, a one-inch line, drawn below the plot label showing the scale factor used to plot the vectors.

VECTOR - VELOCITY OR MASS FLUX

allows the selection of which vectors are plotted: velocity or mass flux.

INVERT AXIS

specifies the view direction of the plot. A normal view of an X-Y plot displays the data viewed along the Z axis toward increasing Z values. A reverse view of the same plot displays the data viewed toward decreasing Z values.

GAS OR SOLIDS

specifies the phase.

SCALE FACTOR

requests a factor to scale the vectors. The default scale factor is for a 1/2-inch vector. The maximum vector length of any vector is also requested.

```
Vector Scale Factor
     Maximum data range - 1.08E+00 METERS/SEC
  Enter scale factor (units/inch)  [  2.20  ] :
  Vector Maximum Length
  Enter length (inches)  [  0.500] :
```

Once the vector scale factor has been chosen the vector plot is displayed on the terminal or written to the plot file. Entering a <RETURN> will display the main menu.

3.4.4.6 Variable vs Time Plots. Plots of selected variables versus time are selected with the response T. The user selects the variable(s) to be plotted and inputs the appearance of the plot.

Minimum and maximum times are entered after the prompts:

```
Enter Minimum Time [    ] :
Enter Maximum Time [    ] :
```

The default values are the minimum and maximum value in the time data file.

VARIABLE SELECTION

The variable(s) to be plotted are selected from the table of variables that was input to FORCE2 in the CONTROL Paragraph using the STORE statement (see Section 4.6.4.2 of the FORCE2 input description). This table is displayed as

```
Variable Description for [Title of Run]
   1   Variable #1   Location
   2   Variable #2   Location
   3   Variable #3   Location
   .
   .
   N   Variable #N   Location

Enter Number(s) for Variable (8 max) :
```

The variables and their locations are displayed in the format that they were input to FORCE2.

The appearance of the plots is controlled by the Plot Options Menu.

```
Plot Options
   1. Plot legend      [Yes ]
   2. Plot symbols     [Yes ]
   3. Multiple plots   [NONE]
   4. User size        [NO  ]
   5. File output      [NO  ]
Enter option or <RETURN> to end :
```

These options are identical to those for the Profile plots (Section 3.4.4.2) and produce the same effects.

3.4.4.7 Streamlines. Streamlines are selected with the response, S, and produce streamline plots of the gas or solids flow field calculated by FORCE2. User input consists of selecting the phase, the appearance of the plot, and the streamline spacing, the magnitude of the flow rate between streamlines. The magnitude of the flow rate between streamlines is constant. The streamline plot options are selected from the following menu:

```
Streamline options
   1. Boundary                  [YES]
   2. Axis labels               [YES]
   3. Plot labels               [YES]
   4. Streamline lines -
          Straight or Curved    [STR]
   5. Streamline spacing -
          Value or Number       [VAL]
   6. Gas or solid flow         [GAS]
Enter option or <RETURN> to end :
```

Pressing <RETURN> will exit the menu.

BOUNDARY

This option determines if the geometric boundary will appear on the plot. A YES in the brackets is the default and indicates that boundary data will appear on the plot. A file containing the boundary description is created by the user and is read by the post-processor. When geometry is not defined, then the default in brackets will display [N/A] and the boundary option is not applicable.

AXIS LABELS

This option determines if axis labels will appear on the plot. A YES in the brackets is the default and indicates that axis labels will appear on the plot.

PLOT LABELS

This option determines if the title text will appear on the plot. A YES in the brackets is the default and indicates that plot labels will appear on the plot.

STREAMLINE LINES

Selecting this option plots streamlines as contour lines of flow rate within a control volume rectangle using values defined at the corners of the rectangle. Bilinear interpolation is used to determine the path of a contour within this rectangle. The streamline plotting will either generate straight lines between the faces of this rectangle or plot curves resulting from the interpolation functions. Experience indicates that straight-line segments usually produce better plots.

Options are straight or curved lines. The default [STR] is straight lines.

STREAMLINE SPACING

This option specifies the method of selecting streamline spacing: a value for streamline spacing or the number of lines on the plot. The program also prompts for the minimum and maximum streamline values to plot.

Value defines the streamline spacing. The magnitude of flow rate between lines will be equal to value, starting from the minimum streamline value.

```
Streamline Spacing
   Streamline data range -   -3.49E-08 to 9.69E-01
Enter streamline spacing [ 0.100   ] :
```

The default value is approximately 10 streamlines per plot. The streamline data range displayed is the total range of flow rate data for all cases and ranges specified. The default streamline spacing is based on the minimum (D_{MIN}) and maximum (D_{MAX}) data for the selected variable as follows:

$$Spacing - (D_{MAX} - D_{MIN}) / (Number - 1)$$

The default spacing is then rounded before being displayed.

Number selects the number of streamlines on the plot.

Enter the number of lines per plot [10] :

The default is the last value entered or a value of 10.

GAS OR SOLID FLOW

The phase for which streamlines will be plotted is selected with this option.

3.4.4.8 New Data Set. A new set of field data is selected with this option. The user enters the response, N. During FORCE2 execution, predictions can periodically be written to the field variable data file. The frequency at which data is written is controlled by input to FORCE2. The data set is selected by entering the desired problem time. For steady execution, problem "time" is defined by dividing the steady iteration number by 1000.

The case number and run title are displayed by:

Data for CASE : Case # --- Run Title

The user then enters the problem time at which new data is selected according to:

Enter time of data to plot t1 to t2 [t3] :

where t1 is the initial problem time of all the field data sets, t2 the final time, and t3 the time of the current data set.

3.4.4.9 Update Units. This option allows the user to modify the unit conversion file from the main plotting menu after the field data and time data files have been read. The user enters the response, U, for this option. Prompts will then appear for modifying the unit conversion file.

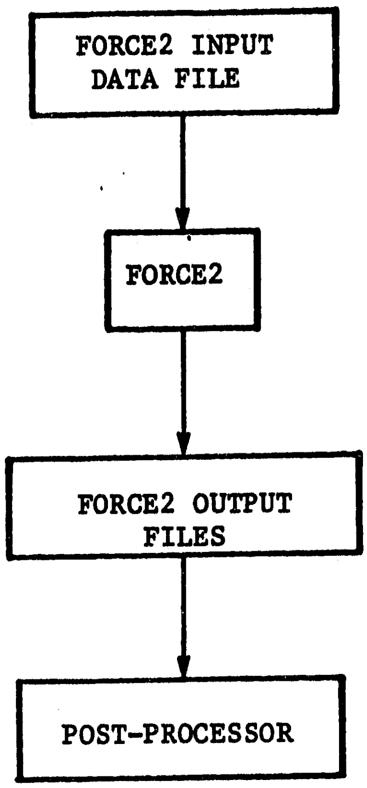


Figure 3-1. Key elements in developing FORCE2 predictions.

4.0 FORCE2 INPUT DESCRIPTION

4.1 OVERVIEW

Files that are read or written by FORCE2 are listed in Table 4-1. The input file on Fortran Logical Unit 5 and the runtime file on Unit 11 are required to execute the program. The required information in both files is described in this section. FORCE2 input and output for a sample problem are given in Section 6.0.

The FORCE2 input file on Fortran Unit 5 (Table 4-1) contains a description of the problem geometry, material properties, operating conditions and solution options. The input is designed to be easy to use and remember. It is free form and uses KEYWORDS to identify data values instead of the usual FORTRAN input consisting of fixed field formats. The input file is organized in sections called PARAGRAPHS. Paragraphs which describe general features of the problem must always be supplied but paragraphs describing a particular process are only required if that process is part of the problem. Default values are also used extensively to reduce input.

4.1.1 Paragraphs

Each PARAGRAPH consists of:

- o a paragraph HEADING
- o STATEMENTs specifying input for the paragraph
- o a BLANK LINE indicating the end of the paragraph

Paragraphs must be arranged in a fixed order as shown in Table 4-2, which also identifies optional paragraphs.

Table 4-1

FORCE2 INPUT AND OUTPUT FILES

FORTRAN Logical Unit	Input/ Output	File Type	Description
1	I	Binary	Restart information. This is the file written to Unit #2 at the end of the previous run.
2	O	Binary	Restart information. This file is written at the end of a run.
5	I	ASCII	FORCE2 input file. It contains description of problem, operating conditions, etc.
6	O	ASCII	FORCE2 output file for printing.
11	I&O	ASCII	FORCE2 iteration control file. It contains parameters that control the execution of the code.
20	O	ASCII	A log file that contains iteration information.
69	O	Binary	Graphical post-processor file that contains all the field variables.
70	O	Binary	Graphical post-processor file that contains the selected field variable versus time data.
71	O	Binary	Erosion post-processor file. It includes velocity and void fraction predictions that are subsequently input to the erosion model, EROSION.

Table 4-2

FORCE2 INPUT PARAGRAPHS

Paragraph Heading	Type	Contents
1. (none)	optional	Restart control
2. GEOMETRY	required	Flow field geometry, finite difference grid and print control for geometry
3. PROPERTIES	required	Gas phase, viscosity, density and reference pressure
4. FLOW	required	Flow field solution method, flow devices, gravity input
5. CONTROL	required	Number of iterations, relaxation parameters and field variable print control
6. PERMEABILITIES	optional	Surface permeabilities, volume porosities
7. INITIALIZATION	required	Initialize boundary and interior values for any field variable
8. FLOFLAGS	required	Wall boundary conditions

4.1.2 Statements

The STATEMENTS which make up the body of a paragraph can usually appear in any order. The exceptions are noted in the paragraph descriptions later in this manual. A statement consists of a sequence of one or more ELEMENTS. Elements are separated by spaces, commas or tabs at the user's discretion. Each element is either a KEYWORD or NUMBER. The first element in a statement is always a keyword.

Example:

```
CARTESIAN 20,10,5
```

This statement consists of four elements, namely:

CARTESIAN	keyword
20	number
10	number
5	number

Keywords must be upper case. Spaces, commas and tabs can be used interchangeably as element separators. Both of the following are equivalent to the previous example.

```
CARTESIAN, 20,10,5
CARTESIAN, 20  10  5
```

The following statement is an error because of improper separation of elements.

```
CARTESIAN 20,,10,5
```

In the last case, the two successive commas are interpreted as an element with an unspecified value (a null element).

Statements may be written on more than one line; however, elements must be complete on a single line. A comma must be used as the separator following the last element on a line to indicate that the statement is incomplete and continued on the next line. Line length is limited to 80 characters.

Example:

```
CARTESIAN,
20,10,5
```

A statement consists of all lines up to and including the first line which does not end in a comma.

Spaces and tabs can be used to make the input more readable. Any number of spaces or tabs can be used as a delimiter to space the elements of a statement. Spaces or tabs can be used before the first element on a line for indentation.

Comments are initiated by a dollar sign ($). If the $ is the first character on a line, it must be in column 1. Otherwise, it can appear anywhere on a line. All text following the $ is treated as a comment.

4.1.3 Notation

The input statements used with FORCE2 are described in the remaining sections of this appendix. The complete form of each statement is given using the following notation.

- o UPPER CASE letters denote a KEYWORD. Keywords can be abbreviated to the first four characters.

- o LOWER CASE letters denote an element to be supplied by the USER as described.

- o Brackets [] surrounding an item denote that the item is optional.

- o A column list of elements, one above the other, indicates that one of the elements in the list is to be selected.

- A lower case n used as a suffix for a KEYWORD indicates a coordinate direction; n = 1, n = 2, or n = 3.

4.1.4 Field Variables

The field variables in the flow model are listed in Table 4-3. The field variable names are used on input to control printing, relaxation and initialization. It should be noted that either the CGS (Centimeter-Gram-Second) or MKS (Meter-Kilogram-Second) system of units can be used.

Table 4-3

FORCE2 FIELD VARIABLES

Field Variable	Description
RHOG	Gas density
RHOS	Solids density
VISCG	Gas viscosity
VISCS	Solids viscosity
VFRG	Gas void fraction
VFRS	Solids void fraction
U1G	U1 gas velocity component
U2G	U2 gas velocity component
U3G	U3 gas velocity component
U1S	U1 solids velocity component
U2S	U2 solids velocity component
U3S	U3 solids velocity component
P	Local static pressure
FGAS	Gas mass source at feed ports
FSOL	Solids mass source at feed ports
GSOU	Gas mass source to simulate gas production due to combustion

4.2 RESTART PARAGRAPH

Since FORCE2 employs an iterative solution procedure, it is often desirable to solve a problem in several steps called runs. Each run after the initial run begins where the previous run ended. FORCE2 accomplishes this by creating an optional restart file containing all of the information required to continue the solution. Creation of a restart file is controlled by the restart control paragraph. A restart run requires both an input file and a restart file. As indicated in Table 4-1, the restart file created by FORCE2 is assigned to FORTRAN Logical Unit 2. When it is subsequently used as an input restart file in a succeeding run, it is assigned to Unit 1. In addition to RESTART control, a title for the run may be included in the RESTART paragraph. The run title is an optional input. The title will appear on printed output and will be included with all the post-processor data files (see Table 4-1).

The RESTART control paragraph has no header and may consist of two statements. It is always the first input paragraph. The RESTART keyword is optional for the initial run but required for any subsequent restart run. The title keyword is optional for all runs.

```
            [ RESTART ]        SAVE
                               NOSAVE

      SAVE        Specifies whether or not a restart file is created
      NOSAVE      by this run
```

The optional run title consists of 40 characters and is entered as follows.

```
[TITLE]          Title of run
```

4.2.1 Initial Run

The RESTART keyword must not be included in an initial run. The SAVE keyword causes a restart file to be created. If the RESTART paragraph is omitted on the initial run, a restart file will not be created.

4.2.2 Restart Run

The RESTART paragraph is always required for a restart run. The RESTART statement causes the program to read a restart file and to continue the previous solution. If SAVE is specified a new restart file will be created.

The restart input file can contain additional paragraphs which modify conditions for a restarted run. This allows the user to continue a run or to run several similar problems without starting over from scratch. After the first problem, subsequent problems can be run by restarting from the previous solution and specifying the input necessary to modify the original problem to the new problem. However, <u>the GEOMETRY paragraph must not be changed for a restart run</u>.

A restart run can also be used to print variables which were not printed on the original run. The CONTROL paragraph contains commands for controlling printing. A restart with zero iterations in the CONTROL paragraph can be used for printing additional field variables.

4.3 GEOMETRY PARAGRAPH

FORCE2 can model a three-dimensional Cartesian coordinate system.

The conservation equations for flow are solved by a finite difference technique using a control volume approach. The solution domain is divided into a number of rectangular control volumes (Figure 4-1). All variables except velocity are computed and stored at discrete locations called nodes. A node is associated with each control volume and located at the center of the control volume. Additional nodes are included on the boundaries at the center of each control volume face and at the corners of the solution domain. Nodes are shown as dots in Figure 4-1.

Velocity components are computed and stored at control volume faces because a staggered grid system is used in the solution of the flow equations. Velocity nodes are located at the center of the control volume faces normal to the associated velocity direction. The arrangement for a 2-D geometry is depicted in Figure 4-2. The velocity node numbering convention with respect to the control volume should be noted: velocities on the lower faces of the control volume are assigned the control volume number in the particular coordinate direction. This convention should be kept in mind when initializing field variables.

The geometry paragraph describes the discretization of the problem geometry into control volumes. Control volumes are specified by the positions of the control volume faces.

4.3.1 Solution Domain

The CARTESIAN statement specifies the number of nodes in the X1, X2 and X3 directions. It must be the first statement in the geometry paragraph.

CARTESIAN   n1 n2 n3

- n1 Number of nodes in the X1 direction, including boundary nodes
- n2 Number of nodes in the X2 direction, including boundary nodes
- n3 Number of nodes in the X3 direction, including boundary nodes

---

4.3.2 Control Volume Face Positions

Positions of control volume faces are specified in separate statements for each coordinate direction. Either absolute face positions or the relative spacing between faces can be specified.

Statements for specifying face positions are the same for each direction. In the remainder of this subsection, a lower case n will be used to represent the coordinate direction.

Actual Face Positions

```
Xn          P1,...,Pi
```

Xn	Identifies actual face position specification for direction n.
P1,...,Pi	Coordinate values for face positions. The number of positions specified is one less than the number of nodes specified in the CARTESIAN statement.

The face positions can be specified in any order and are sorted into ascending order by the program.

Face Spacing

The distance between coordinate faces can be specified by this command. If only one distance is specified, it is assumed to hold for all cells in that direction.

```
DXn          s1 [,si] ...

      DXn          Identifies specification statement for spacing of cell
                   faces in direction n.  The maximum number of spacings is 2
                   less than the number of nodes specified in the CARTESIAN
                   statement.

      s1 [,si] ... Distances between faces.  If only one distance is
                   specified, it is assumed to hold for all cells.  If the
                   number of specified cell spacings is less than the number
                   of cells, the last spacing will be used for the remaining
                   cells.
```

It is not necessary to specify all cell spacings if some are identical. In this case, the spacing for the first cell is specified and the following cells are specified as null values.

Example:

For 5 cells with a spacing of 0.5 meters followed by a cell with a spacing of 0.7 meters, the following could be coded.

```
DX1  0.5, , , , ,0.7
```

4.3.3 Boundary Model Input

Input to the boundary model consists of a sequence of statements specifying blockages and boundary conditions. There are two distinct types of blockages, solid blockages and planar blockages. Solid blockages obstruct a volume of the geometry. Planar blockages obstruct a surface and have no volume. Statements for describing solid blockages, planar blockages and boundary conditions are described in this section.

The input statements for the general boundary model use the standard FORCE free format input. A statement consists of a sequence of one or more elements. Each element is either a keyword or a number. Elements may be separated by spaces, commas or tabs at the user's discretion. The first element of a statement is always a keyword.

Solid Blockage Statement

A solid blockage obstructs a volume of the problem geometry. Its faces must be perpendicular to coordinate directions. For Cartesian coordinates a solid blockage is a rectangle in 2-D geometries and a rectangular solid in 3-D geometries. More complex blockages can be described by specifying several solid blockages adjacent to each other.

The solid blockage statement specifies the extent of the blockage in each coordinate direction. Its general format is shown below.

```
BLOCK CELLS X1-min, X1-max, X2-min, X2-max [,X3-min, X3-max]
```

where

BLOCK	identifies a blockage statement
CELLS	indicates that this is a solid blockage specification
Xn-min	is the position of the lower face of the solid in direction Xn
Xn-max	is the position of the upper face on the solid in direction Xn

---

Coordinate limits (Xn-min, Xn-max) may be omitted if they correspond to the physical boundaries of the geometry and the limits following them are also omitted.

Blockages may not be specified on a restart run.

Examples:

```
BLOCK CELLS 1.3, 2.5, 0.0, 0.5
```

This describes a solid blockage with faces at X1=1.3 and 2.5 and X2=0.0 and 0.5. For a 2-D geometry this would be a rectangle. For a 3-D geometry it would be a rectangular solid that extends to the physical boundaries in the X3 direction.

Planar Blockage Statement

A planar blockage obstructs a surface of the problem geometry and has no volume. It must be perpendicular to a coordinate direction. Its edges must be parallel to the other coordinate directions. In Cartesian coordinates a planar blockage is a line segment in 2-D geometries and a rectangle in 3-D geometries. Complex planar blockages can be described by specifying several planar blockages adjacent to each other.

The planar blockage statement specifies the direction perpendicular to the blockage, its position in that direction and the extent of the blockage in the other directions. Its general form is shown below.

BLOCK Xn position Xi-min, Xi-max [,Xj-min,Xj-max]

where:

BLOCK	identifies this as a blockage statement
Xn	is the direction perpendicular to the blockage
position	is the position of the blockage in direction Xn
Xi-min	is the position of the lower edge of the blockage in direction Xi (for n=1, i=2 otherwise i=1)
Xi-max	is the position of the upper edge of the blockage in direction Xi (for n=1, i=2 otherwise i=1)
Xj-min	is the position of the lower edge of the blockage in direction Xj (for n=3, j=2 otherwise j=3)
Xj-max	is the position of the upper edge of the blockage in direction Xj (for n=3, j=2 otherwise j=3)

Coordinate limits (Xk-min, Xk-max) may be omitted if they correspond to the physical boundaries of the geometry and the limits following them are also omitted.

Blockages may not be specified on restart runs.

Examples:

BLOCK X2   0.0

This statement describes a planar blockage perpendicular to direction X2 at X2=0.0 and bounded by the physical boundaries in other directions.

BLOCK X1   0.5   1.0, 2.5, 0.0, 0.5

This statement describes a planar blockage perpendicular to direction X1 at X1=0.5 with edges at X2=1.0 and 2.5 and X3=0.0 and 0.5.

Finally, block cells <u>must be</u> specified in the FLOFLAG paragraph.

4.3.4 <u>Physical Boundary Conditions</u>

The specification of physical boundary conditions can depend on the mode of execution (STEADY or TRANSIENT specified in the FLOW paragraph). The boundary conditions used at external boundaries of the problem in the STEADY mode are as follows:

- o Blocked boundaries are treated as blockages and boundary conditions are determined by the blockage model.

- o Unblocked X1 boundaries.

  If flow is into the problem, Dirichlet boundary conditions are used for all variables. If flow is out of the problem, zero gradient boundary conditions are used for all variables except U1 which is determined by continuity. The direction of flow is determined by the initial velocities.

- o Unblocked X2 or X3 boundaries

  Either Dirichlet or symmetry boundary conditions are used for all variables depending on boundary condition (described below).

As indicated above, in the STEADY mode, flow into the domain can be parallel to any coordinate and flow out of the domain must be parallel to the X1-direction. The boundary conditions used at external boundaries of the problem in the TRANSIENT mode are as follows:

- o Blocked boundaries are treated as blockages and boundary conditions are determined by the blockage model.

- o Unblocked boundaries with flow into the domain: Dirichlet boundary conditions are used for all variables.

- o Unblocked boundaries with flow out of the domain: Constant pressure or continuitive outflow conditions are applied. For constant pressure, velocities are calculated. For continuitive outflow, zero gradients in velocity, pressure and void are applied.

- o Unblocked symmetry boundaries: Symmetry boundary conditions are applied. Symmetry results in free-slip conditions in both phases at the boundary.

For both modes, the following applies.

- o Inflow and outflow boundary conditions should be defined in the FLOFLAG paragraph.

- o Free-slip, partial slip or no-slip conditions for flow parallel to blocked surfaces should be defined in the FLOFLAG paragraph.

The physical boundary condition statement specifies the boundary conditions to be used on a single X2 or X3 boundary. The statement specifies the boundary and the boundary conditions to be used. Four statements must be used to specify all of the boundary conditions.

```
BOUNDARY  X2 LOWER    DIRICHLET
          X2 UPPER    SYMMETRY
         X3 LOWER
         X3 UPPER
```

where:

X2 LOWER	specifies the X2 boundary with smallest X2 coordinate
X2 UPPER	specifies the X2 boundary with largest X2 coordinate
X3 LOWER	specifies the X3 lower boundary
X3 UPPER	specifies the X3 upper boundary
DIRICHLET	indicates the Dirichlet (known value) boundary conditions are used
SYMMETRY	indicates that this is a symmetry boundary

If no boundary conditions are specified for any X2 or X3 boundary, Dirichlet boundary conditions are used.

Boundary conditions may not be specified on restart runs.

Examples:

BOUNDARY X2 LOWER SYMMETRY

This statement describes the X2 boundary corresponding to the smallest X2 coordinate as a symmetry boundary. This statement would be used to specify the boundary conditions for the centerline of a cylindrical geometry.

BOUNDARY X2 UPPER DIRICHLET

This statement specifies that the X2 boundary corresponding to the largest X2 coordinate uses Dirichlet boundary conditions.

4.3.5 Geometric Information Output Control

As an option, all of the geometric information calculated by the program can be printed. Normally, only the basic information provided as input is printed.

```
PRINT 1
      2
      ALL
```

PRINT	Identifies the geometry output control statement.
1	Limited geometry printout without blockage print.
2	Limited geometry printout with blockage print.
ALL	All of the calculated geometric information is to be printed.

4.4 PROPERTIES PARAGRAPH

The PROPERTIES paragraph defines the viscosity and density of the gas and solids phases. Reference temperature and pressure, common to both phases, are also defined. These data are always used in the first iteration of the flow field solution.

For subsequent iterations, the density specified in the properties paragraph is used in the solution. The perfect gas law can be specified as an option for the properties paragraph.

The molecular viscosities specified in this paragraph are used for all applications. Viscosity as well as other properties can be a constant, a polynomial function up to fourth order, or a user-programmed function. The statements are:

```
DENSITY     GAS       a0                       [a1,...,a4]*
            SOLIDS    USER [FUNCTION]          n-1
                      FUNCTION                 n-1
                      IDEAL (with GAS only)

VISCOSITY   GAS       a0                       [a1,...,a4]
            SOLIDS    USER [FUNCTION]          n-1
                      FUNCTION                 n-1
```

a_i	coefficients of the polynomial (property = $a_0 + a_1T + a_2T^2 \ldots a_4T^4$)
n	number of the user-programmed property function
IDEAL	use the ideal gas law for density

The USER FUNCTION (or FUNCTION) commands execute user-programmed subroutines DENSO and VISCO. Currently there are no user-defined routines.

---

* For gas density, a polynomial is specified for the specific volume (i.e., $1/\rho = a_0 + a_1T \ldots a_nT^n$, $n \geq 1$). For a constant density, input only a_0 where $\rho = a_0$.

* For solids density, a polynomial specified according to $\rho = a_0 + a_1T \ldots a_nT^m$.

This feature permits special properties for fluids or solids to be added in the future.

If the ideal gas law is used for density, then the molecular weight of the gas must be defined.

MOLECULAR [WEIGHT] value

value   The molecular weight of the gas.

The gas constant, R, in the ideal gas law

$$\rho = P \cdot m/[R \cdot (T + T_{ref})]$$

is set with the statement

GAS [CONSTANT] value

value   The gas constant R (default R = 1.0).

The reference temperature, T_{ref}, used in the ideal gas law and the reference enthalpy are set with the REFERENCE statement.

REFERENCE TEMPERATURE value

value   The reference temperature

The temperature field may be initialized by using the temperature statement.

TEMPERATURE value

value The nominal temperature.

Reference densities for the gas and solid phases are used for normalizing solution parameters. Solution speed and convergence are improved by defining a reference (or nominal) density for each phase. Reference densities are set as follows.

GAS REFERENCE DENSITY value
SOLIDS

value The reference density

The pressure field solved by FORCE is actually a pressure deviation from the reference pressure. The actual pressure level for a particular cell is the sum of the local pressure deviation and the reference pressure. The reference pressure is a known pressure level within the problem geometry. Both the pressure value and location are specified with this statement.

PRESSURE value [n1, n2, n3]

[n1, n2, n3] the reference pressure location. Location 2,2,2 is assumed if not specified.

A compressible formulation for the viscous terms is an option in the transient mode. It is selected by defining a second viscosity coefficient for either or both phases with the following statement.

SECOND [VISCOSITY] [COEFFICIENT] GAS or SOLIDS C_1

C_1 second viscosity coefficient

The second viscosity coefficient is normally taken as 2/3 according to Stokes' hypothesis, i.e.:

$$\lambda + C_1 \cdot \mu = 0$$

where λ = coefficient of bulk viscosity

C_1 = second coefficient of viscosity

μ = molecular viscosity

If C_1 is specified as a negative number (the default is -1.0), an incompressible formulation is used.

4.5 FLOW PARAGRAPH

The solution method and special features of the flow field are specified here. Special features of the flow include gravity and distributed resistances due to obstructions.

4.5.1 Type of Simulation

FORCE2 can perform either a transient or steady simulation. The mode is selected with one of the following statements:

```
TRANSIENT
  or
STEADY
```

4.5.2 Solution Parameters for the Steady Simulation

In the transient mode, the implicit multifield technique, Rivard (1977), is used to predict the hydrodynamics. The solution is controlled by the timestep, iteration and mass convergence parameters specified in the Control Paragraph.

In the steady mode, a modified SIMPLER solution algorithm (Patankar 1980; Schnipke 1986) is used. This method is based on solving the momentum equations with a fixed pressure field and then correcting the velocities and pressures so that conservation of mass is satisfied. An iterative solution method is necessary. For each macro solution step, iterative solutions of the momentum equations, pressure correction equations, and continuity equations are performed. A matrix solution method is used. The number of macro steps is specified in the Control Paragraph.

The iterative solution of the pressure correction equations is controlled by the following:

SWEEP PRESSURE nswp

nswp	Number of pressure equation sweeps

At each solution sweep the following is performed:

i) Solve along lines in the X2-X3 plane at each X1 location, marching "out" and "back" along the X1 axis.

ii) Solve along lines in the X1-X3 plane at each X2 location, marching "out" and "back" along the X2 axis.

iii) Solve along lines in the X1-X2 plane at each X3 location, marching "out" and "back" along the X3 axis.

For a two dimensional problem only steps i) and ii) are performed with solutions along X2 and X1 lines, respectively.

The iterative solution of the momentum equations is controlled by solution region and sweeping parameters. The solution region is defined by SETn statements, where n is the coordinate direction. Within the defined region, iterative solutions will be performed. It should be noted that the program does not determine if all the regions cover the entire solution domain. The defaults for the momentum equation solution define the entire solution domain; however, specifying regions with the SETn statements overrides this. The form of the SETn statements is defined in the section 2.8 Initialization Paragraph.

The sweeping parameters for the momentum equation solution are stored in a table and each set then executed to perform the solution. A set of these parameters for the momentum equation solution is defined by the following:

```
SWEEP  nswm  Xn  [PLUS]   [ADD]
                 [MINUS]  [REPLACE]
                          [DELETE]
```

nswm	Number of sweeps
Xn	Sweep direction (Xn = X1, X2, or X3)
PLUS	Means sweep in the plus Xn direction; default: PLUS followed by MINUS on each sweep
MINUS	Means sweep in the minus Xn direction
ADD	Add the current parameters to sweeping control table
REPLACE	Delete all previous entries and then add current parameters
DELETE	Delete all previous entries

For example the statement

```
SWEEP 10  X1  PLUS
```

results in the following solution scheme for all momentum equations

- o 10 sweeps down the X1-axis
- o At each nodal location on the X1-axis, the momentum equations are solved along lines in the X2-X3 plane

The statements

```
Set 1  10, 20
 Set 2  20, 30
  Set 3  3, 6
      Sweep 10 X1
      Sweep 20 X2
      Sweep 30 X3
```

result in the following solution scheme for all momentum equations:

- The sweeping strategy will be applied to all nodes within the region defined by

  X1-Axis: From node #10 to node #20
  X2-Axis: From node #20 to node #30
  X3-Axis: From node #3 to node #6

- Ten sweeps down the X1 axis, solving the momentum equations along line sin the X2-X3 plane

- Twenty sweeps down the X2 axis, solving the momentum equations along lines in the X1-X3 plane

- Thirty sweeps down the X3 axis, solving the momentum equations along lines in the X1-X2 plane

The following should be noted regarding the momentum equation solution parameters:

- The default solution region (default: entire domain) is recommended. Defining multiple regions is recommended only for the more experienced FORCE2 user.

- Each SWEEP statement without the REPLACE or DELETE specifications adds a set of solution parameters to the control table. The table is retained on restart runs. Consequently, SWEEP statements that were specified on the initial run are executed on all subsequent restart runs unless replaced or deleted.

Finally, the phase continuity equations are solved for gas void fraction using the sweeping strategy specified for the momentum equations.

4.5.3 Particle Properties

Particle size and shape factor (sphericity) are specified with the following:

```
DIAMETER   value
value      Particle  diameter
```

```
SPHERICITY   value
value        Particle sphericity
```

4.5.4 Gas/Solids Modeling Parameters

The solids stress (G) in the solids momentum equations is calculated as a function of gas void fraction (ε) according to

$$G(\varepsilon) = G_n \cdot EXP[-S_m \cdot (\varepsilon - \varepsilon^*)]$$

The normalized stress (G_n), slope (S_m), and packing void (ε^*) are specified with the following statements:

```
SOLIDS  STRESS  PARAMETER  GNORM   Gn
SOLIDS  STRESS  PARAMETER  SLOPE   Sm
SOLIDS  STRESS  PARAMETER  EPSTAR  ε*

Gn      Normalized Stress  (default: .1 N/M**2)
Sm      Slope              (default: 600      )
ε*      Packing void       (default: .384     )
```

As noted by Lyczkowski (1989), two sets of momentum equations have been developed to predict solids flow. In one set, denoted Model A, the pressure gradient is included; in the other, denoted Model B, the pressure gradient is not. The solids hydrodynamic model is selected with one of the following:

```
MODEL A
  or       (default : MODEL B)
MODEL B
```

4.5.5 Drag Formulation

Two models (denoted scalar and vector) are available for calculating interfacial drag. In the scalar model, relative velocity is calculated based on the magnitudes of the gas and solids total (vector sum) velocities. In the vector model, relative velocity is calculated based on the gas and solids velocities in the coordinate directions of interest. The drag model is selected with the following:

```
SCALAR DRAG
   or      (default: SCALAR DRAG)
VECTOR DRAG
```

4.5.6 Flow Obstructions

The flow obstruction section of the FLOW paragraph specifies "porous" flow obstructions that do not completely block the flow within or at the face of a control volume. The loss factors to determine the associated unrecoverable pressure drop are specified here. Constant factors can be specified or can be calculated in a separate routine that is provided by the user. Each resistance (constant or user defined) and the associated geometry are stored in tables (one for each phase) that are retained on restart runs.

The input data specification for a flow obstruction starts with a heading and is followed by statements defining the geometry and loss factor if appropriate.

For a constant loss factor, the obstruction is identified as follows:

```
DISTRIBUTED [RESISTANCE] GAS [N]
            or
DISTRIBUTED [RESISTANCE] SOLIDS [N]

     MULTIPLIER [FACTOR]          Kmult
     LOSS       [FACTOR]          K
     X1                           X1[X2min, X2max, X3min, X3max]
     X2                           X2[X1min, X1max, X3min, X3max]
     X3                           X3[X1min, X1max, X2min, X2max]
     X1         [RANGE]      X1min, X1max, X2min, X2max, X3min, X3max
     X2         [RANGE]      X1min, X1max, X2min, X2max, X3min, X3max
     X3         [RANGE]      X1min, X1max, X2min, X2max, X3min, X3max
     DELETE
```

where:

GAS	means the loss factor will be applied in the gas phase
SOLIDS	means the loss factor will be applied in the solids phase
N	resistance number. N is used only on restart runs to change parameter (K, Kmult, Coordinates) of an existing distributed loss factor or to delete factor number, N.
Kmult	is a multiplier for K
K	is the loss factor
X1,X2,X3	identifies the flow direction. The distributed loss will be applied to flow in the XN (N = 1, 2 or 3) direction.
x1	is the position of the obstruction on the X1-axis for a resistance to flow in the X1-direction.
x2	is the position of the obstruction on the X2-axis for a resistance to flow in the X2-direction.
x3	is the position of the obstruction on the X3-axis for a resistance to flow in the X3-direction.
xnmin	is the position of the lower edge of the obstruction on the Xn-axis.
xnmax	is the position of the upper edge of the obstruction on the Xn-axis
RANGE	indicates that the resistance factor will be applied to all nodes in the identified range
DELETE	means delete resistance number N in the resistance table

If the loss factor is to be calculated in a user supplied routine (called subroutine USRRES), the following statements identify this option:

```
USER [DEFINED] [RESISTANCE] GAS [N]
                 or
USER [DEFINED] [RESISTANCE] SOLIDS [N]

     RESISTANCE  [NUMBER]  num
     X1                             X1[X2min, X2max, X3min, X3max]
     X2                             X2[X1min, X1max, X3min, X3max]
     X3                             X3[X1min, X1max, X2min, X2max]
     X1             RANGE       [X1min, X1max, X2min, X2max, X3min, X3max]
     X2             RANGE       [X1min, X1max, X2min, X2max, X3min, X3max]
     X3             RANGE       [X1min, X1max, X2min, X2max, X3min, X3max]
     TUBE [DIAMETER] DT
     DELETE
```

where:

GAS	means the loss factor will be applied in the gas phase
SOLIDS	means the loss factor will be applied in the solids phase
N	resistance number. N is used only on restart runs to change the parameters of an existing user-defined resistance or to delete resistance number N.
num	a parameter that is passed to subroutine users. It can be used to identify correlations, etc.
X1,X2,X3	identifies the flow direction. The resistance will be applied to flow in the XN (N = 1, 2 or 3) direction.
x1	is the position of the obstruction on the X1-axis for a resistance to flow in the X1-direction.
x2	is the position of the obstruction on the X2-axis for a resistance to flow in the X2-direction.
x3	is the position of the obstruction on the X3-axis for a resistance to flow in the X3-direction.
xnmin	is the position of the lower edge of the obstruction on the Xn-axis
xnmax	is the position of the upper edge of the obstruction on the Xn-axis

RANGE	indicates that the resistance factor will be applied to all nodes in the identified range
Dt	tube diameter to be used with the ANL bundle resistance model (this input applicable only when subroutine ANRES is used instead of USRRES).
DELETE	means delete resistance number N

It should be noted that each DISTRIBUTED or USER resistance statement without the DELETE option adds an entry to the resistance table. The table is retained on restart runs. Consequently, resistances that were specified in the initial run are included on all subsequent runs unless deleted.

4.5.7 Mass Sources

Gas and solid mass sources may be included in the problem using the FGAS, FSOL and GSOU field variables identified in Table 4-3. The variables, FGAS and FSOL, are used to simulate gas (FGAS) and solids (FSOL) inflow to the system at locations not on the boundaries of the computational domain (i.e., they are used to simulate interior feed ports). FGAS and FSOL are intended to model local mass sources. The variable, GSOU, is used to simulate gas generation due to combustion and is intended to be a distributed mass source. These local and distributed mass sources must first be activated with the following:

- For a local gas source using FGAS

FEED PORT GAS

- For a local solids source using FSOL

FEED PORT SOLIDS

---

o For a distributed gas source using GSOU

MASS SOURCE GAS

The sources are then defined by initializing FGAS, FSOL and/or GSOU in the INITIALIZATION paragraph. The appropriate units are:

Variable	Units
FGAS	Mass/time
FSOL	Mass/time
GSOU	Mass/(time*volume)

4.5.8 Body Forces

The gravity input is for the buoyancy source term in the momentum equations. The GRAVITY statement defines the magnitude of gravitational acceleration and its orientation with respect to the coordinate system.

GRAVITY g ϕ_g θ_g

g — The magnitude of the gravitational acceleration in the system units. (default: $g = 9.8$ m/sec^2)

ϕ_g — Angle in degrees between the minus X1 direction and the gravitational acceleration vector (see Figure 4-3).

θ_g — Angle in degrees between the projection of the gravitational acceleration vector into the X2-X3 plane and the positive X2 direction (see Figure 4-3).

The default gravity statement is:

```
GRAVITY   9.8   0.0   0.0
```

4.6 CONTROL PARAGRAPH

The CONTROL paragraph contains commands to control the problem solution. Information specified in this paragraph includes the number of iterations, relaxation parameters, and print control. RELAXATION and PRINT CONTROL can be applied to any of the FIELD VARIABLES listed in Table 4-2.

4.6.1 Iterations in the Steady Mode

The ITERATIONS statement specifies the number of macro iterations to be performed in a steady flow.

ITERATIONS n

n number of solution iterations if a steady run (default: 0).

If there is no ITERATIONS command or if the number of iterations is zero, no solution iterations are performed. By including the appropriate print commands, a run with zero iterations which is restarted from a previous solution can be used to print results which were not printed originally.

4.6.2 Relaxation in the Steady Mode

The finite difference equations that must be solved for the steady-state solution are strongly nonlinear and coupled. Because of this, relaxation is used to stabilize the iterative solution process.

Relaxation reduces the change in a variable from one iteration to the next, so that the solution does not diverge.

Separate relaxation parameters are available for each variable so that they may be relaxed independently.

Two forms of relaxation are available in FORCE, under-relaxation and inertial relaxation.

Under-relaxation

Under-relaxation modifies the change in a variable by a constant factor. If the change in the variable, ϕ, predicted by solving the finite difference equations without under relaxation is d, then the relaxed change is r·d where r is the under-relaxation factor.

$$\phi_{new} = \phi_{old} + r \cdot d$$

A typical value for r is 0.5. A value of 0.0 inhibits changes and a value of 1.0 is equivalent to no under-relaxation.

The under-relaxation parameter for a field variable is specified by the relaxation statement.

RELAX var r

var	name of field variable
r	under-relaxation value (default: 0.5)

Under-relaxation is applied in the mass conservation equations used to calculate gas void fraction. Based on Carver's (1983) approach, under-relaxation is applied to improve solution convergence at low (VMIN) and high (VMAX) void fractions. The input data specification starts with a heading and is followed by statements defining the limiting void and relaxation parameters. The heading and statements are as follows:

```
VOID [RELAXATION] [PARAMETER]
     VMAX    vmax    omax
     VMIN    vmin    omin
     NOMINAL         onom
```

where:

vmax	Maximum void. For voids above this value the under-relaxation factor omax will be applied
omax	Under-relaxation factor above a gas void fraction of vmax
vmin	Minimum void. For voids below this value the under-relaxation factor omin will be applied
omin	Under-relaxation factor below a gas void fraction of vmin
onom	Under-relaxation factor above voids of vmin and below voids of vmax.

In a typical fluid bed application with the critical void (ϵ*, defined in the FLOW paragraph) specified as .38, void fraction may be under-relaxed according to the following:

```
VOID RELAXATION PARAMETER
     VMAX   .9999   .01
  VOID RELAXATION PARAMETER
     VMIN   .38   .01
  VOID RELAXATION PARAMETER
     NOMINAL    1.0
```

For some applications, under-relaxing body forces (caused by gravity) has been found to improve solution convergence. The approach is to gradually apply gravity forces as the solution develops according to

$$g_{eff} = g_{act} \cdot F$$

with

F = N/ngr   for N<ngr

F = 1.0   for N>ngr

where

g_{eff} = "Effective" gravitational acceleration
g_{act} = Actual gravitational acceleration defined in the FLOW paragraph
ngr = Number of macro iterations taken to apply the gravity forces
N = Macro iteration number

Relaxation of the body forces is accomplished with the following:

```
RELAX  GRAVITY  ngr
   ngr     Number of macro iterations taken to apply gravity forces
```

The momentum equations in a particular direction are coupled through the drag forces. Solution convergence is sometimes improved by relaxing this linkage during the simultaneous solution for gas and solids velocities. Relaxing the linkage due to interphase drag is accomplished by

---

```
RELAX  DRAG  GAS      r
             SOLIDS
```

where: GAS — Means relax the interphase drag in the gas momentum equations.

SOLIDS — Mean relax the interphase drag in the solids momentum equations

r — Under-relaxation factor (default : 1.0)

For example, the statements

```
RELAX DRAG GAS      0.5
RELAX DRAG SOLIDS   0.0
```

result in the following

- In the gas momentum equation, half of the drag force is modelled by the coupling to the solids equation and half as a constant term based on current solids velocities.

- In the solids momentum equation, all the drag force is modelled by a constant term based on current gas velocities. With 0.0 under-relaxation, there is no coupling to the gas equation during the simultaneous solution.

Inertial Relaxation

When a variable is calculated from a finite difference equation, inertial relaxation may be used. The finite difference equation for a variable, ϕ, at a node, p, may be written,

$$a_p \phi_p = \sum_{nb} a_{nb} \phi_{nb} + S_\phi$$

This is relaxed by adding an inertial term $I\phi_p$ to both sides of the equation.

$$(a_p + I)\phi_p = \sum a_{nb}\phi_{nb} + S + I\phi_{old}$$

The relaxation factor I is defined in terms of a_p and an inertial relaxation parameter t,

$$I = a_p t/(1-t)$$

A typical value for t is 0.5. A value of 0.0 is equivalent to no inertial relaxation. Experience indicates that inertial relaxation is usually necessary for the velocities to prevent divergence of the solution.

The inertial relaxation parameter for a field variable is specified by the inertia statement. If the field variable is not specified, inertia is applied to all the field variables.

INERTIA [var t]

var name of variable

t inertial relaxation parameter (default: 0)

4.6.3 Controls in the Transient Mode

Problem time step, mass convergence criteria, maximum problem time and maximum allowable iterations are some of the parameters that control the transient execution of FORCE2. The solution can be stopped based on maximum problem time or maximum number of problem timesteps according to the following:

STOP [TIME] t

and/or

MAXIMUM TIME STEPS n

t Maximum problem time

n Maximum number of timesteps for this run

Problem timestep is specified with the following

TIMESTEP dt

dt Problem timestep

Convergence for a single timestep is based on satisfying

$\varepsilon \cdot \rho \cdot V \cdot epsg > |D \cdot dt|$

where

ε	=	Gas void fraction
ρ	=	Gas density (mass/length**3)
V	=	Volume of the control volume (length**3)
epsg	=	Mass convergence criteria
D	=	Gas mass residual (mass/time)
dt	=	problem timestep

for all control volumes in the solution domain.

The mass convergence criteria, epsg, is specified with the following:

MASS [CONVERGENCE] [CRITERIA] epsg

epsg Mass convergence criteria (default: .0001)

The momentum and continuity equations are solved for a single timestep by performing many sweeps (macro iterations) over the solution domain. On each macro iteration, operating conditions are predicted on a cell by cell basis. Several (micro) iterations at each cell are needed to achieve convergence. In some applications, particularly during the first several timesteps, an excessive number of iterations may be needed to achieve convergence. The convergence difficulties can usually be attributed to poor initial conditions. Consequently, the number of iterations can be controlled by the following:

MAXIMUM MACRO ITERATIONS nma
MAXIMUM MICRO ITERATIONS nmi

nma	Maximum number of macro (sweeps over the entire domain) iterations (default: 200)
nmi	Maximum number of micro (solution iterations at a node) iterations (default: 20)

4.6.4 Single Phase/Two Phase Control

In many applications, parts of the solution domain will contain only gas. In these areas, only the gas equations have to be solved. The "cutoff" for no solids is based on gas void fraction and is specified with the following:

MAXIMUM VOID epsv

epsv	Maximum gas void. The solids equations are not solved for gas void fractions above this limit. (default: .99)

4.6.5 Output Control

The PRINT statement controls those outputs not controlled by other paragraphs. This includes printing of each field variable, statistics, debugging information and page size.

Field Variable Print Control

The PRINT control command specifies the printing options for a field variable. A variable may be printed at the beginning or end of a run. PRINT control statements carry over to RESTART runs.

```
PRINT   var   [INITIAL]  [FINAL]  [OFF]  [CONVERT [TO]] 'units'  [VIA]  A  B
```

var	name of field variable
INITIAL	print prior to first iteration
FINAL	print at the end of the run
OFF	turn off printing of those field variables that are normally printed
CONVERT	convert the units in printout
units	the name of the converted units up to 8 characters
A, B	constants in the unit conversion equation $\phi_{new} = A \cdot \phi_{old} + B$

Example:

```
PRINT  P  FINAL  CONVERT  TO  'PSI'  VIA  1.4504E-04
```

Cause pressure to be printed at the end of the run and converts pascals to psi. It does not change the units of pressure in the FORCE2 calculations or in the restart file.

4.6.5.1 Printer Output. During transient execution field variables may be printed based on problem time or number of timesteps. This may be accomplished with the following:

```
PRINT INTERVAL TIME  dtp
      and/or
PRINT INTERVAL STEPS npr
```

dtp	Problem time interval between print outs
npr	Problem time steps in the transient mode or iterations in the steady mode between print outs

---

4.6.5.2 Graphical Post Processor Output. Two data files are written for the graphical post processor: i) a file with all field variables and ii) a file with selected field variables versus time. During transient execution, the latter is used to collect timewise predictions at user defined locations in the domain.

The frequency at which all field variables are written to a post-processor file is specified by the user. The input data specification starts with a heading and is followed by a control statement as follows:

```
POST PROCESSOR DATA
      SAVE  ALL   na

      na         Number of timesteps between the writing of all field
                 variables to a file
```

For example, the statements

```
POST PROCESSOR DATA
     SAVE  ALL  1000
```

causes all the field variable to be written every 1000 timesteps. In the steady model, all field variables are written at the end of the run.

The frequency at which selected variables are written to a post-processor file is also specified by the user. The variables and their location in the domain are stored in a FORCE2 table. This table is retained on restart runs. Consequently, the STORE statement (below) always adds entries to this table. Currently, the number of entries is limited to 10. The input data specification starts with a heading and is followed by statements that define the variables and the write frequency. This is accomplished with the following:

```
POST PROCESSOR DATA
      SAVE  SELECTED  [VARIABLES]   ns
      DELETE
      STORE   var  [AT]  i1, i2, i3
```

where:

ns	Number of timesteps between the writing of the selected field variables to a file
DELETE	Means to delete last entry in the data storage table
STORE	Means to write the variable to the post processor file
var	is the field variable name (Table 3)
i1,i2,i3	node locations on the X1, X2, and X3 axis respectively for the field variable.

For example the statements

```
POST  PROCESSOR  DATA
      SAVE SELECTED   VARIABLES  100
      STORE  U1G  AT   2,2
      STORE  U1S  AT   2,2
      STORE  VFRG AT   2,2
```

cause the selected variables - gas and solids velocities in the X1 directions and gas void fraction all at node 2,2 - to be written to a graphical post-processor file every 100 timesteps.

4.6.5.3 Output for Erosion Model. Velocity and void fraction predictions are written to a file for subsequent input to the fluid bed erosion model. The frequency at which this data is written is specified with the following:

```
EROSION  [MODEL]  [DATA]  ne
```

ne	Number of timesteps between the writing of the hydrodynamic data for the erosion model.

4.7 PERMEABILITY/POROSITY PARAGRAPH

Obstructions in the flow field can reduce the flow area and volume available for the gas and solids phases. The governing momentum and continuity equations include area permeabilities and volume porosities to account for partial blockages of the solution domain where

Area Permeability = Free flow area/Total area

Volume Porosity = Volume available to the phases/Total Volume

The permeabilities and porosities are specified in the PERMEABILITY paragraph. It is done by first specifying the region using the SETn statement and then specifying the permeability or porosity with additional statements. The SETn statement is defined in section 4.8, Initialization Paragraph. The assignment statements for porosity and permeability are described below.

<u>Assigning Porosity</u>

Porosity is specified with the following statement:

POROSITY val

val	Volume porosity of all nodes in the region defined by the SETn statement(s) (default: 1.0)

<u>Assign Permeabilities</u>

The area permeabilities are specified with the following:

```
X1-PERMEABILITY   px1
X2-PERMEABILITY   px2
X3-PERMEABILITY   px3
```

px1 — Area permeability in the X1-direction over the region defined by the SETn statement(s)

px2 — Area permeability in the X2-direction over the region defined by the SETn statement(s)

px3 — Area permeability in the X3-direction over the region defined by the SETn statement(s)

(defaults: px1 = px2 = px3 = 1.0)

Examples:

i) To specify a volume porosity of .5 over the entire domain in a 3-D problem:

```
SET1
SET2
SET3
   POROSITY   0.5
```

ii) To specify an X1-permeability of .5 over X1 nodes (1 to 10), X2 nodes (2 to 20) and X3 node (3 to 30):

```
SET1  1,  10
SET2  2,  20
SET3  3,  30
   X1-PERMEABILITY   0.5
```

4.8 INITIALIZATION PARAGRAPH

Values are assigned to field variables (Table 4-3) in the INITIALIZATION paragraph. Values can be assigned at the boundaries (boundary values) and the interior points (initial estimates) of the solution domain. SETn statements are used to define the region in which the values are initialized. Once a region is defined with SETn statements, values within the region can be assigned to one or more field variables. One statement is required for each field variable.

All field variables initialized in this paragraph are printed before the iterative solution begins. The field variable will not be initially printed on subsequent runs unless specified by the PRINT statement in the CONTROL paragraph.

4.8.1 SETting Regions

The general form of the SETn statement is:

SETn [m1 [,m2 [,m3]]]

n	the coordinate direction (1, 2 or 3)
m1	the node number at the lower limit of the region
m2	the node number of the upper limit of the region (default: m1)
m3	node increment, the nodes included in the region are m1, m1 + m3, ... m1 + 2m3, (default: 1)

The usual interpretation of node numbers has been expanded to allow specification of nodes relative to the last node. If the last node is n, then 0 is interpreted as n and a negative number, -n, is interpreted as m2-n.

Example:

SET1 1,0

Includes all nodes in direction 1 in the region

SET2 1,-1

Limits the region in direction 2 to all but the last node

4.8.2 Assigning Values

The field variable assignment statement consists of two elements, the field variable name and the value to be assigned.

```
var      value

      var        field variable name

      value      numerical value
```

Example:

```
      U1   1.0               $ Assigns the value 1 to the axial velocity.
      VISCOSITY 1.067E-5    $ Assigns the value 1.067E-5 to viscosity.
```

All locations within the current region are assigned the value specified. If no region has been specified, the value is assigned to all locations in the geometry.

4.8.3 Pressure Initialization

The pressure field can be initialized based on a minimum fluidization condition. In some applications, this approach will provide reasonable starting conditions for the solution. The appropriate statement is:

```
MINIMUM   FLUIDIZATION   idir

      idir        Coordinate direction (1, 2 or 3) of the pressure gradient
```

An initial pressure field is developed based on the following:

- o The specified initial gas and solids void fractions
- o One-dimensional gas flow in direction idir with no solids motion
- o Minimum fluidizing gas velocity

4.9 FLOFLAGS PARAGRAPH

Fluid boundary conditions at inflow, outflow and blocked control volumes are defined in the FLOFLAG paragraph. The definitions here should be consistent with the geometry defined in the GEOMETRY paragraph. Control volumes are identified using the SETn statement and the boundary conditions defined with the following statements:

```
GAS     iflag
SOLIDS  iflag
```

where:

GAS	Mean a boundary condition on the gas phase
SOLIDS	Mean a boundary condition on the gas phase
iflag	is the boundary condition flag defined as follows

iflag	Meaning
-3	Blocked control volume with No-Slip velocity boundary conditions
-2	Blocked control volume with Partial-Slip (applicable to solids phase only) velocity boundary conditions
-1	Blocked control volume with Free-Slip velocity boundary conditions
0	Non-Blocked no flow control volume
1	Free flow control volume
2	Inflow control volume on the domain boundary
3	Continuitive outflow volume on the domain boundary
4	Constant pressure control volume on the domain boundary (applicable only to transient mode)

Example:

i) To specify no slip boundary conditions for the gas phase and partial slip conditions for the solids phase along the lower X2 boundary:

```
SET1
SET2  1, 1
SET3

    GAS     -3
    SOLIDS  -1
```

ii) To specify gas inflow only along the lower X1 boundary:

```
SET1  1, 1
SET2
SET3
    GAS    2
    SOLIDS 0
```

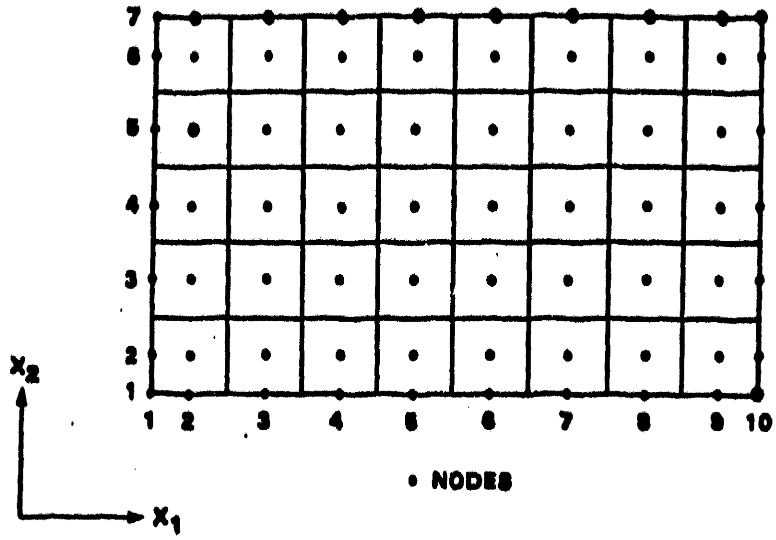
4.10 FORCE2 RUNTIME FILE

The runtime file on Unit 11 (see Table 4-1) contains parameters that control execution of the code. Because it is closed after the parameters are read, the file can be modified during execution of the code. It contains the following three parameters:

GO or STOP
Iter
nread

where:

GO	means continue execution of FORCE2
STOP	means stop execution. All restart and post-processor files are written and closed.
Iter	number of iterations (steady mode) or timesteps (transient mode). This information is written by the code and should not be modified.
nread	read frequency for the file. The file will be read every nread iteration (steady) or nread timesteps (transient).



THIS GEOMETRY IS SPECIFIED BY THE STATEMENT
CARTESIAN 10 7
• MAIN NODE

Figure 4-1. Nodes for all variables except velocity.

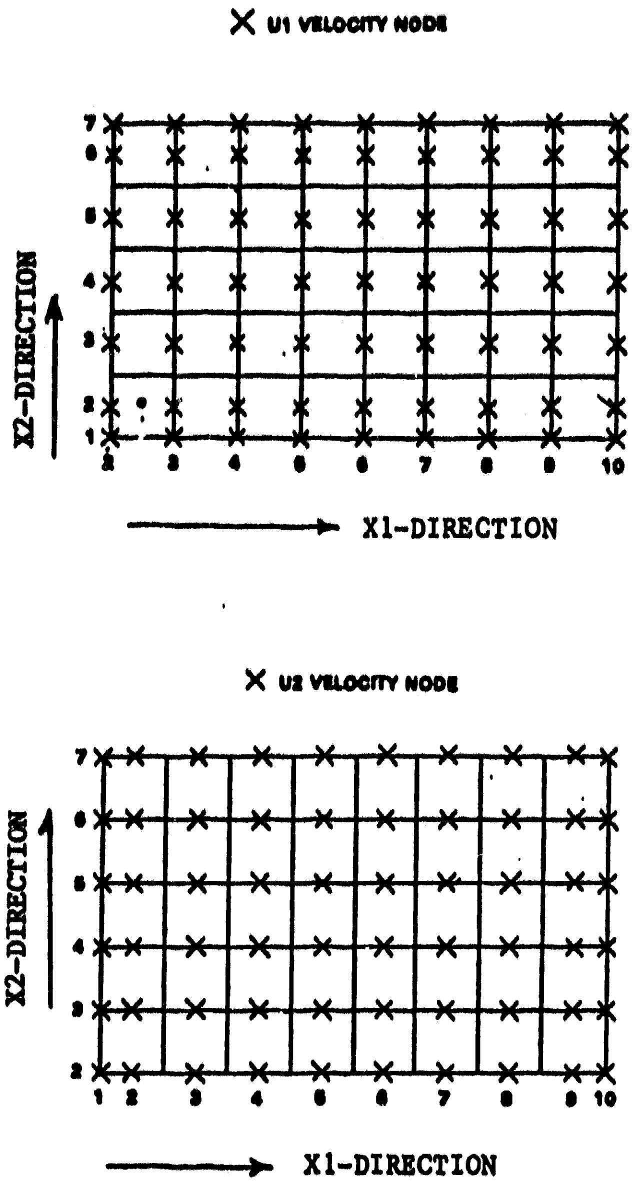
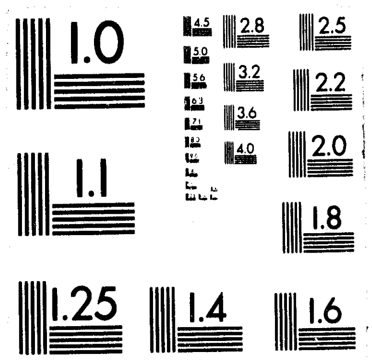


Figure 4-2. FORCE velocity nodes in a typical two-dimensional flow field.


1.0
1.1
1.25
1.4
1.6
1.8
2.0
2.2
2.5
2.8
3.2
3.6
4.0
4.5
5.0
5.6

2 of 3

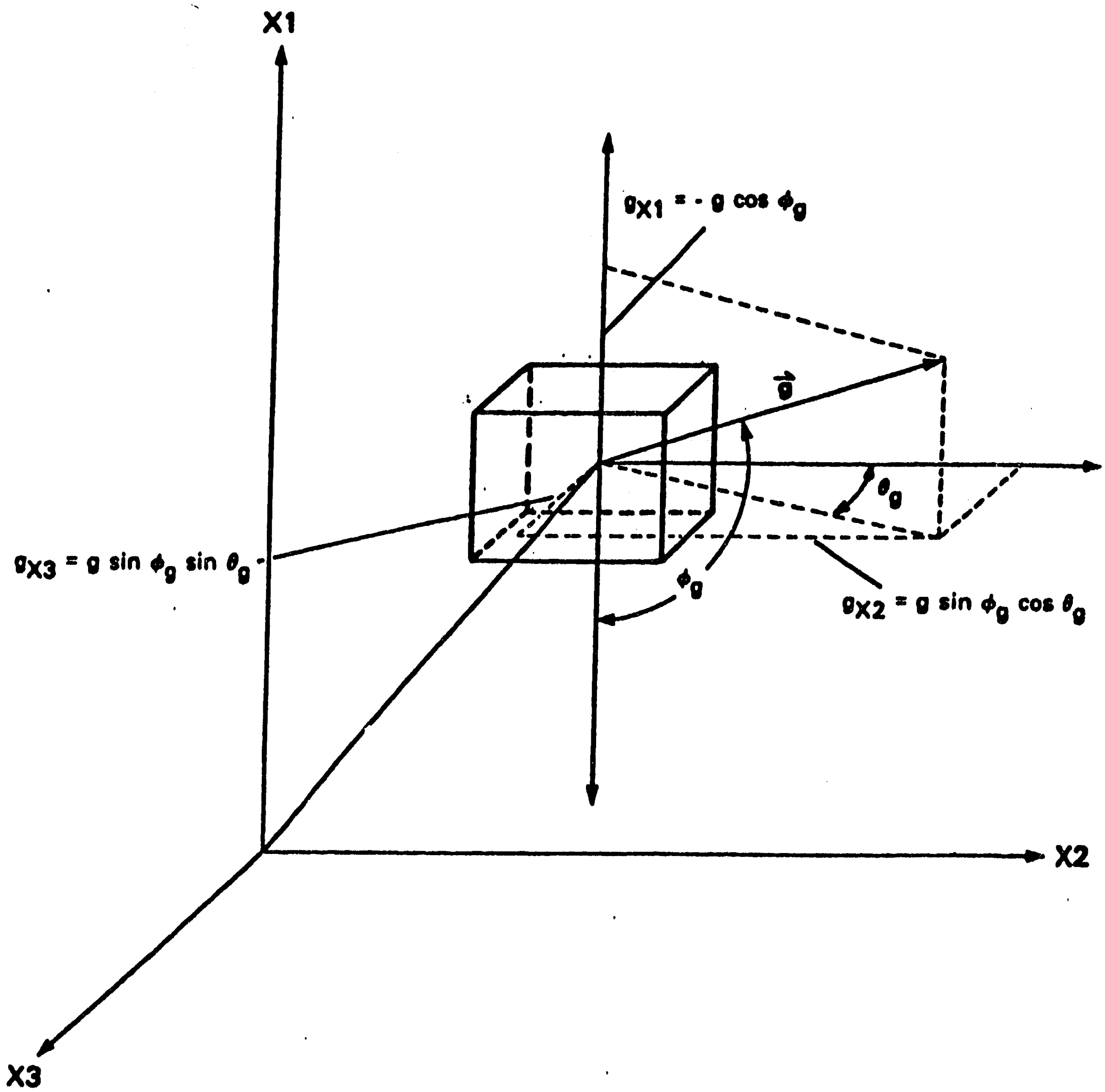


Figure 4-3. Gravity direction in Cartesian coordinates.

5.0 FORCE2 OUTPUT DESCRIPTION

5.1 OVERVIEW

Files that are read or written by FORCE2 are listed in Table 4-1. This section focuses on the printer output file (Unit #6) and the log file (Unit #20).

5.2 PRINTER OUTPUT (UNIT #6)

This file contains the following:

- A banner page with the program name, revision number and revision date.

- A listing of the input data for the run. If it is not a restart run and the pressure field is initialized based on minimum fluidization conditions, the minimum fluidizing velocity (fluid velocity in area available for gas flow, <u>not</u> superficial velocity) and associated gas void fraction are displayed after the FLOFLAGS Paragraph heading.

- A summary of the domain geometry and boundary condition types on the X2 and X3 boundaries. Detailed geometry output may be obtained by the appropriate print command in the Geometry Paragraph.

- A summary of fluid properties.

- A listing of the distributed and user-defined resistances.

- A listing of the velocity, pressure, and void fraction residuals at each iteration if the run is a steady simulation.

- The field variables, identified in the Control Paragraph, are displayed. This output is repeated at the specified problem times, timesteps, or iterations based on the print control entered in the Control Paragraph. If print intervals are not specified, the selected field variables are printed only at the start (using the INITIAL keyword) or end (using the FINAL keyword) of the run.

- If the solution for void fraction during a transient simulation diverges, execution of FORCE2 is stopped and the following message written:

```
--------------------------------------------------------------------------------
The Solution for Void Fraction is Diverging.
Reduce Timestep.  Data to restart will be restored
Plane #, Node, VOIDS, VOIDG:  KPLN, N, εs, εg

Step #, Macro Iter, Micro Iter:   NIT, ITRY
   Operating Parameters Restored
   Time,    Step No.:              T, NS
--------------------------------------------------------------------------------
```

where:

KPLN - nodal location on the X1 axis
N - node number in the X2-X3 plane
$\varepsilon_s, \varepsilon_g$ - predicted gas and solids void fractions
NIT - macro iteration number
ITRY - micro iteration number

During transient execution, operating conditions are saved every fifth timestep. If the solution for void fraction diverges, the last saved conditions are restored, restarts written with these conditions, and FORCE2 execution stopped. The parameters T and NS are the problem time and step number at which conditions are restored.

5.3 LOG FILE (UNIT #20)

Some important solution parameters are listed on the log file. The information will depend on the type of simulation as follows:

Steady Simulation

The log file is arranged as follows

Iteration #: I
Solids mass, %, gas in, gas out: S, P, Wi, Wo

where

- I - iteration number
- S - current solids inventory (mass units or problem)
- P - percent increase or decrease in solids mass based on the inventory for the initial run, i.e.,

$$P = \frac{M - S}{M} \times 100$$

M = initial inventory

- Wi - total gas flow rate entering the domain
- Wo - total gas flow rate exiting the domain

The gas flow rates will be equal at each iteration. Finally, the log file is printed at each MREAD iteration where MREAD is the read frequency parameter on the FORCE2 runtime file (see Section 4.10).

Transient Simulation

The log file is arranged as follows:

Line 1	: Time, Tot. Steps, TRYS: T, NS, NIT
Line 2	: Max Resid, I, J, K: RM, ±KPLN, J, K
Line 3	: Solids Mass, %, Gas In, Gas Out: S, P, Wi, Wo
Line 4	: Non-convergent Nodes: N
Lines 5-10 (max)	: ITRYS, I, J, K, Max-R: IT, KPLN, J, K, R

where

T - problem time

NS - problem timestep number

NIT - number of macro iterations

RM - absolute value of maximum gas residual

±KPLN - nodal location along the X1-axis. The sign indicates the sign of the residual

J,K - nodal location along X2 and X3 axes

S,P,Wi,Wo - as defined above for the steady simulation. However, for the transient simulation, the gas flows are generally not equal.

Lines 4,5,---10 will not appear if the mass convergence criteria is achieved for all nodes. Otherwise, the following is displayed:

N - number of nodes for which convergence was not achieved

IT - number of micro iterations required at this node on the last macro iteration

KPLN,J,K - nodal location on the X1, X2 and X3 axis

R - gas residual at the node

The gas residual, R, is defined as

$$R = \left[\frac{\partial \rho'}{\partial t} - \frac{\partial(\rho' u)}{\partial x} - \frac{\partial(\rho' v)}{\partial y} - \frac{\partial(\rho' w)}{\partial z} - s\right] \cdot \Delta t$$

and

$$RM = |R|$$

where

ρ' = macroscopic density

= $\varepsilon \cdot \rho_g$

ε = gas void fraction

ρ_g = gas microscopic density

u,v,w = gas velocities in x, y and z direction

R = volumetric gas source

t = time

x,y,z = coordinate directions

Δt = problem timestep

If the solution for void fraction diverges, the message noted above is also written to the log file.

6.0 EXAMPLES

Two sample applications are provided to demonstrate FORCE2 and its post-processor.

6.1 FLUFIX STANDARD PROBLEM

This sample problem has been used by Lyczkowski (1986) to demonstrate the FLUFIX code. The problem statement below was taken directly from the FLUFIX manual (Lyczkowski, 1986).

The geometry of the sample problem given here for a fluidized bed is shown in Figure 6-1. The computational region is 19.685 cm wide, 58.44 cm high. The cell dimensions are $\Delta x = 0.635$ cm and $\Delta y = 4.87$ cm, so that the number of computational cells is 31 in the x direction and 12 in the y direction, for a total of 372. In the figure, the numbers in parentheses refer to key cell numbers (I,J). Symmetry about the central jet is assumed; hence the actual bed width is 39.37 cm. (In previous modeling work without an obstacle, symmetry was assumed and the agreement with experimental data was good.) The jet half-width is 0.635 cm (one cell width). The jet velocity is 578 cm/s, and the secondary air velocity of 26.0 cm/s maintains the bed without a jet at minimum fluidization. The particle diameter is 503 μm, and the density is 2.44 g/cm^3. The obstacle is placed two nodes above the jet and is two nodes (1.27 cm) wide by two nodes (9.74 cm) high. Because the initial bed height is 29.22 cm (six cells high), the obstacle lies completely within the bed.

The boundary and initial conditions are next described. At the inlet (J = 2), the axial gas velocity of 26.0 cm/s is set equal to the minimum fluidization superficial velocity as computed from the Ergun equation (with solids velocity set to zero), using an assumed gas volume fraction at minimum fluidization of 0.42. Because no solids are entering, the inlet porosity is set to 1.0. The pressures in the dummy cells at the top (J = 14) are set equal to atmospheric pressure (1.013×10^5 Pa), and $V_s = 0$ at the exit (J = 13); that is, wire mesh is simulated to prevent solids carryover. The pressures in the bottom row of dummy cells (J = 1) are set equal to atmospheric pressure plus 1.2 times the total bed weight (1.0549 x 10^5 Pa). On all solid surfaces except the inlet, outlet, and line of symmetry, no-slip boundary conditions are used (i.e., normal and tangential velocities for each phase are set equal to zero). Free-slip boundary conditions are used along the line of symmetry and at the inlet and outlet. Initially, the radial gas velocity is zero, the axial gas velocity is equal to the interstitial gas velocity at minimum fluidization, and the solids radial and axial velocities are zero. The bed porosity is uniform at 0.42. The initial pressure distribution corresponds to the hydrostatic bed height. At time, t, greater than zero (0+), the gas flow through the jet is increased to 578 cm/s. A fixed timestep of 0.1 ms is used.

6.1.1 FORCE2 Input Data

The geometry arrangement used in the FORCE2 model is shown in Figure 6-2. The input data for the initial and restart runs is given in Appendix B.

6.1.2 Running FORCE2

Two FORCE2 transient runs were made for this sample problem: an initial run of 10 timesteps followed by a restart run of 40 timesteps. In the restart run, time data was stored for subsequent plotting with the post-processor. The output listing and log file for the initial run are given in Appendix B.

6.1.3 Running the Post-Processor

FORCE2 predictions after the restart run are displayed with the post-processor. The field variable and time data files were called:

Field Data: p33x14.ppf
Time Data: p33x14.ppt

The boundary data file is given in Appendix B.

The post-processor instructions (session file) in Table 6-1 will create the geometry, gas vector, pressure contour and void/time plots given in

Figures 6-3 through 6-6 based on the field data at the end of the run. Note that the symbol <CR> has been added to the session file in Table 6-1 to indicate that the default value was chosen. This symbol does not appear in the session file written by the post-processor.

6.2 CAPTF-3D BUNDLE

As part of a joint Government-industry research program (Metal Wastage in Fluidized Bed Combustion) to better understand erosion in fluidized beds, an air-fluidized bed with five tubes was tested in the computer-aided particle tracking facility (CAPTF) at the University of Illinois (private communications, 1990). A transient simulation of tests conducted on this 3-D bundle was performed with FORCE2. The FORCE2 model and transient runs are outlined below.

Salient features of the bed and tube bundle are depicted in Figure 6-7. The bed is 12 inches wide and 12 inches deep with a static height of 16 inches. The bundle consists of three whole tubes and two half tubes mounted on the side walls. Additional features and operating conditions are tabulated in Table 6-2.

Table 6-2
PARAMETERS FOR THE 3-D CAPTF BUNDLE TEST

Parameter	Value
Mean particle diameter	.0513 cm
Particle density	2.49 G/cc
Tube OD	5.08 cm
Fluidizing velocity (superficial)	52 cm/s
Approximate minimum fluidizing velocity (superficial)	27 cm/s
Static bed height	40.64 cm
Gas void fraction at minimum fluidizing condition (estimated)	.404

6.2.1 FORCE2 Input Data

Important features of the FORCE2 model geometry are shown in Figure 6-8. The input data for the initial and restart runs is given in Appendix C. Some important characteristics of the model include:

- Three-Dimensional - Three active cells are used to model the full bed depth (X3-direction). Approximately 1134 active control volumes make up the model.

- Windbox and Distributor Plate - The windbox and distributor plate are modeled with different length control volumes and the FORCE2 distributed resistance and permeability/porosity options. For the windbox, a user-defined resistance function was used to implement Ergun's equation for pressure drop (the windbox is a packed bed of spheres). For the distributor plate, small permeabilities in the X2 and X3 directions along with large distributed loss factor were specified to prevent gas flow in these directions. The size (in the X1-direction) of the distributor plate was chosen to be approximately equal to the length of the lower nodes in the bundle (2.54 cm) based on gas residence time/required timestep considerations. If the plate nodes were the same length as the actual plate (.635 cm), the problem timestep would probably be controlled by the plate nodes based on the material Courant condition. Because the model includes the windbox and distributor plate, the air flow distribution to the bed is calculated by FORCE2, not specified as an in-flow boundary condition.

- Variable Length Nodes Above the Bed i the X1-Direction - This FORCE2 feature is used to reduce model size by eliminating nodes in the area above the bed.

- Tubes Modeled with Blocked Cells

- Pressure Field Initialization - The pressure field is initialized based on calculated minimum fluidizing conditions.

- Combined No-Slip/Partial-Slip Boundary Conditions at the Walls and Tube - No-slip boundary conditions for the gas phase and partial-slip conditions for the solids phase are applied at all solid surfaces (walls and tubes).

6.2.2 Running FORCE2

A series of transient runs were made using the restart feature. The approach taken was as follows:

- 0.0 - .7 Sec: The gas in-flow velocity was maintained at the minimum fluidizing condition. This initial phase of the simulation results in some solids motion, particularly around the tubes.

- .7 - 1.5 Sec: The gas in-flow velocity was increased from 27 cm/sec to the final operating value of 52 cm/sec in three restarts over this time period.

- 1.5 - 2.0 Sec: The gas in-flow velocity was maintained at the final value.

- 2.0 - 3.067 Sec: The gas in-flow velocity was maintained at the final value and pressure predictions tabulated for comparison to test data.

- 3.067 - 3.070 Sec: Sample run for this document.

6.2.3 Running the Post-Processor

FORCE2 predictions at 3.070 seconds as well as selected operating conditions over the time span from 3.067 to 3.07 seconds are displayed with the post-processor. The field variable and time data files were called

Field Variables:	captf.ppf
Time Data:	captf.ppt

The boundary data file for this problem is given in Appendix C.

The post-processor instructions (session file) in Table 6-3 will create the vector, pressure profile, and pressure vs time plots shown in Figures 6-9, 6-10, and 6-11, respectively. The symbol <CR> has been added to the session file shown in Table 6-3 to indicate that the default value was chosen. This symbol does not appear in the session file written by the post-processor.

Table 6-1

POST-PROCESSOR COMMANDS TO CREATE THE PLOTS FOR THE STANDARD FLUFIX PROBLEM (FIGURES 6-3, 6-4, 6-5 AND 6-6)

```
2501
<CR>
<CR>
p33x14.ppf
10-50 Timesteps
p33x14.ppt
<CR>
<CR>
Standard FLUFIX Problem
<CR>
N
.1
<CR>
<CR>
<CR>
Data After 50 Timesteps
G
<CR>
<CR>
<CR>
.17
<CR>
V
<CR>
<CR>
<CR>
<CR>
<CR>
200
<CR>
C
<CR>
<CR>
<CR>
<CR>
4
<CR>
P
<CR>
<CR>
<CR>
<CR>
<CR>
<CR>
T
.001
<CR>
1
<CR>
E
```

Table 6-3

POST-PROCESSOR COMMANDS TO CREATE THE PLOTS FOR THE CAPTF 3-D BUNDLE PROBLEM (FIGURES 6-9, 6-10 AND 6-11)

```
2501
<CR>
<CR>
captf.ppf
30670-30700 Timesteps
captf.ppt
<CR>
<CR>
CAPTF 3-D Bundle Problem
Data After 30700 Timesteps
N
3.07000
<CR>
<CR>
<CR>
Data After 30700 Timesteps
V
3
3
<CR>
<CR>
<CR>
.1
<CR>
300
<CR>
P
<CR>
3
<CR>
<CR>
3,9
P
<CR>
T
3.067
3.07
6
<CR>
E
```

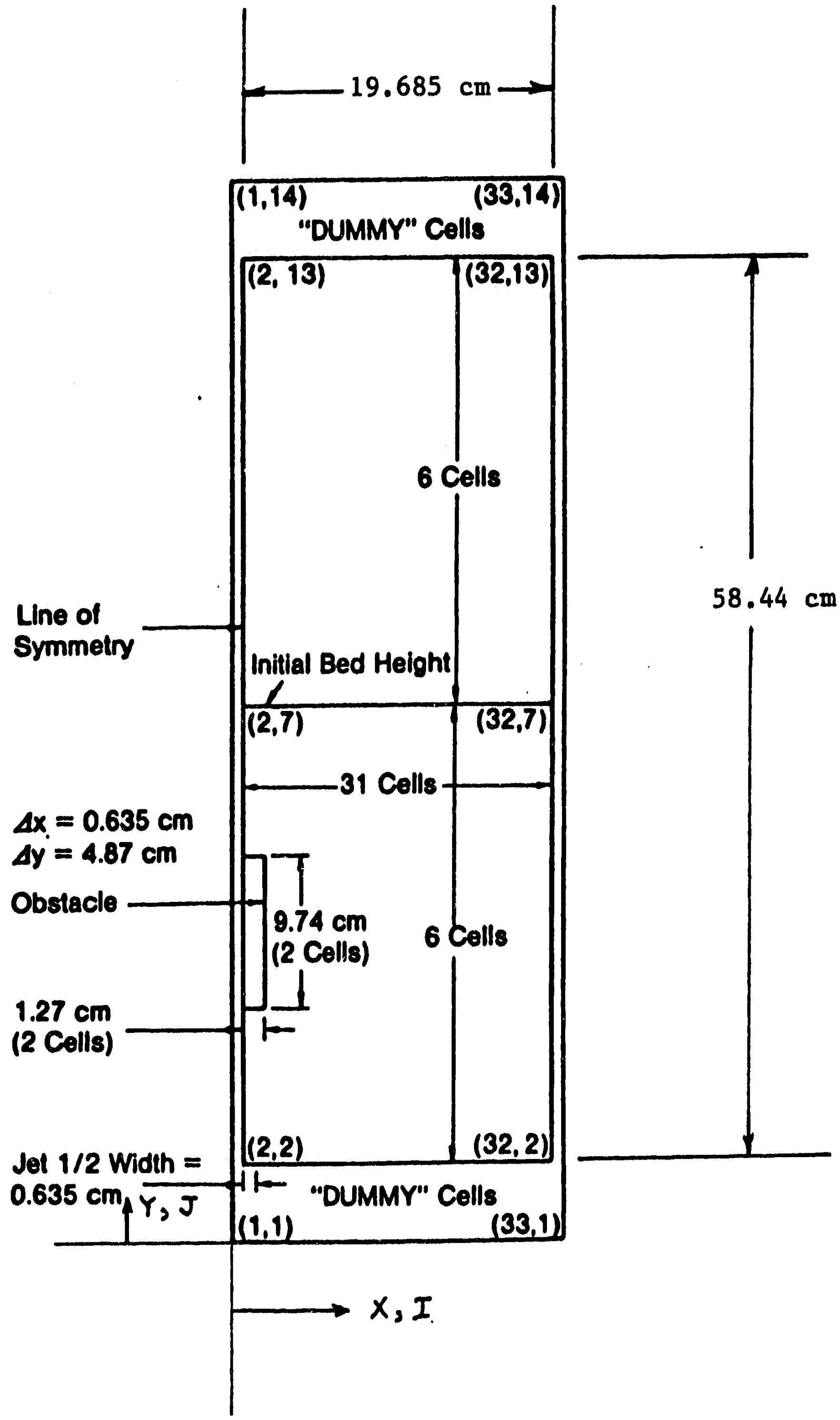


Figure 6-1. FLUFIX sample problem geometry.

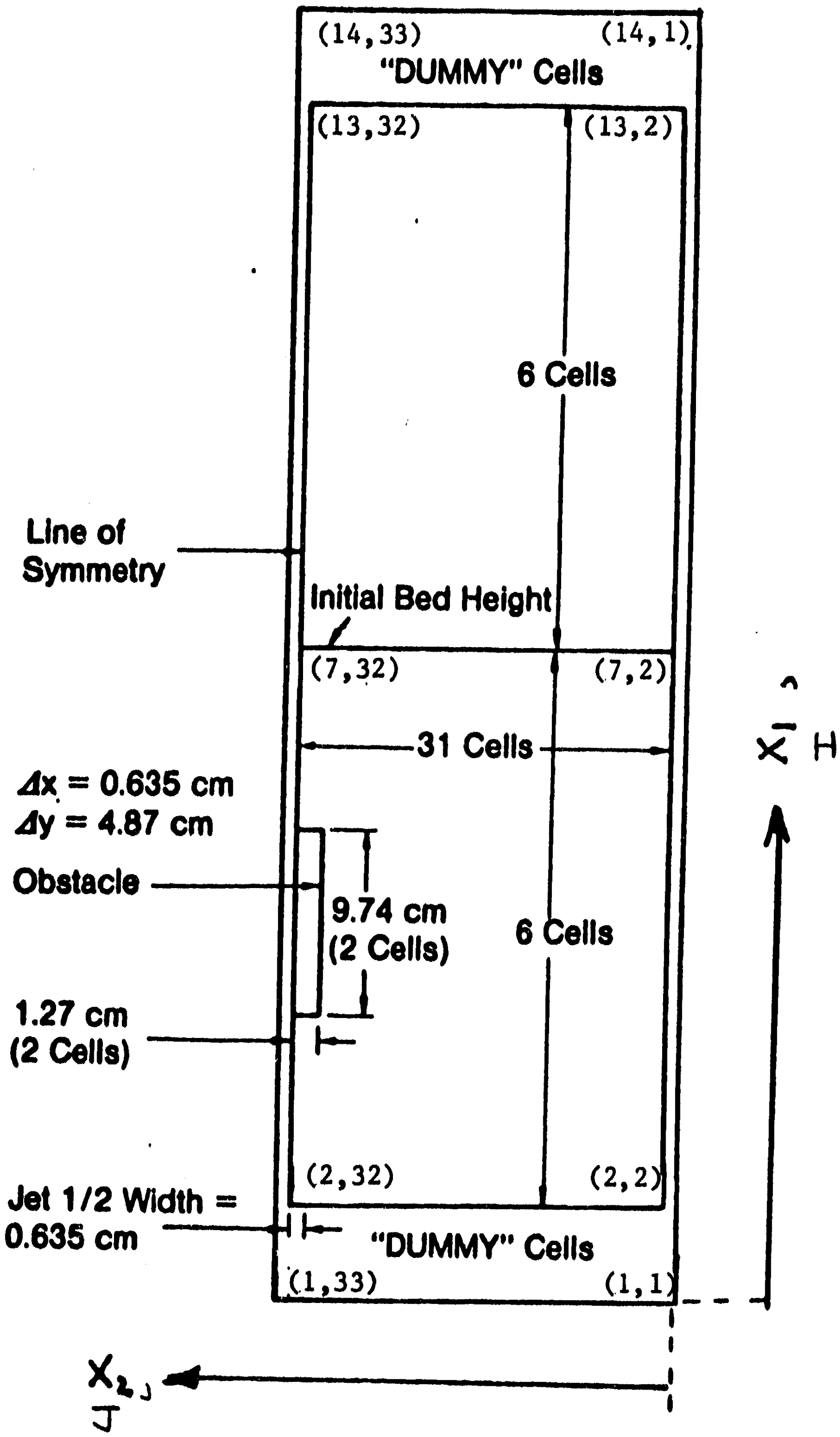


Figure 6-2. FORCE2 sample problem geometry.

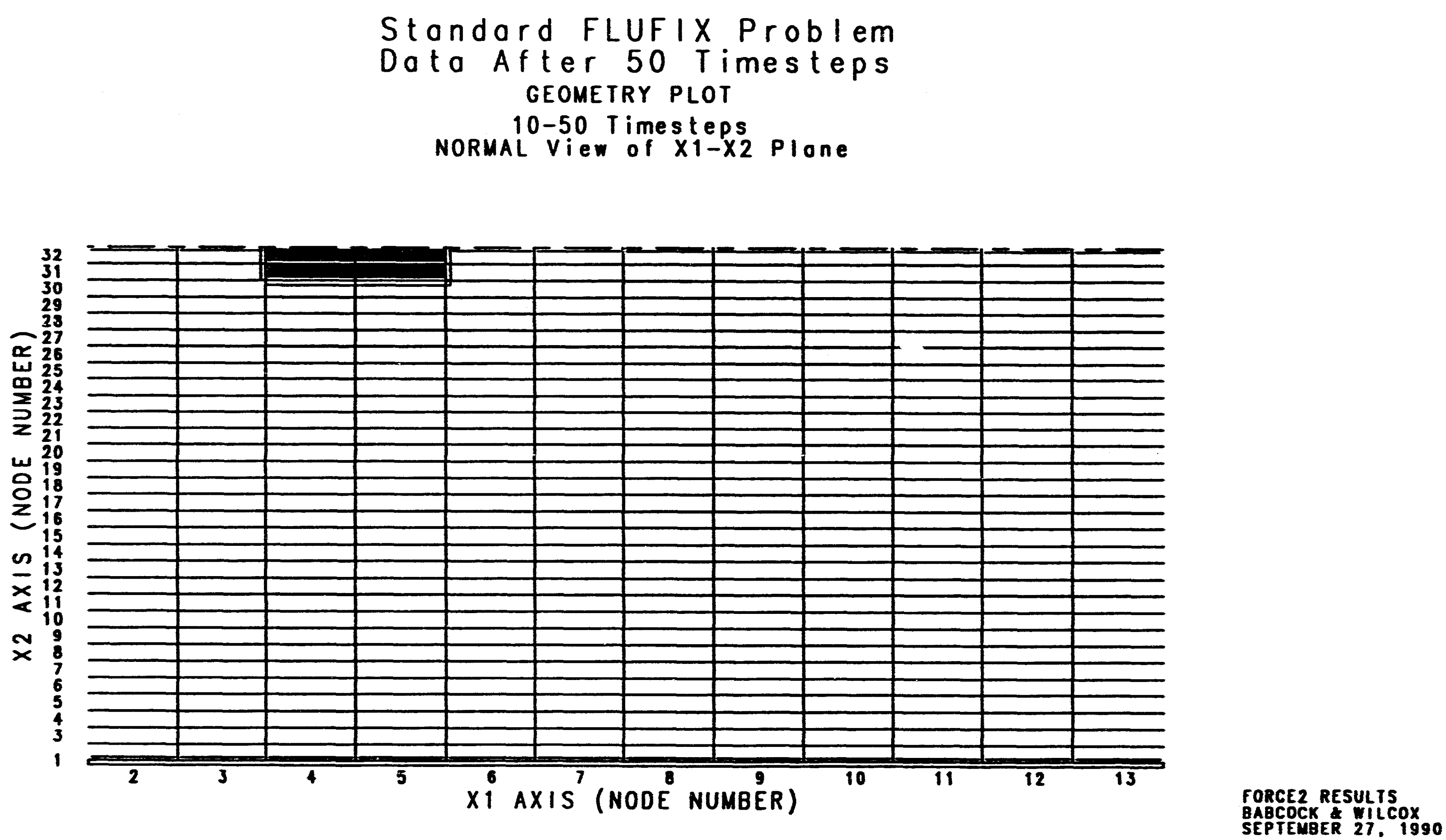


Figure 6-3. FLUFIX standard problem geometry.

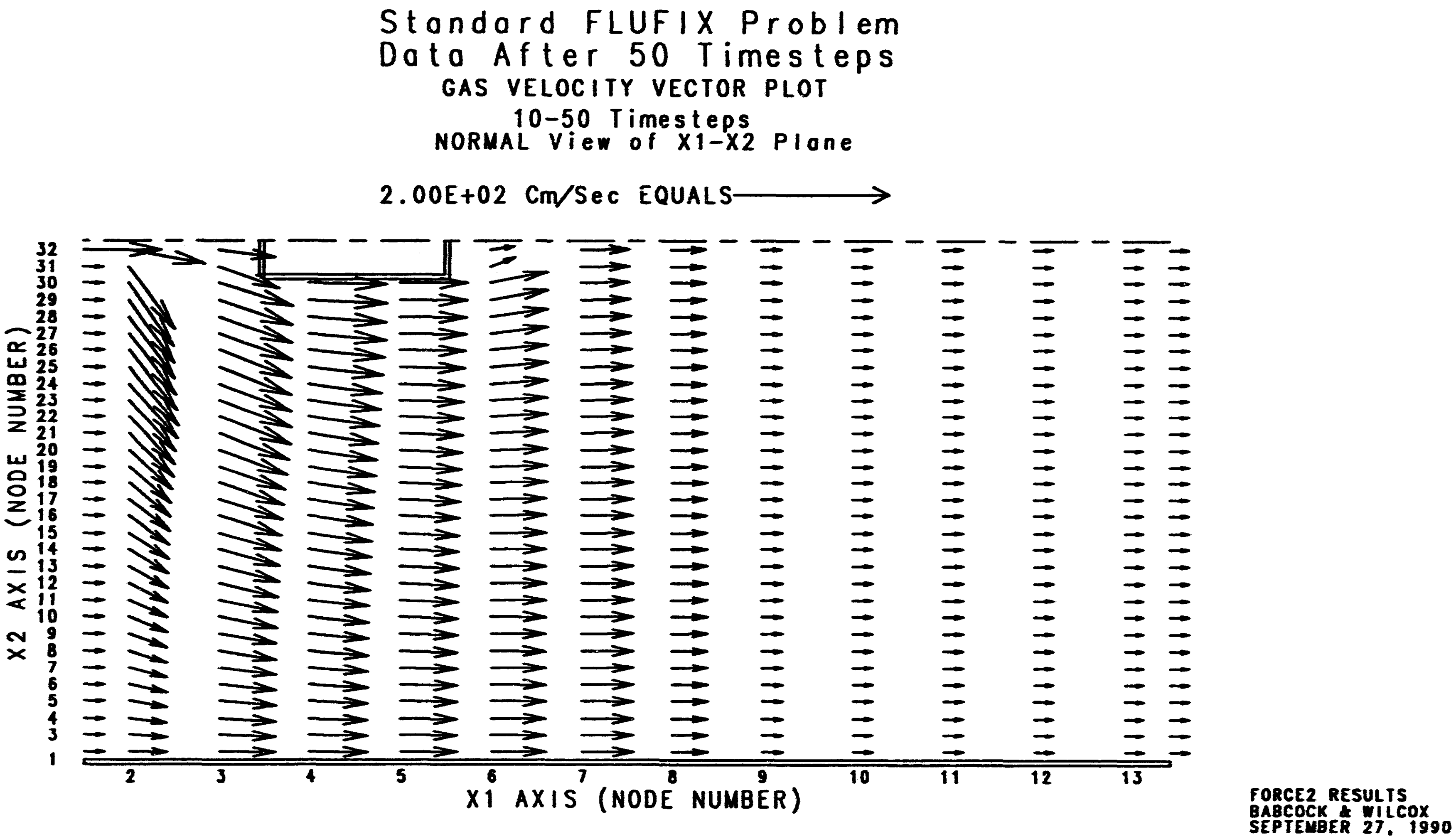


Figure 6-4. Gas velocity vectors after 50 timesteps, standard FLUFIX problem.

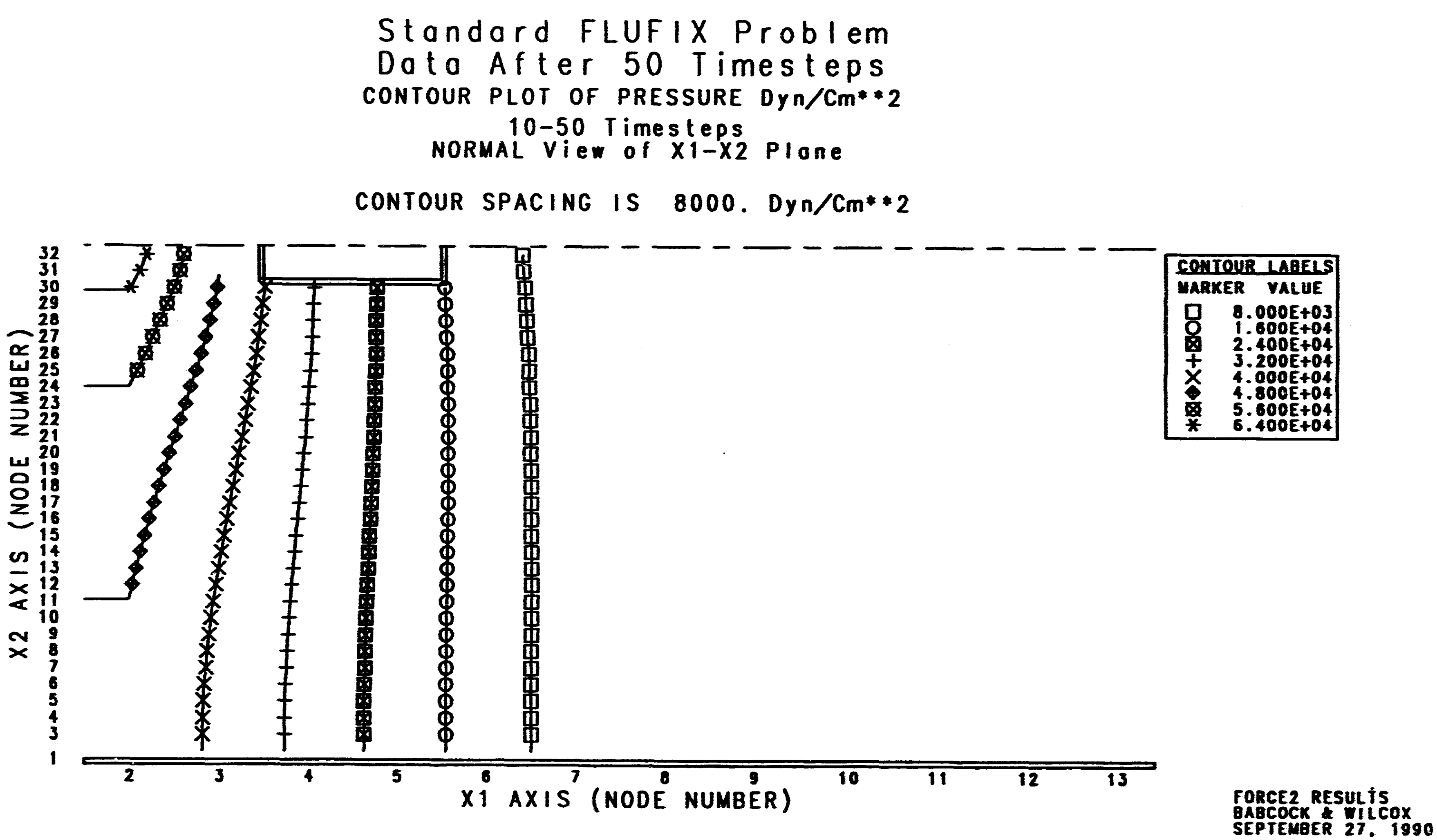


Figure 6-5. Contours of pressure after 50 timesteps, standard FLUFIX problem.

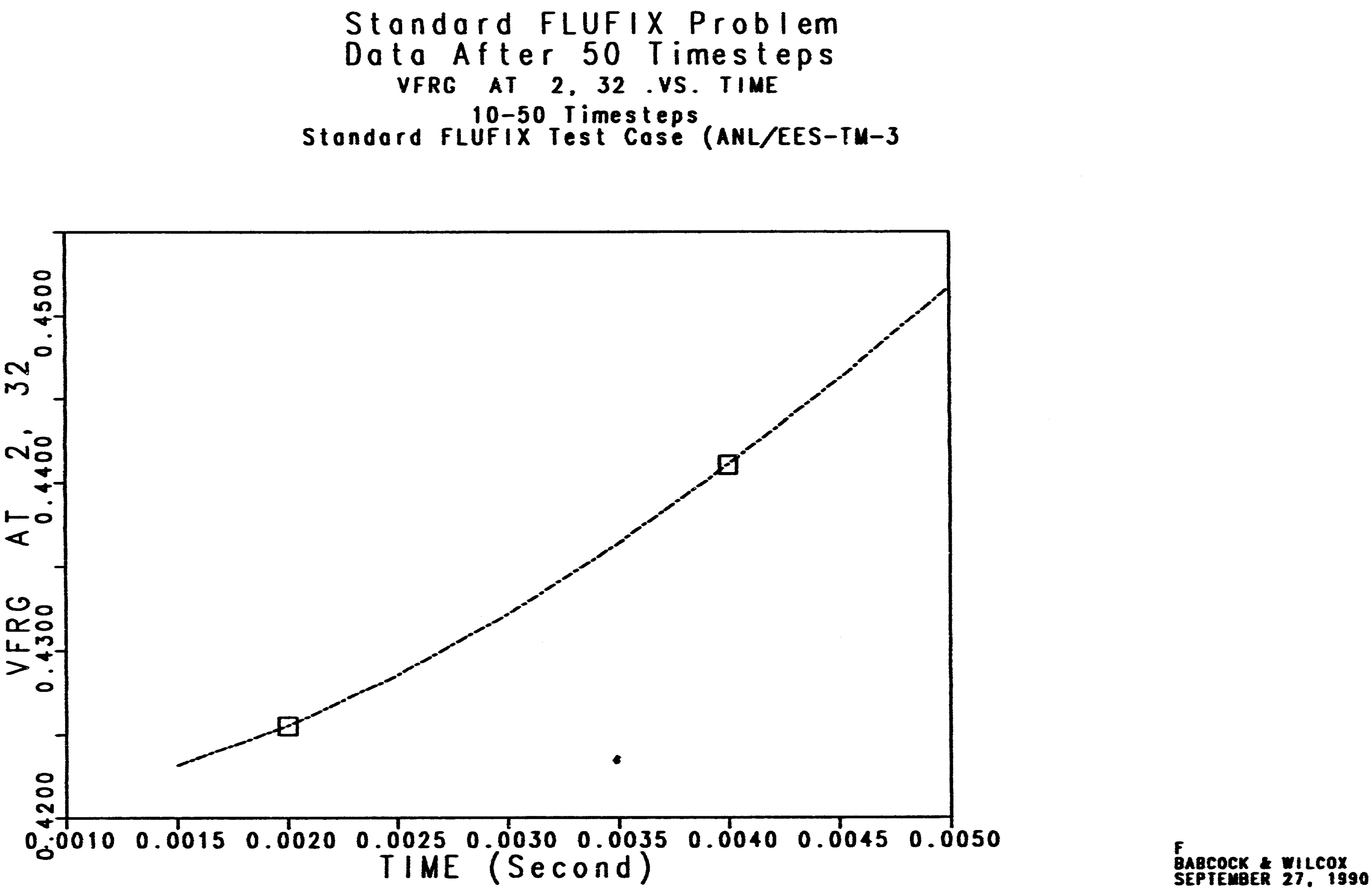


Figure 6-6. Gas void fraction above the inlet jet from .001 to .005 seconds, standard FLUFIX problem.

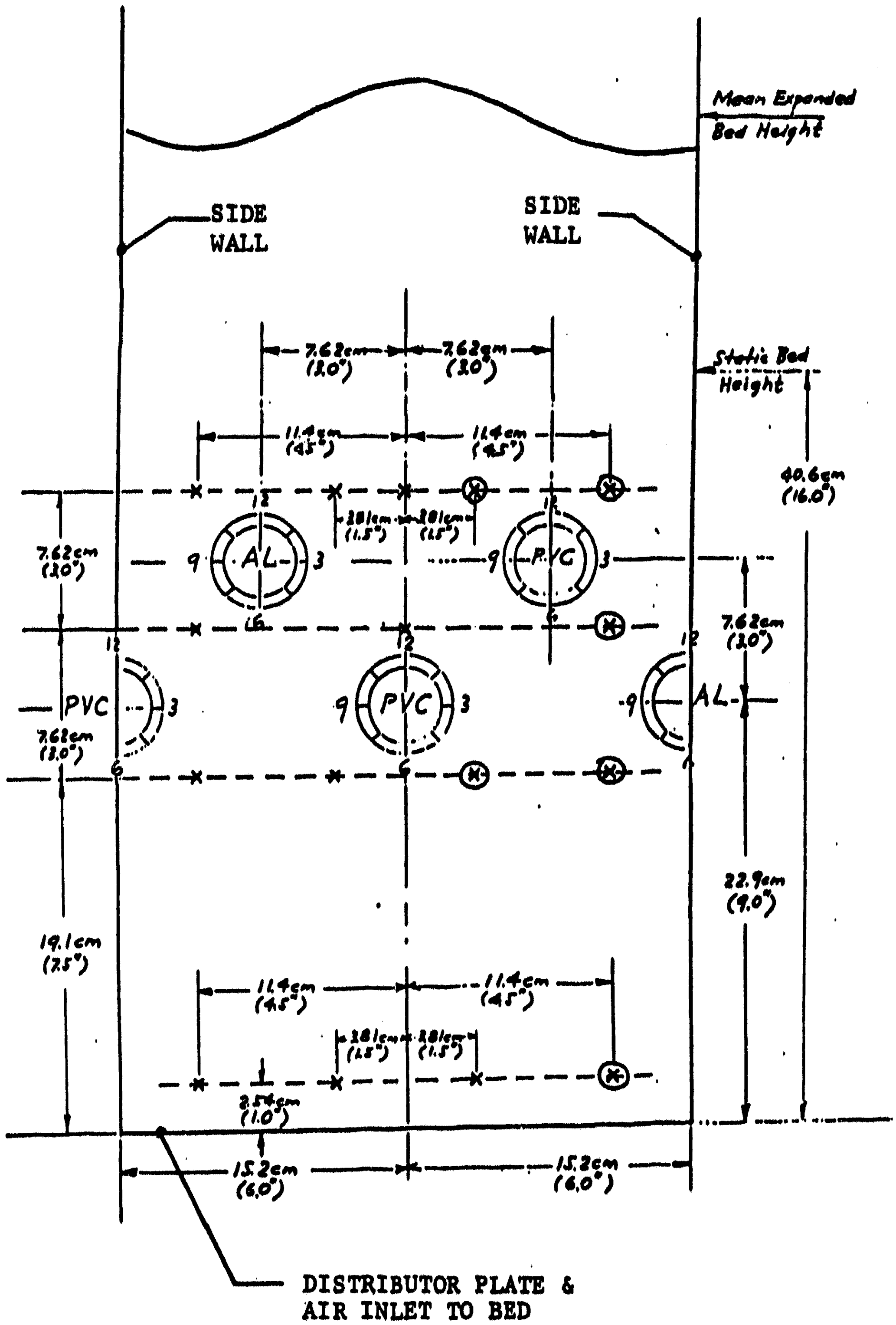


Figure 6-7. 3-D, 12-in. square fluidized bed tested in the Computer-Aided Particle Tracking Facility (CAPTF) at the University of Illinois.

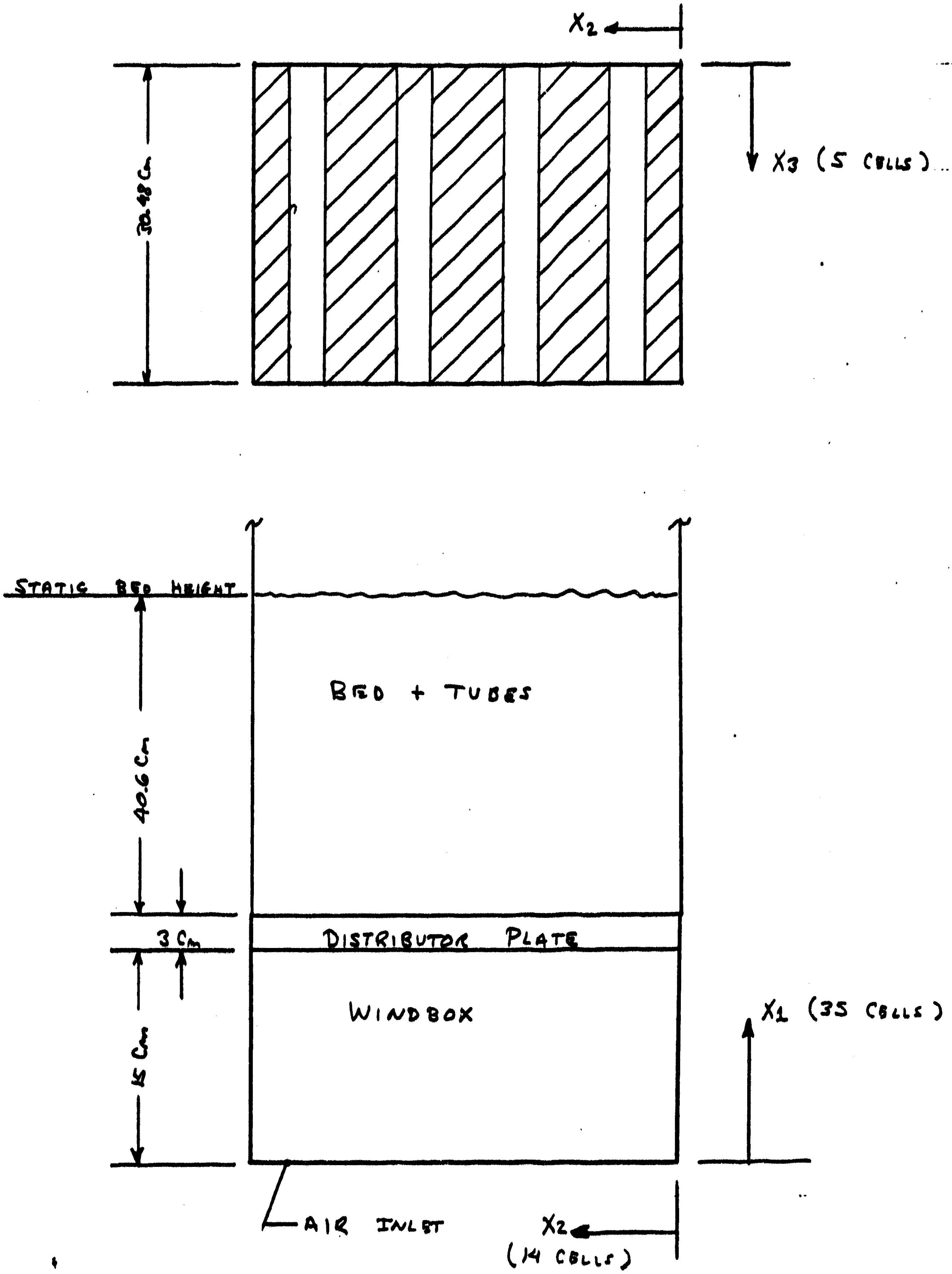


Figure 6-8. FORCE2 geometry model of the 3-D bed tested in the CAPTF.

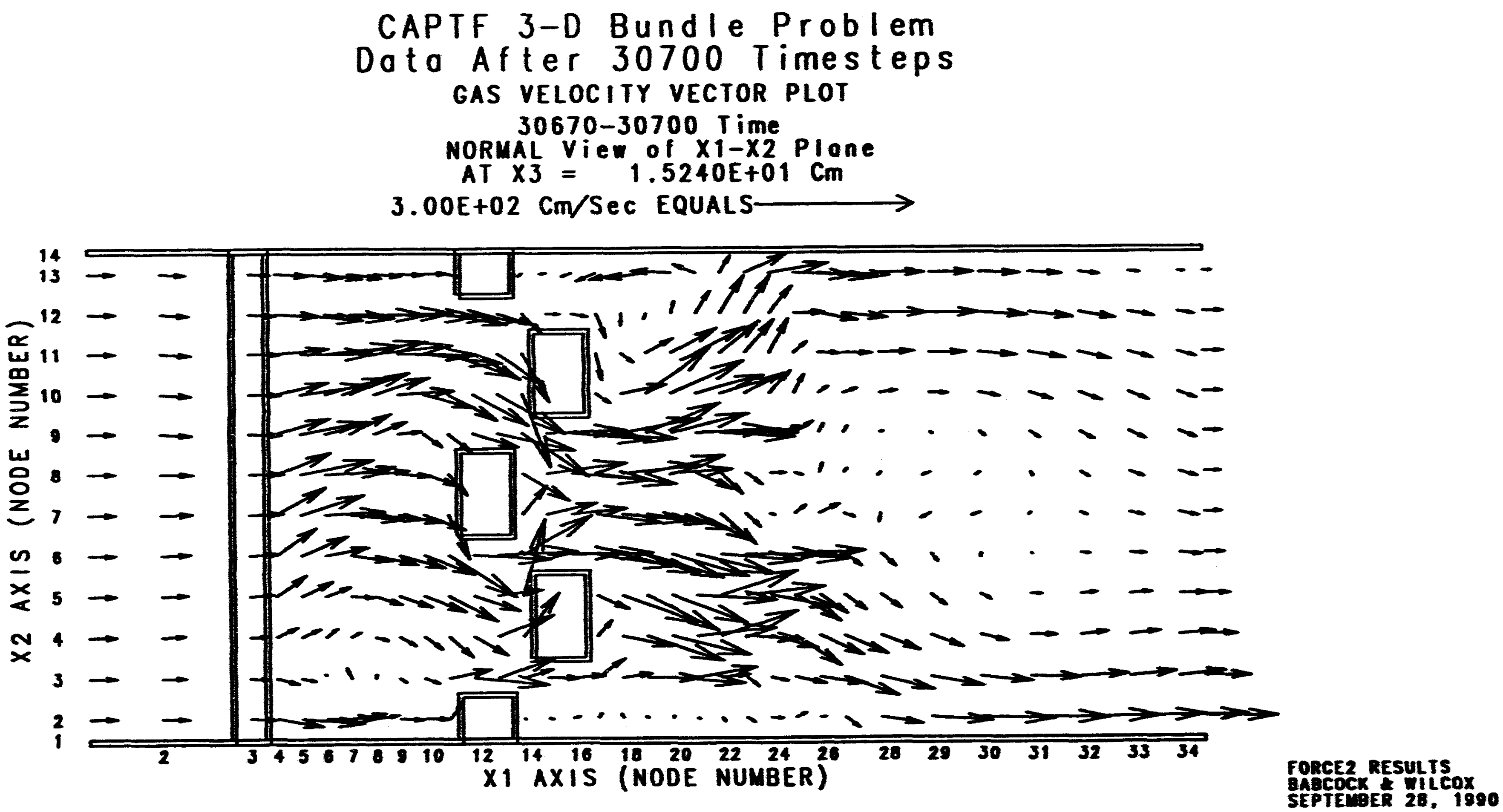


Figure 6-9. Gas velocity vectors at the center plane of the bundle after 30,700 timesteps, CAPTF 3-D bundle problem.

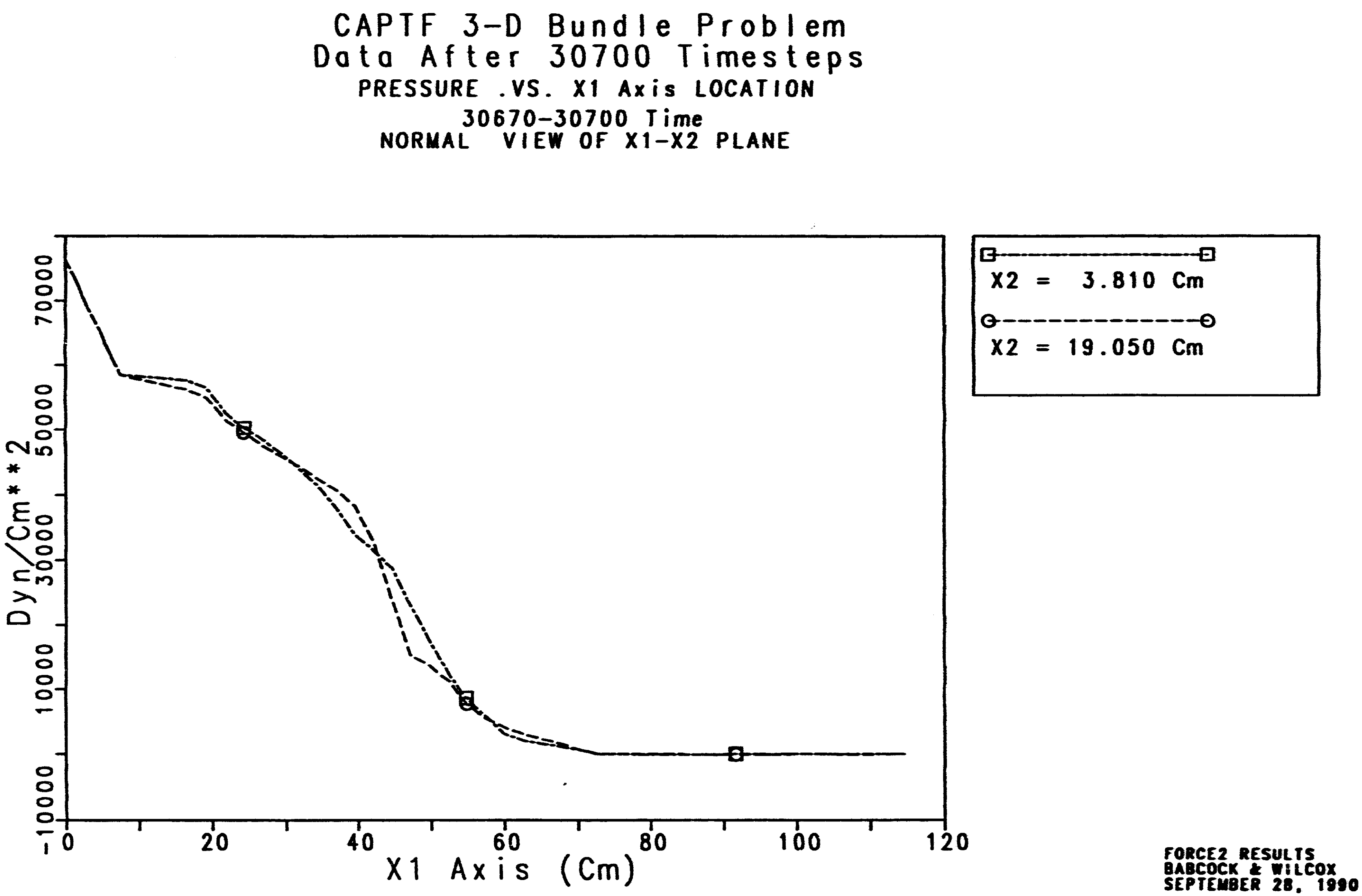


Figure 6-10. Pressure profiles at the center plane of the bundle after 30,700 timesteps, CAPTF 3-D bundle problem.

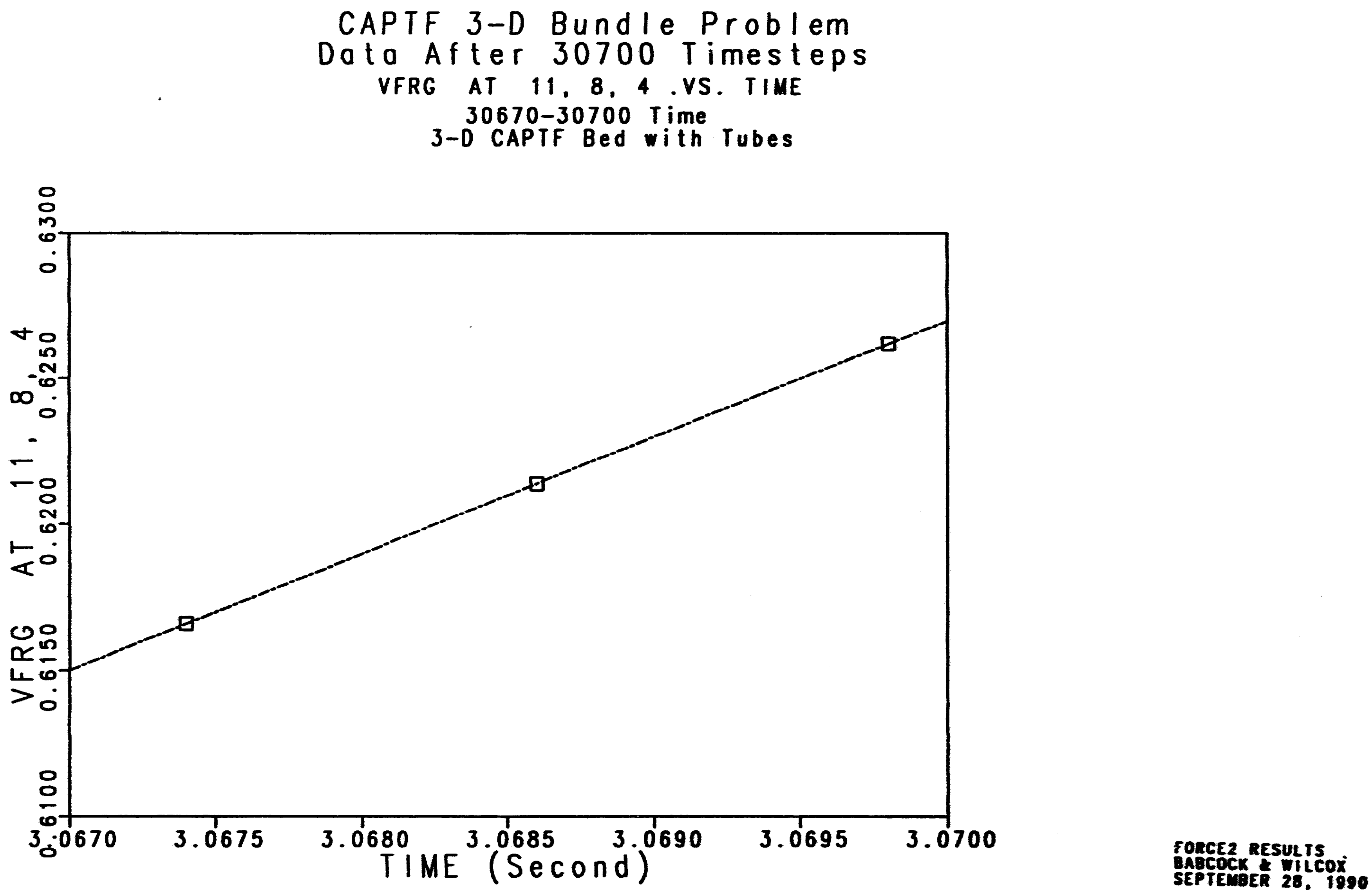


Figure 6-11. Gas void fraction at location (11,8,4) from time 3.067 to 3.070, CAPTF 3-D bundle problem.

7. DETAILED USER'S REFERENCE

7.1 OVERVIEW

This section contains a detailed user's reference for running FORCE2 and the post-processor. Command procedures for executing both programs on a Sun 4/260 super-workstation and VAX are provided. FORCE2 writes two binary files (field data and time data files) that are used by the post-processor. FORCE2 also writes binary files for restarting a run and for developing input to the ANL erosion model. FORCE2 output includes tabular data for selected operating conditions.

The post-processor converts FORCE2 output into graphical form (as plots on a graphics terminal or printer input that can be used to create hardcopy plots).

7.2 EXECUTING FORCE2

7.2.1 Running FORCE2

The communication between FORCE2 and its files is depicted in Figure 7-1. The Input Data File and Input Restart File are read by FORCE2. All the others, with the exception of the Runtime File, are written by the program. Information is both read from and written to the Runtime File.

The input data file is the primary channel of communication between FORCE2 and the user. All information contained in the input data file is stored on the output restart file after the first run is made. Therefore, the data contained in the input data file should only be specified for the first run. Restart runs only require restart statement, flow paragraph and control paragraph.

Restarts are accomplished using the input restart file, which is created as the output restart file on the first run. It is then reassigned and becomes the input restart file on the next restart run.

The FORCE2 output listing contains an echo printout of the input data file. It also redisplays the input data and other defaults after the input data file is processed and contains a printout of:

- o blocked cells and cell faces
- o cell-by-cell initial conditions and boundary conditions
- o residuals of all equations after each iteration when performing a steady run
- o cell-by-cell results for each field variable

Two binary files are written for the graphical post-processor. One contains all the field variables at specified times during a transient or at specified iterations during a steady run. The other contains field variables at specified locations in the domain at selected times during a transient. This second file is written only during a transient simulation. The predictions on both files are converted to graphical output by the post-processor. The times, timesteps, or iterations at which data is written to these files are specified in the FORCE2 input data file.

The erosion model data file contains hydrodynamic predictions (velocities, void fractions, and pressure) throughout the flow field at specified times during a transient simulation. This data can be used to develop hydrodynamic input for the ANL erosion model.

The log file is an ASCII file that is written periodically during both transient and steady simulations. It contains gas mass residuals (transient solution) or solids inventory (steady solution) that is used to monitor the solution.

The runtime file can be used to control execution of FORCE2. It is optional. Information is read from and written to this file. It can be edited during the run to stop execution of FORCE2.

Table 7-1

FORCE2 FILES AND ASSOCIATED FORTRAN LOGICAL UNITS

FORCE2 File	Fortran Logical Unit
Input Data	5
Input Restart	1
Output Restart	2
Output Listing	6
Post-Processor	
Field Variables	69
Time Data	70
Erosion Model Data	71
Runtime	11
Log	20

The Fortran units assigned to each of these files are listed in Table 7-1.

Command procedures for executing the program on a Sun 4/260 and VAX are given in Appendix A. These procedures make the appropriate file assignments and run FORCE2. They provide an outline for procedures that may be needed for other computer systems.

7.2.2 Restarts

A flow problem is continued using restart data files created by FORCE2 at the end of the last run. If the run is a restart, FORCE2 reads the data from the restart data file and subsequently reads the input data file. Therefore, any problem updates in the input file overwrite the restart data. When the continued run is finished, FORCE2 makes a new restart file. A restart file contains

- o problem description data organized on the file according to FORCE2 input paragraphs

- field variables such as velocity, pressure, void fraction

Once a problem has been set-up and checked, only a small amount of data is required for restarts. Typically, a RESTART statement, flow paragraph and control paragraph are required as shown below.

```
$
$       Typical  FORCE2  Restart  File
$
RESTART  SAVE
TITLE  Standard  FLUFIX  Problem

FLOW
TRANSIENT
$
$  End of FLOW  Paragraph

CONTROL
MAXIMUM TIMESTEPS  20
$
$ Print Out Control
$
PRINT  FINAL  U1G
PRINT  FINAL  U1S
PRINT  FINAL  U2G
PRINT  FINAL  U2S
PRINT  FINAL  VFRG
$
$  End of CONTROL Paragraph
```

This Input file is used to continue a transient simulation. The RESTART SAVE statements control the run by restarting it and saving the results in the output restart file. The transient mode is specified by the statement TRANSIENT in the FLOW paragraph. This statement is optional because the execution mode (transient or steady) is included in the restart file. The maximum number of problem timesteps is specified in the CONTROL paragraph along with the variables to be printed in the FORCE2 output listing.

7.3 POST-PROCESSOR

7.3.1 Overview

The post-processor is a FORTRAN program capable of producing a variety of plots from 3-dimensional FORCE2 calculations. Input from the FORCE2 post-processor files and interactive user input is used by the post-processor to define the graphics output. The program produces output interactively on a graphics display terminal or writes graphical output for a variety of hardcopy plotting devices. A general diagram of the post-processor showing the input and output is presented in Figure 7-2.

The basic plots that can be generated by the post-processor are:

- o Geometry including control volumes
- o Contour plots
- o Profile plots
- o Velocity vector plots
- o Streamlines in 2-D applications
- o Variable vs time plots in transient applications

For three-dimensional geometries, plots can be made in any of the three principal directions including reverse views of these principal directions. Figure 7-3 presents an overview of the post-processor structure showing the initial data input phase followed by the main menu and possible plot types.

An example of a post-processor plot is shown in Figure 7-4 with the major sections of the plot labeled.

PROGRAM CONVENTIONS

During interactive input the program displays information from the FORCE2 data to allow the user to better answer the program request. This information can consist of data ranges, suggested responses and default responses. The conventions used to display these are shown below.

: A user response is expected

( ) Suggested or possible user responses are shown inside the parentheses

[ ] Default responses are shown in brackets and are normally shown with the prompt. The default is chosen by pressing the <RETURN> key only (abbreviated as <cr>).

Many prompts allow more than one selection or entry; in this case, the program expects entries to be separated by commas. In all cases the program will accept a single entry.

Other prompts expect a range of input such as the node range selection. The user may define a range of nodes by preceding the second entry of a pair with a dash (minus) sign. For example, to select node ranges from 2 through 15 and also 20 through 29, the user would respond as shown below:

Select node range for X1 Axis (1,-30) [1,-30]: 2, -15, 20, -29

The menu options for the different plot types are usually toggle inputs: selecting the numbered item, switches the selection from YES to NO or vice versa. Options other than YES or NO are possible for some menu items. Consider a portion of the contour menu shown below.

```
Contour options
3.  Plot labels            [YES]
4.  Contour labels         [NO]
5.  Contour lines   :
       Straight or Curved  [STR]
Enter option or <RETURN> to end:
```

Note that the plot labels will be plotted, the contour labels will not be plotted and contour lines are straight. If the menu option 4 is entered, the contour labels will be selected for plotting and the menu redisplayed. Entering a <RETURN> will end the menu input and will prompt for the next input following the menu.

In one menu item of the profile menu, options for multiple plots are chosen from a second menu.

At any time during the input phase the user can interrupt the program, canceling the previous input by hitting a ^Z (holding the Control key while pressing the Z key). Usually the user interrupt function returns you to the main plot selection menu.

When plotting data on an interactive graphics terminal, the program expects the <RETURN> key to be pressed to continue on to the next graph.

MANUAL CONVENTIONS

Major headings or selections are shown in bold print. Information displayed by the program and examples of user responses are shown in small bold print.

7.3.2 Running the Post-Processor

The method of running the post-processor will depend on user preferences and computer system. The approach described here is used by B&W to run on a Sun 4/260 super-workstation. It is not the only way of executing the program. However, it is recommended that the B&W approach be adopted and modified for the user's particular computer system.

The post-processor is best run using an interactive command procedure. The procedure sets up the files to be read and/or written by the processor and then runs the post-processor program. The command procedure (shellscript) listed in Table A-4 of Appendix A is used by B&W.

Graphical output that is used to make hardcopy plots can be developed in one of the following ways:

Interactive Sequence

The post-processor is used to interactively display problem geometry, FORCE2 predictions, etc., on a graphics terminal. In this mode, an interactive graphics terminal (such as a Tektronix 4014) is selected as the GKS device in the GKS menu. Each time the post-processor is run in this interactive mode, a session file is created that contains a list of all user entries. After the user has displayed the desired plots at his terminal, the appropriate GKS plotting device and output file name are added to the associated session file. This modified session file is then used to run the post-processor. This final step generates a graphics file that will produce hardcopy plots on the chosen GKS device.

Non-Interactive Sequence

The post-processor is run from a non-graphics terminal. In this mode, a plotting device is selected in the GKS menu. The post-processor creates a graphic file that will produce hardcopy plots on the chosen device. The only difference between this mode and the Interactive Sequence, described above, is that the plots are not displayed on the terminal (the post-processor output device is a plotter).

The command procedure used by B&W to execute the program is given in Table A-4 of Appendix A. This unix script defines the files that will be read by the program. The logical names for these files are listed in Table 7-2 along with whether the file is optional or required to execute the program. Finally, the data used to develop profile plots will be written on Fortran Unit 12 with the name PP_PLOTOUT if the file output option is requested. This feature allows the user to port the profile data to other plotting software if desired.

Table 7-2

LOGICAL FILE NAMES FOR THE FILES READ BY THE POST-PROCESSOR

Logical File Name	File	File Type	Requirement
PPF	Field variable data written by FORCE2 on FORTRAN Unit #69	Binary	R*
PPT	Variable vs time data written by FORCE2 on FORTRAN Unit #70	Binary	O*
CONVUNIT	Unit conversion data file	ASCII	R
PP_BOUND1	Geometry data file	ASCII	O
PP_SESSION	Session file	ASCII	O

*R = Required; O = Optional

The session file (PP_SESSION) convention should be noted. If a session file exists, the post-processor will use it. Otherwise, a session file is created. Consequently, for interactive operation of the program, there should be no session files present.

7.3.3 Input Phase

The program uses data files for unit conversion, boundaries of the geometry, and FORCE2 predictions. The program saves the user's input in a session file for later use. With B&W's approach, the command procedure establishes default file names for the post-processor and then runs the processor. This initial step is not absolutely necessary because the post-processor also prompts the user for the FORCE2 file names.

GKS MENU

The post-processor is a FORTRAN program that creates graphical output for a variety of devices using files written by FORCE2. The type of device is listed in the GKS Menu. The request, with available options, is shown below.

```
---- GKS DEVICE SELECTION ----
<Enter> User defined workstation
   2501 4014 TEKTRONIX Terminal
   3100 4107 TEKTRONIX Terminal
  10300 CGM Metafile, Clear Text
   1103 HP 7475 plotter (size A)
   1325 HPLaserJet, Portrait  (150 dpi)
   1328 HPLaserJet, Landscape (150 dpi)
   1900 PostScript, Portrait  (8.5x11.0)
   1901 PostScript, Landscape (8.5x11.0)
   5300 Suntools device, grey

  Enter device type:
```

This menu may have to be modified or extended, depending on the user's GKS package and plotting hardware. This menu is displayed in subroutine DEVICE.

The post-processor has been optimized to work with the 4014 terminal (2501) and the postscript, landscape printer (1901). Other devices may function but do not necessarily present the graphical output accurately. A list of additional, user-defined, devices supported by the post-processor can be found in the Grafpak's GKS drivers' documentation.

Non-interactive output is normally written to a file and then printed after the post-processor session is complete. The filename is requested after a non-interactive device type is entered. If the filename does not have an extension, then an extension is provided by the post-processor. The extensions provided are dependent upon the device type and are listed below.

Type	Ext.	Type	Ext.	Type	Ext.
10300	cgm	1325	hp0	1900	ps0
1103	hp6	1328	hp1	1901	ps1

UNITS CONVERSION

requests the name of a file containing the variable unit name and units conversion coefficients. The prompt for this filename follows:

Enter name of unit conversion file [CONVUNIT.DAT]:

If a file with the name CONVUNIT.DAT exists, depress the <RETURN> key. Any unit conversion file can be assigned with the system logical name 'CONVUNIT' and the default will be valid.

A sample of a unit conversion file, CONVUNIT.DAT, is shown in Figure 7-5 containing the FORCE2 variables and their default unit names. Aside from the FORCE2 variable names in the file, three other names are defined. These are:

GEOM - geometry units used to define X_1, X_2, X_3 locations
MFLX - mass flux vector (units name only)
STRM - streamline (units name only)

Note that the first line of the units conversion file is not used by the program.

The user must respond to the command:

Update unit conversion data (Y or N) [N]?: Y

This command will change either the unit conversion coefficients, the unit name or may add a new unit name to the conversion file. This may be done interactively while running the post-processor.

When this prompt is answered with a Y, then the following prompt appears.

Enter variable name to be changed, <RETURN> for list,
NEW to add variable, or END to finish :

If a <RETURN> is entered, then the following list of variables is displayed.

Name	Unit	*** A ***	*** B ***	Name	Unit	*** A ***	*** B ***
U1G	CM/SEC	1.00E+00	0.00E+00	VFRG		1.00E+00	0.00E+00
U1S	CM/SEC	1.00E+00	0.00E+00	VFRS		1.00E+00	0.00E+00
U2G	CM/SEC	1.00E+00	0.00E+00	RHOG	G/CM**3	1.00E+00	0.00E+00
U2S	CM/SEC	1.00E+00	0.00E+00	RHOS	G/CM**3	1.00E+00	0.00E+00
U3G	CM/SEC	1.00E+00	0.00E+00	GEOM	CM	1.00E+00	0.00E+00
U3S	CM/SEC	1.00E+00	0.00E+00	STRM	G/CM/SEC	1.00E+00	0.00E+00
P	DYN/CM**2	1.00E+00	0.00E+00	TIME	SECOND	1.00E+00	0.00E+00

Enter variable name to be changed, <return> for list,
NEW to add variable, or END to finish :

To update a variable the user enters the name of the variable and replies to the subsequent prompts. Changing the geometric units, from centimeters to inches, is shown below.

Enter unit description for GEOM [CM] : in
Enter conversion factor A, (A*value+B) [1.000E+00]: .394
Enter conversion factor B, (A*value+B) [1.000E+00]:
Enter variable name to be changed, <RETURN> for list,
NEW to add variable, or END to finish: END

Unit conversion coefficients are applied to a variable ϕ according to the equation:

$$\phi_{\text{new units}} = A \cdot \phi_{\text{old units}} + B$$

where A and B are coefficients read from the units conversion file. GEOM unit conversion applies to the following geometric data:

X1, X2 and X3 location, and
the geometry boundary data.

SESSION FILE

A session file is created each time the user interactively runs the post-processor and contains a list of all user entries. This file can be used to rerun a previously-defined set of plots by assigning it the logical name PP_SESSION and rerunning the post-processor. It should be noted that if a session file exists with the name-PP SESSION, the post-processor will use it. To develop hard copies of the plots, the session file is modified by i) changing the GKS output device (from a graphics terminal to a hard copy plotting device) on the first line of the file and ii) adding the output file name after the GKS device selection. Session files that were developed during interactive operation of the post-processor (GKS output device 2501, a 4014 Tektronix Terminal) and subsequently modified to produce a file for printing on the postscript, landscape printer (GKS output device 1901, PostScript, Landscape) are shown below. The post-processor was run with the modified session file to create the postscript file, p33x14.psl.

INITIAL SESSION FILE
```
--------------------
2501


p33x14.ppf
Standard Problem
p33x14.ppt

Plot Title #1
Plot Title #2
g



.2

e
```

MODIFIED SESSION FILE
```
---------------------
1901
p33x14


p33x14.ppf
Standard Problem
p33x14.ppt

Plot Title #1
Plot Title #2
g



.2

e
```

FORCE2 POST-PROCESSOR FILES

Two files containing the hydrodynamic predictions for the post-processors are written by FORCE2 (written to FORTRAN Units 69 and 70, see Table 4-1). Up to three sets of files can be accessed by the program simultaneously. The program prompts the user for the name of the file containing the field variables.

Enter filename of FORCE2 data [PPF] :

The program also prompts the user for the title for the data file and prints it on the graphics output.

Enter data file identification (16 chars.) :

The program also prompts the user for the name of the file containing the time data (variable vs time).

Enter FORCE2 time data file name [<Enter> = none] :

If a time data file has not been created, enter a carriage return and execution will continue.

When reading the field data, the program will display the number of FORCE2 variable data items as they are read.

When more than one field data file is used, a second file can be read by answering the next prompt with a "Y". In this case the restart prompts will be displayed for the second file. A maximum of three sets of restart data can be used during a session. These files must be FORCE2 runs with the same type of geometry, i.e., all 2-D Cartesian, etc.

BOUNDARY DATA

The boundaries and other key features of the geometry can be defined in a geometry data file. It contains the selected views and should be named or assigned the logical name, PP_BOUND1. A description of the data file follows.

BOUNDARY DATA FILE The boundary data file consists of two major parts. The first defines the view and the amount of boundary data, while the second defines the geometry for each plane or group of planes in the selected view. The geometry within a specific plane is described as one or more continuous sections consisting of straight lines between defined vertices.

The vertices of a segment are normally defined in a clockwise direction. The definition of a vertex consists of a boundary type and the X and Y coordinates. The boundary types available are shown in the table below.

Code	Line Type	Purpose
1	Boundary line	Normal boundary
2	No line	Inlet
3	No Line	Outlet
4	Dash-dot	Symmetry line
0	Solid line	Special
5-10	Zeta line types #4 to #9	Special

A venturi geometry and the data file are shown in Figure 7-6. The coordinates of the points are tabulated in Table 7-3. For this example the geometry is divided into two sections: section 1 defines the venturi while section 2 defines the flow restriction.

The first line consists of four values on a single line defining:

1) the view direction

1 - X2-X3 plane
2 - X1-X3 plane
3 - X1-X2 plane

2) The number of planes or group of planes for which unique boundaries are defined.

3) The maximum number of sections within any group.

4) The maximum number of vertex pairs with any section.

The next group of data consists of lines defining a single section of geometry. The first four entries define

1) The beginning plane for this section

2) The ending plane for this section. For 2-D geometries both 1) and 2) would be equal to 1.

Table 7-3

COORDINATES OF POINTS IN VENTURI GEOMETRY (FIGURE 7-6)

Point	X1	X2
1	0.0	0.0
2	0.0	0.0508
3	0.0525	0.0508
4	0.1057	0.0325
5	0.1271	0.0316
6	0.3085	0.0508
7	0.4572	0.0508
8	0.4572	0.0
9	0.0	0.0
10	0.1033	0.0
11	0.1031	0.01017
12	0.1312	0.0098
13	0.3036	0.0
14	0.1033	0.0

3) The section number, usually numbered consecutively from one.

4) The number of vertex pair entries.

The boundary is defined by line segments, described by the boundary type and the X and Y vertices of the segment.

Boundary data files for the two sample problems (see Section 6.0) are given in Appendix B.

LOGO AND TITLES

Each plot contains a 2-line title and logo placed in the lower right corner (three lines of text consisting of analysis code name (FORCE2), the company or division name and the current date). The user is prompted to enter the Company or Division name (25 characters maximum) and the two title lines as follows:

Enter Division Name [BABCOCK & WILCOX] :

Enter plot title #1 (40 chars) :
#2 (40 chars) :

Titles should be entered in capital letters (for graphics terminal display).

7.3.4 Main Menu

The main plotting menu allows selection of the field data to be plotted and the desired plot type.

Main Plotting Menu

=====================================

E End (Terminate) Program
P Profiles
N New Data Set
G Geometry
C Contours
V Velocity Vectors
T Variable vs Time
S Streamlines
U Update Units
Enter Selection :

The streamline option will appear only for 2-D problems.

7.3.4.1 General Input. The view, case, range and scale selections for plots are discussed first. A description for each plot type (profile, geometry, etc.) listed in the main plotting menu follows:

VIEW

applies to all but axisymmetric plots. The user must select the view containing the information to be plotted. A view is defined by its axis labels (i.e., X1-X2), as shown here.

```
Select view
   1.  View of X2-X3 Plane
   2.  View of X1-X3 Plane
   3.  View of X1-X2 Plane
Enter selection [3] :
```

CASE

selects one or more field variable data files to plot. This menu is <u>not</u> displayed when only one file is read.

```
Restart Cases
   1.  CASE 1
   2.  CASE 2
Enter Selections (1, 2) [1, 2] :
```

RANGE

The range input specifies the plane along the view direction and the range of data. For each case that is entered the user chooses the plane number. Up to 20 individual planes may be entered at this time. Separate plots will be made for each entry unless the profile plot option is chosen. For that option, multiple plane entries will usually be plotted on the same graph (see the profile plot option). The program may display the previous entries as the default.

```
Restart case #1 - TEST CASE
View of X1-X2 Plane
Select Plane Locations on X3 Axis (20 max)
     X3 Axis Nodes   5, Length   0.00 to  0.10
Enter Node Number(s) : 3
```

The user is also prompted for the range of data that applies to each of the axes in the selected view. Default values are listed and the user is referred to the section on program conventions.

```
Enter grid (node) range
   Select node range for X1 AXIS (1, -12) [ 1, -12]:
   Select node range for X2 AXIS (1,  -6) [ 1,  -6]:
```

SCALE

is required for all plots except profile plots. The program will prompt the user as follows:

```
Restart case #1_ TEST CASE
   Geometry data for         X1 AXIS   and  X2 AXIS
      Nodes            =       12              6
      Length (min)     =      0.00           0.00
             (max)     =      1.00           0.10
      Default scale    =      8.75          62.50
Enter Scale Factor (inches of paper/unit of length)
   X1 AXIS [    8.75]:
   X2 AXIS [   62.50]:
```

The actual size of the plot is a function of the scale factors entered and the geometry. The user enters the number of nodes and length for each axes and a suggested scale factor to plot the data on the graphics display or with the Zeta plotter on 8-1/2 x 11 paper. The prompt will include the suggested or the previously entered scaling factor.

7.3.4.2 Profiles. Profiles are selected with the response, P, and permit creation of X-Y plots for selected FORCE2 results. Figure 7-7 shows an example of a profile plot identifying portions of the plot specific to profile plots. One or more curves can be drawn per plot. An additional curve can represent:

- o New dependent variable data (variable)
- o Dependent variable data from a second restart file (case)

o Plane - dependent variable data from another location along the axis normal to the view (plane)

o Dependent variable data from another location along the axis perpendicular to and in the plane of the abscissa, X axis (elevation)

Separate plots can be made with one of the four items listed above.

The last option creates a file containing profile plot data.

```
Profile plot options
        1.  Plot legend    [YES  ]
        2.  Plot symbols   [YES  ]
        3.  Multiple plots [NONE ]
        4.  User size      [NO   ]
        5.  File output    [NO   ]
    Enter option or <RETURN> to end :
```

Pressing <RETURN> will plot the selected profile after the required information is entered.

ABSCISSA SELECTIONS

enable the user to choose the abscissa, X, of the selected view.

```
Enter Abscissa for X axis
   1.  X1 AXIS
   2.  X2 AXIS
Select Axis [1] :
```

The abscissa is selected prior to selection of the grid (node) range, which defines: 1) the range of the abscissa and 2) the location(s) along the ordinate for which dependent variable data will be plotted. Multiple entries or a range of nodes will create a separate curve on the same plot for each entry. The prompts are repeated with example input demonstrating usage of the entire X1-axis and selection of node locations 3 and 5.

Enter grid (node) range

Select node range for X1 AXIS (1, -12) [1, -12]: 1, -12

Select node range for X2 AXIS (1, -6) [1, -6]: 3, 5

PROFILE VARIABLE SELECTION

names the dependent variable data to be plotted. A list of the variable names can be displayed by typing a <RETURN>. If more than one variable is entered, a separate curve on the same plot will be created for each entry.

The list of variable descriptions and names (prompts) is shown below. Variable name entries <u>must</u> be in capital letters.

Name	Description	Name	Description	Name	Description
U1G	X1 GAS VEL.	U3G	X3 GAS VEL.	VFRS	SOLDIS VOID FRA
U1S	X1 SOLIDS VEL.	U3S	X3 SOLIDS VEL.	RHOG	GAS MACRO DEN
U2G	X2 GAS VEL.	P	PRESSURE	RHOS	SOLIDS MACRO DE
U2S	X2 SOLIDS VEL.	VFRG	GAS VOID FRACT.		

Enter Data Names, (10 Max) <CR> for list :

PROFILE PLOT LEGEND

allows the user to determine if the profile legend is to be displayed on the plot. The legend is shown in Figure 7-7. The default is YES and plots the legend. Entering a 1 will toggle the default to NO.

PROFILE PLOT SYMBOLS

can be turned-on or -off with this option. Identifying symbols plotted on each curve are shown in Figure 7-7. The default is YES and plots symbols. Entering a 2 will toggle the default to NO.

MULTIPLE PROFILE PLOTS

are created from the multiple plot menu shown below:

```
Multiple Plot Menu
   1.  Case
   2.  Plane
   3.  Elevation
   4.  Variable
   5.  None
Enter option [3] :
```

The profile option produces a single plot containing a line for each case, plane, elevation and variable selected. Multiple plots can be made for each entry of the selected item; case, plane, elevation or variable. For example, if multiple data names are entered and option 4 is selected, separate plots will be created for each variable input: the same X and Y scale range is used for all plots.

Note: elevation indicates entries at node locations along the ordinate axis in the selected plane and view.

Note: some combinations of menu selections will not completely label the plotted lines. This problem occurs when a single but different plane is selected for multiple cases.

PROFILE SIZE SELECTION

permits control of the length of the X and Y axis, and the Y axis range and the title. When option 4, USER SIZE, is selected, the user is asked to enter lengths, ranges and a title as shown below. These prompts are displayed after the plot options menu is complete.

```
User input axis specifications
   Enter X-Axis Length [  7.50] :
   Enter Y-Axis Length [  5.00] :
   Enter Y-Axis Minimum [  0.000E+00] :
   Enter Y-Axis Maximum [    1.09    ] :
   Enter Y-Axis Title [METERS/SEC]:
```

Default values have been set to plot on the Zeta plotter with 8-1/2 x 11 paper or on the graphics screen.

PROFILE DATA FILE OUTPUT writes a data file containing the values used to create the profile data. The file is assigned to unit 12 and named PP_PLOTOUT. This file can be used to plot the FORCE results using other graphics packages. An example of this file is shown in Figure 7-8. The first five lines contain information describing the file and data structure. The remaining lines list the data for each plot.

7.3.4.3 Geometry. Geometry is selected with the response, G, and produces plots of the modeled geometry created by the FORCE2 program. Control volumes, blocked cells and faces and the geometric boundary are plotted. The geometric options menu allows the user to determine the appearance of the plot, the selection of the view, and range and scale. Figure 7-9 shows an example geometry plot that identifies parts of the plot specific to geometry plots. The options for view, range and scale factors for geometry are described earlier in Section 7.3.4.1, General Input. The geometry plots are selected from a menu shown below.

```
Geometry options
   1.  Boundary         [YES]
   2.  Axis Labels      [YES]
   3.  Plot Labels      [YES]
   4.  Grid lines       [YES]
   5.  Node centers     [YES]
   6.  Blocked cells    [NO ]
   7.  Blocked faces    [NO ]
   8.  Invert X axis    [NO ]
Enter option or <RETURN> to end :
```

Entering a <RETURN> will exit the menu.

The units and value of the FORCE geometric data can be modified with the keyword GEOM in the units conversion file. See Section 7.4.3, Input Phase.

BOUNDARY

determines if the geometric boundary will appear on the plot. A YES in the brackets is the default and indicates that boundary data will appear on the plot. A file containing the boundary description is created by the pre-processor and is read by the post-processor. This file contains boundaries for only one view. When geometry for the selected view is not defined, then the default in brackets will display [N/A] and the boundary option is not applicable.

AXIS LABELS

determines if axis labels will appear on the plot. See Figure 7-4 for an example of axis labels. A YES in the brackets is the default and indicates that axis labels will appear on the plot.

PLOT LABELS

determines if the title text will appear on the plot. A YES in the brackets is the default and indicates that plot labels will appear on the plot.

GRID LINES

determines if the grid lines, lines plotted along the control volume faces, will appear on the plot. Figure 7-9 shows an example of grid lines. A YES in the brackets is the default and indicates that grid lines will appear on the plot.

NODE CENTER

determines if the node center markers, a plus sign, will be plotted at the center of each control volume. Figure 7-9 shows an example of node center markers. A YES in the brackets is the default and indicates that node center markers will appear on the plot.

BLOCKED CELLS

determines if blocked cells and other unused control volumes defined in FORCE will be marked. An example of a blocked cell is shown in Figure 7-9. A YES in the brackets indicates that blocked cells will be marked. The default is NO.

BLOCKED FACES

determines if the individual faces of the control volumes, defined as blocked in the FORCE2 program, are marked on the plot. These block faces are defined with a line just inside the cell adjacent to the blocked face. See the example in Figure 7-9. A YES in the brackets indicates the blocked face will be marked. The default is NO.

INVERT AXIS

specifies the view direction of the plot. A normal view of an X-Y plot displays the data viewed along the Z axis toward increasing Z values. A reverse view of the same plot displays the data viewed toward decreasing Z values. Figure 7-10 shows a simplified example of these views. A [NO], the default, indicates a NORMAL view will be plotted. A [YES] indicates a REVERSE view is used.

7.3.4.4 Contours. Contours are selected with the response, C, and produce a contour plot of the named data calculated by FORCE2. User input consists of selecting the appearance of the plot, selecting the contour line spacing and selecting the variable(s) to plot. Figure 7-11 shows an example of a contour plot identifying portions of the plot that are specific to contour plots. The options for view, range and scale factors are described earlier in Section 7.3.4.1, General Input. The contour plot options are selected from a menu, shown below.

```
Contour options
1.  Boundary                    [YES]
2.  Axis labels                 [YES]
3.  Plot labels                 [YES]
4.  Contour labels              [NO ]
5.  Contour lines   :
      Straight or Curved       [STR]
6.  Contour spacing  :
       Value or Number          [VAL]
7.  Invert X axis               [NO ]
Enter option or <RETURN> to end :
```

Pressing <RETURN> along will exit the menu.

BOUNDARY

determines if the geometric boundary will appear on the plot. A YES in the brackets is the default and indicates that boundary data will appear on the plot. A file containing the boundary description is created by the pre-processor and is read by the post-processor. This file contains boundaries for only one view. When geometry for the selected view is not defined, then the default in brackets will display [N/A] and the boundary option is not applicable.

AXIS LABELS

determines if axis labels will appear on the plot. See Figure 7-4 for an example of axis labels. A YES in the brackets is the default and indicates that axis labels will appear on the plot.

PLOT LABELS

determines if the title text will appear on the plot. A YES in the brackets is the default and indicates that plot labels will appear on the plot.

CONTOUR LABELS

determines if markers will be written on the plot to identify selected contours. When contours markers are selected (a YES response), additional data is required to define placement on the graph. This information is input after the contour menu is complete and before the plot is made. The default is NO contours markers.

AXIS identifies the direction to mark the contours, and is used only when CONTOUR LABELS are turned on. If the default option 1 is chosen as shown

```
Enter Axis for Contour Marking
   1.  X1 AXIS
   2.  X2 AXIS
Select Axis [1] :
```

markers will be placed on contour lines that cross node centerlines which are parallel to the X1 axis. If option 2 is selected, the markers are placed parallel to the X2 axis.

RANGE selects the range of node centerlines on which markers will be placed. This option is used only when CONTOUR LABELS are turned on. For example:

> **Enter grid (node) range for contour markers**
> **Select node range for X2 AXIS (1, -21) [1, -21]:**

Selecting the default will place markers along each node centerline, lines 1 to 21.

NUMBER selects the number of contour lines between marked contour lines. This option is used only when CONTOUR LABELS are turned on. The default value is zero

> **Enter the number of lines between marked contour lines [0]:**

If the default is chosen, every contour line crossing the node centerline will be marked. If a value of 2 is entered, every third contour line be marked and two lines skipped.

CONTOUR LINES

plots selected contours within a control volume rectangle using values defined at the corners of the rectangle. Bilinear interpolation . used to determine the path of a contour within this rectangle. The contour plotting will either generate straight lines between the faces of this rectangle or plot curves resulting from the interpolation functions. Experience indicates that straight line segments usually produce better plots.

The options are straight or curved lines. The default [STR] is straight lines.

CONTOUR SPACING

specifies the method of selecting contour lines spacing: a value for contour spacing or the number of lines on the plot. The program prompts for the minimum and the maximum contour values to plot.

VALUE defines the contour spacing. Contours will correspond to equal multiples of this value, starting from the minimum contour value. In the following example, the contour spacing is entered as 50.0 and the default minimum contour (100) is used; therefore contours will correspond to values of 100.0, 150.0 --- 300.0

```
Contour Spacing
   Contour data range - 9.20E+01 to 3.45E+02
Enter contour spacing  [ 30.0] : 50
```

The default value is approximately 10 contour lines per plot. The contour data range displayed is the total range of contour variable data for all cases and ranges specified. The default contour spacing is based on the minimum (D_{MIN}) and maximum (D_{MAX}) data for the selected variable (case and plane) as follows:

$$\text{Spacing} = (D_{MAX} - D_{MIN})/(\text{Number}-1)$$

The default spacing is then rounded before being displayed.

NUMBER selects the number of contour lines on the plot. For example, for 12 contour lines on a plot:

```
Enter the number of lines per plot  [  10] : 12
```

The default is the last value entered or the value 10.

MINIMUM selects the allowable range of variable data over which
MAXIMUM contour lines will be plotted. For example,

Enter minimum contour value [100.00] :

Enter maximum contour value [300.00] :

The minimum and maximum default values are the nearest exact multiples (based on zero) for the contour spacing.

INVERT X AXIS

specifies the view direction of the plot. A normal view of an X-Y plot displays the data viewed along the Z axis toward increasing Z values. A reverse view of the same plot displays the data viewed toward decreasing Z values.

VARIABLE DATA

prompts to enter the name of the variable data to be plotted. A list of the possible variable names is displayed by typing a <RETURN>. If more than one variable name is entered a separate contour plot is created for each entry. For example,

Enter Data Name (<CR> for list): VFRG

or

Enter Data Name (<CR> for list): U1G, U2G, U3G

for separate plots of gas void fraction and gas velocities with the same scale, etc., See profile variable data selection, Section 7.3.4.2, for an example of the variable list. Note that the variable names must be in capital letters.

7.3.4.5 Vector Plots. are selected with the response, V, and produce a two-dimensional plot for the direction and magnitude of the velocity. The user can determine the plot appearance, velocity or mass flux selections and scale factor input. Figure 7-12 shows an example of a vector plot identifying portions of the plot that are specific to vector plots. The options for view, range and scale factors are described earlier in Section 7.3.4.1, General Input. The vector plots options are selected from a menu shown below.

Vector options

1. **Boundary** [YES]
2. **Axis labels** [YES]
3. **Plot labels** [YES]
4. **Vector label** [YES]
5. **Vector :**
   Velocity or Mass Flux [VCT]
6. **Invert X axis** [NO]
7. **Gas or solids** [GAS]

Enter option or <RETURN> to end :

Pressing <RETURN> exits the menu. The unit name for MASS FLUX is assigned in the units conversion file with the MFLX keyword. See Section 7.3.3, Input Phase.

BOUNDARY

determines if the geometric boundary will appear on the plot. A YES in the brackets is the default and indicates that boundary data will appear on the plot. A file containing the boundary description is created by the pre-processor and is read by the post-processor. This file contains boundaries for only <u>one</u> view. When geometry for the selected view is <u>not</u> defined, then the default in brackets will display [N/A] and the boundary option is not applicable.

AXIS LABELS

determines if axis labels will appear on the plot. See Figure 7-3 for an example of axis labels. A YES in the brackets is the default and indicates that axis labels will appear on the plot.

PLOT LABELS

determines if the title text will appear on the plot. A YES in the brackets is the default and indicates that plot labels will appear on the plot.

VECTOR LABEL

determines if a unit vector, a one-inch line, is drawn below the plot label showing the scale factor used to plot the vectors. The default is [YES] and plots the unit vector. When displaying the plot on an interactive graphics device the vector label length may vary.

VECTOR - VELOCITY OR MASS FLUX

allows the selection of the vector type to plot: velocity or mass flux.

INVERT X AXIS

specifies the view direction of the plot. A normal view of an X-Y plot displays the data viewed along the Z axis toward increasing Z values. A reverse view of the same plot displays the data viewed toward decreasing Z values.

GAS OR SOLIDS

specifies the phase.

SCALE FACTOR

requests a factor to scale the vectors. The prompt displays the maximum range of the data. The default scale factor is for a 1/2-inch vector. The scale factor is defined in units/inch. The maximum vector length of any vector is also requested. When displaying the plot on an interactive graphics device the scale factor length may vary.

```
Vector Scale Factor
   Maximum data range - 1.08E+00 METERS/SEC
Enter scale factor (units/inch)  [  2.20  ] :
Vector Maximum Length
Enter length (inches)  [  0.500] :
```

7.3.4.6 Variable vs Time Plots. Plots of selected variables versus time are selected with the response, T. The user selects the variable(s) to be plotted and inputs the appearance of the plot.

Minimum and maximum times are entered after the prompts:

Enter Minimum Time [ ] :
Enter Maximum Time [ ] :

The default values are the minimum and maximum value in the time data file.

VARIABLE SELECTION

The variable(s) to be plotted are selected from the table of variables that was input to FORCE2 in the CONTROL Paragraph using the STORE statement (see Section 4.6.4.2 of the FORCE2 input description). This table is displayed as:

Variable Description for [Title of Run]

1	**Variable #1**	**Location**
2	**Variable #2**	**Location**
3	**Variable #3**	**Location**
.	.	.
.	.	.
N	**Variable #N**	**Location**

Enter Number(s) for Variable (8 Max) :

The variables and their locations are displayed in the format that they were input to FORCE2.

The appearance of the plots is controlled by the Plot Options Menu:

Plot Options

1. Plot legend [YES]
2. Plot symbols [YES]
3. Multiple plots [NONE]
4. User size [NO]
5. File output [NO]

Enter option or <RETURN> to end :

These options are identical to those for the Profile plots (Section 7.3.4.2) and produce the same effects.

A typical time plot is shown in Figure 7-13.

7.3.4.7 Streamlines. Streamlines are selected with the response, S, and produce streamline plots of the gas or solids flow field calculated by FORCE2. User input consists of selecting the phase, the appearance of the plot, and the streamline spacing, the magnitude of the flow rate between streamlines. The magnitude of the flow rate between streamlines is constant. The streamline plot options are selected from the following menu:

Streamline options

1. Boundary [YES]
2. Axis labels [YES]
3. Plot labels [YES]
4. Streamline lines -
   Straight or Curved [STR]
5. Streamline spacing -
   Value or Number [VAL]
6. Gas or solid flow [GAS]

Enter option or <RETURN> to end :

Pressing <RETURN> will exit the menu.

A typical streamline plot is shown in Figure 7-14.

BOUNDARY

This option determines if the geometric boundary will appear on the plot. A YES in the brackets is the default and indicates that boundary data will appear on the plot. A file containing the boundary description is created by the user and is read by the post-processor. When geometry is not defined, then the default in brackets will display [N/A] and the boundary option is not applicable.

AXIS LABELS

This option determines if axis labels will appear on the plot. A YES in the brackets is the default and indicates that axis labels will appear on the plot.

PLOT LABELS

This option determines if the title text will appear on the plot. A YES in the brackets is the default and indicates that plot labels will appear on the plot.

STREAMLINE LINES

Selecting this option plots streamlines as contour lines of flow rate within a control volume rectangle using values defined at the corners of the rectangle. Bilinear interpolation is used to determine the path of a contour within this rectangle. The streamline plotting will either generate straight lines between the faces of this rectangle or plot curves resulting from the interpolation functions. Experience indicates that straight-line segments usually produce better plots.

Options are straight or curved lines. The default [STR] is straight lines.

STREAMLINE SPACING

This option specifies the method of selecting streamline spacing: a value for streamline spacing or the number of lines on the plot. The program also prompts for the minimum and maximum streamline values to plot.

Value defines the streamline spacing. The magnitude of flow rate between lines will be equal to value, starting from the minimum streamline value.

Streamline Spacing
Streamline data range - -3.49E-08 to 9.69E-01
Enter streamline spacing [0.100] :

The default value is approximately 10 streamlines per plot. The streamline data range displayed is the total range of flow rate data for all cases and ranges specified. The default streamline spacing is based on the minimum (D_{MIN}) and maximum (D_{MAX}) data for the selected variable as follows:

Spacing - (D_{MAX} - D_{MIN}) / (Number - 1)

The default spacing is then rounded before being displayed.

Number selects the number of streamlines on the plot.

Enter the number of lines per plot [10] :

The default is the last value entered or a value of 10.

GAS OR SOLID FLOW

The phase for which streamlines will be plotted is selected with this option.

7.3.4.8 New Data Set. A new set of field data is selected with this option. The user enters the response, N. During FORCE2 execution, predictions can periodically be written to the field variable data file. The frequency at which data is written is controlled by input to FORCE2. The data set is selected by entering the desired problem time. For steady execution, problem "time" is defined by dividing the steady iteration number by 1000.

The case number and run title are displayed by:

Data for CASE : Case # --- Run Title

The user then enters the problem time at which new data is selected according to:

Enter time of data to plot t1 to t2 [t3] :

where t1 is the initial problem time of all the field data sets, t2 the final time, and t3 the time of the current data set.

7.3.4.9 Update Units. This option allows the user to modify the unit conversion file from the main plotting menu after the field data and time data files have been read. The user enters the response, U, for this option. Prompts will then appear for modifying the unit conversion file, as described earlier in Section 7.3.3.

7.3.5 Warning Messages

Warning and error messages consist of four types: notes, warning, error and fatal error.

- Note - is informative messages to the user
- Warning - is a message indicating a recoverable error
- Error - is a message indicating a significant error has occurred and in some cases the program will stop

- Fatal Error - a major error has occurred and the program will stop. The warning messages and a brief explanation of each follow.

** NOTE ** (1) END OF BATCH INPUT FILE

The batch session is complete. An end of file has been encountered while reading the session file. See Session File, Section 7.3.2.

** NOTE ** (1) RANGE ERROR, NEGATIVE VALUE .. CHANGED TO +

This note indicates that an illegal negative value has been modified by the post-processor.

** WARNING ** (2) INVALID DATA NAME. RE-ENTER NAME.

The variable data name entered is not a FORCE2 variable.

** WARNING ** (2) DATA RANGE IS ZERO -- PROCESS STOP

The program encountered an illegal zero value in the range routine and will go to the main menu.

** WARNING ** (2) UNABLE TO OPEN BOUNDARY DATA FILE

The geometry data file could not be opened. The post-processor will continue with no geometric data defined.

** WARNING ** (2) VARIABLE NAME NOT IN LIST (Upper case only)

A warning indicating that the variable name entered is not in the unit conversion data file and cannot be updated.

**** ERROR ** (3) ERROR WHEN READING UNIT CONVERSION FILE**

The unit conversion has an error in it.

**** ERROR ** (3) ERROR WHEN READING RESTART FILE**

File input error while reading the field data or time data files.

****ERROR ** (3) UNABLE TO OPEN RESTART FILE FOR INPUT**

The field data file was not found by the post-processor. Either the file does not exist or the correct name has not been entered.

**** ERROR ** (3) RESTART FILE NOT ENDED PROPERLY**

Error in the field data or time data files.

**** ERROR ** (3) THIS OPTION NOT IMPLEMENTED**

A plot type has been chosen that is not implemented in this version.

**** FATAL ERROR ** (4) UNABLE TO OPEN DIRECT ACCESS FILE FOR OUTPUT**

Data storage error. This error is a system problem and should not occur.

**** FATAL ERROR ** (4) ERROR W'ILE WRITING DIRECT ACCESS FILE**

Data storage error. This problem can occur especially when a boundary file has been modified by the user and the first line of data in the file is incorrect. Other problems such as field data or time data file incompatibility can cause this error message to appear.

** FATAL ERROR ** (4) ERROR WHILE READING DIRECT ACCESS FILE

Data retrieval error. This problem can occur especially when a boundary file has been modified by the user and the first line of data in the file is incorrect. Other problems such as restart file incompatibility can cause this error message to appear.

** FATAL ERROR ** (4) UNABLE TO OPEN CONVERSION FILE FOR INPUT

The unit conversion file does not exist.

** FATAL ERROR ** (4) UNABLE TO OPEN CONVERSION FILE FOR OUTPUT

Data storage error. This error is a system problem and should not occur. Unless the unit conversion file name is wrong.

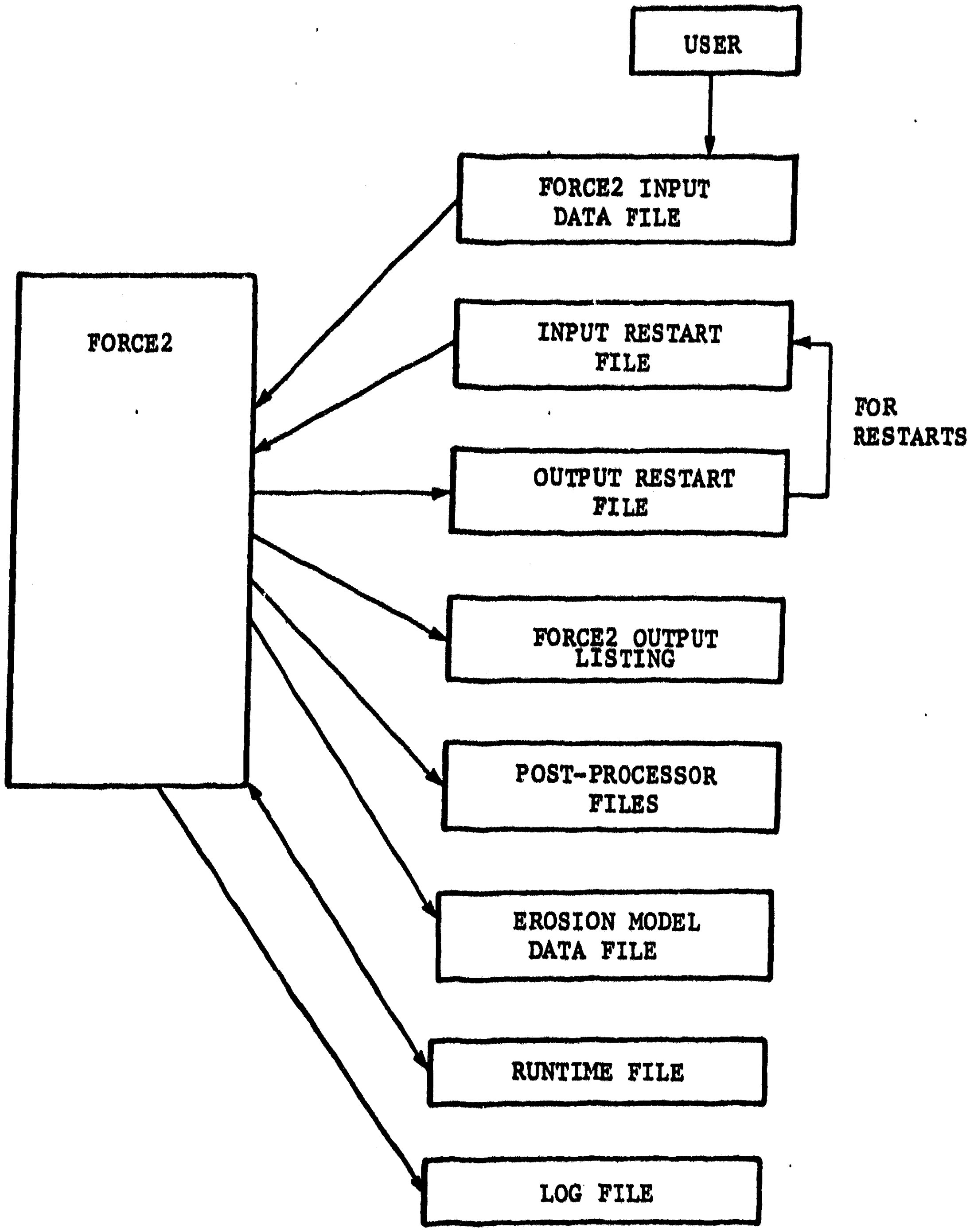


Figure 7-1. FORCE2 file structure.

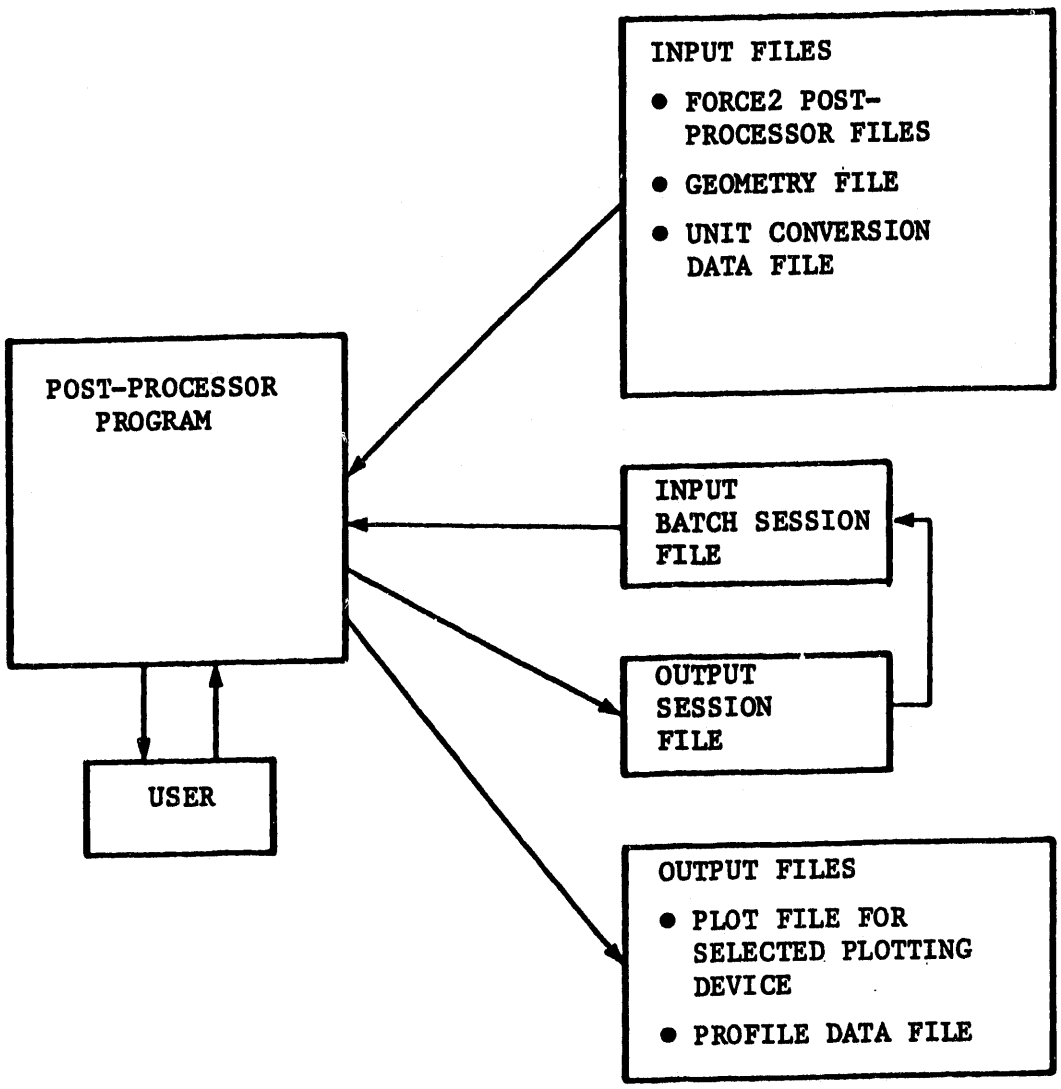


Figure 7-2. Post-processor input and output files.

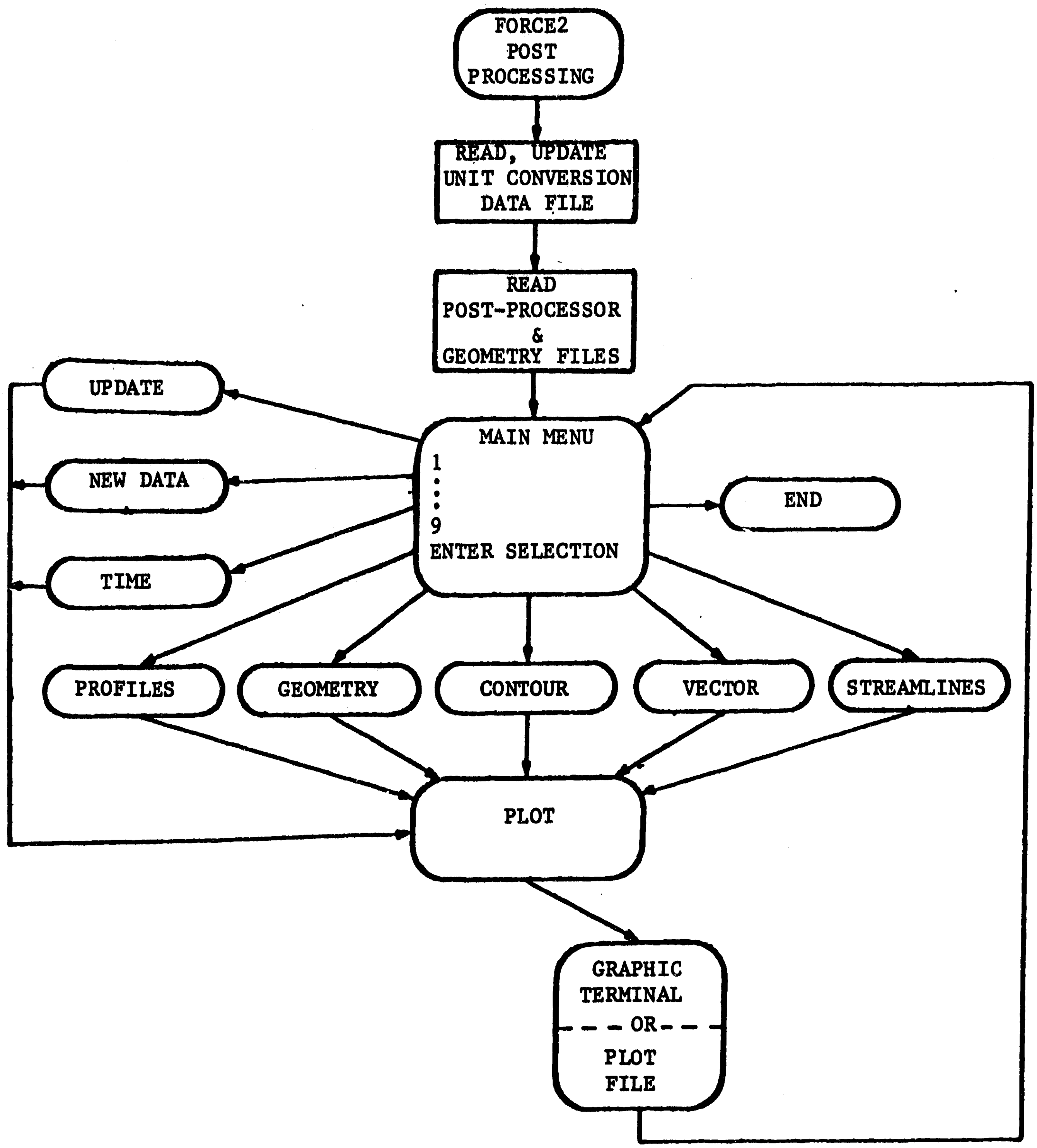


Figure 7-3. Overview of the post-processor program.

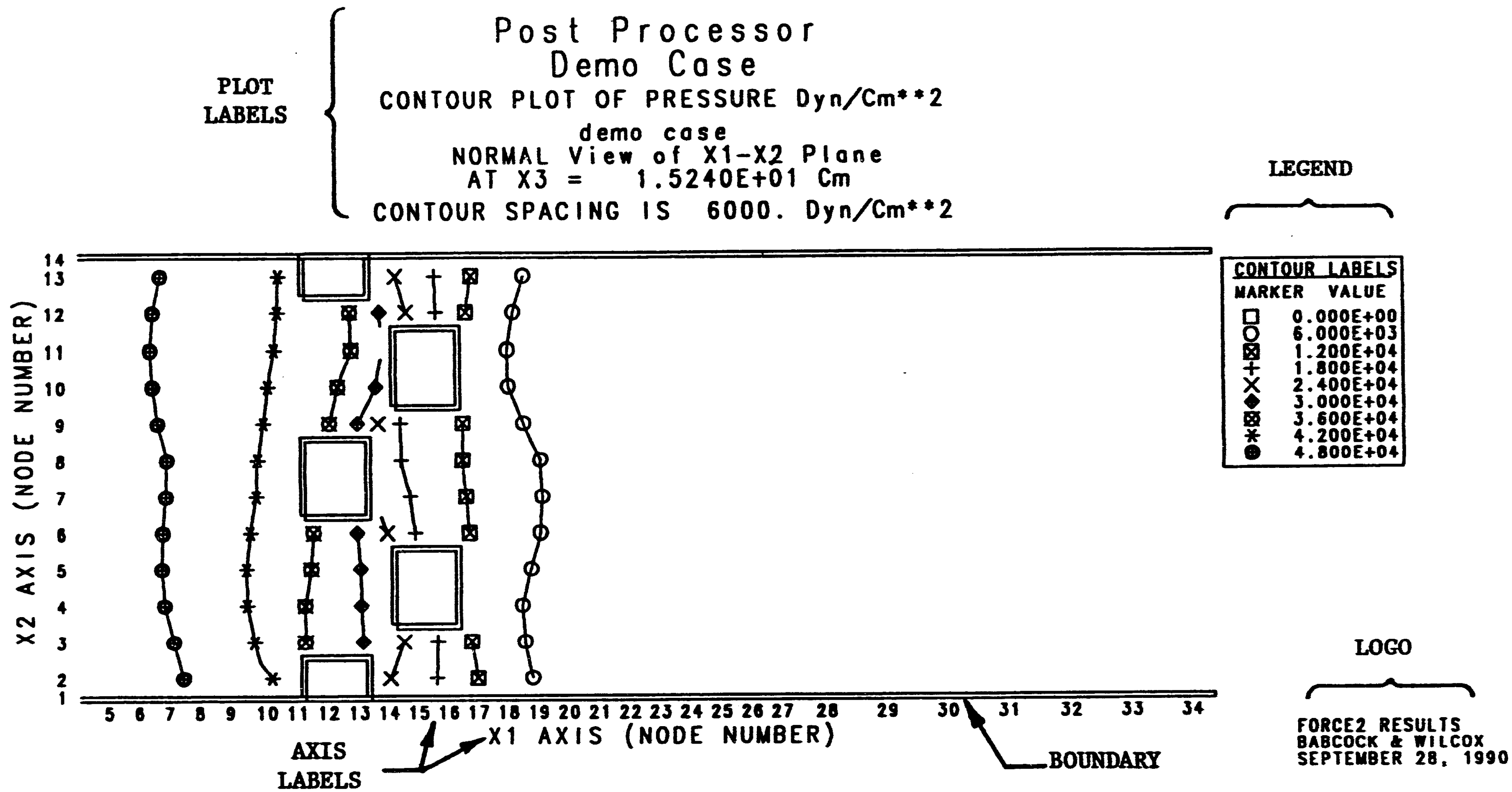
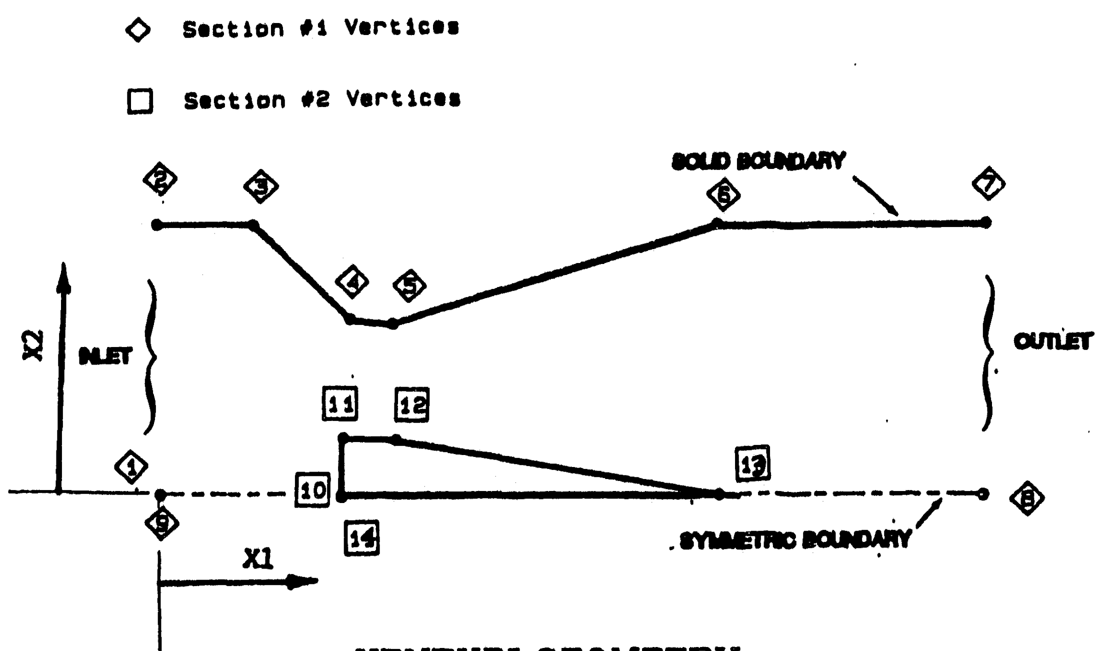


Figure 7-4. Post-processor plot with component parts labeled.

```
name    *  units  *          A          B      (sample line -- first line not read)
U1G     Cm/Sec             1.00       0.00
U1S     Cm/Sec             1.00       0.00       IMMMMMMMMMMMMMMMMMMMMMMM:
U2G     Cm/Sec             1.00       0.00       :  FORCE2 UNITS          :
U2S     Cm/Sec             1.00       0.00       :  CONVERSION FILE       :
U3G     Cm/Sec             1.00       0.00       :                        :
U3S     Cm/Sec             1.00       0.00       :                        :
P       Dyn/Cm**2          1.00       0.00       HMMMMMMMMMMMMMMMMMMMMMMM<
VFRG                       1.00       0.00
VFRS                       1.00       0.00
RHOG    G/Cm**3            1.00       0.00
RHOS    G/Cm**3            1.00       0.00
GEOM    Cm                 1.00       0.00
STRM    G/Cm/Sec           1.00       0.00          STREAM LINE UNITS ONLY
TIME    Second             1.00       0.00          Variable.vs.time plot
MFLX    G/Cm**2/Sec        1.00       0.00          Variable.vs.time plot
```

Figure 7-5. Unit conversion file.



VENTURI GEOMETRY

```
3   1   2   3
    1   1   1   9     2  0.0000  0.0000     1  0.0000  0.0508
                      1  0.0525  0.0508     1  0.1057  0.0325
                      1  0.1271  0.0316     1  0.3085  0.0508
                      3  0.4572  0.0508     4  0.4572  0.0000
                      4  0.0000  0.0000
    1   1   2   5     1  0.1033  0.0000     1  0.1031  0.01017
                      1  0.1312  0.0098     1  0.3036  0.00000
                      1  0.1033  0.0000
```

GEOMETRY FILE

Figure 7-6. Boundary data file for a venturi with a flow obstruction.

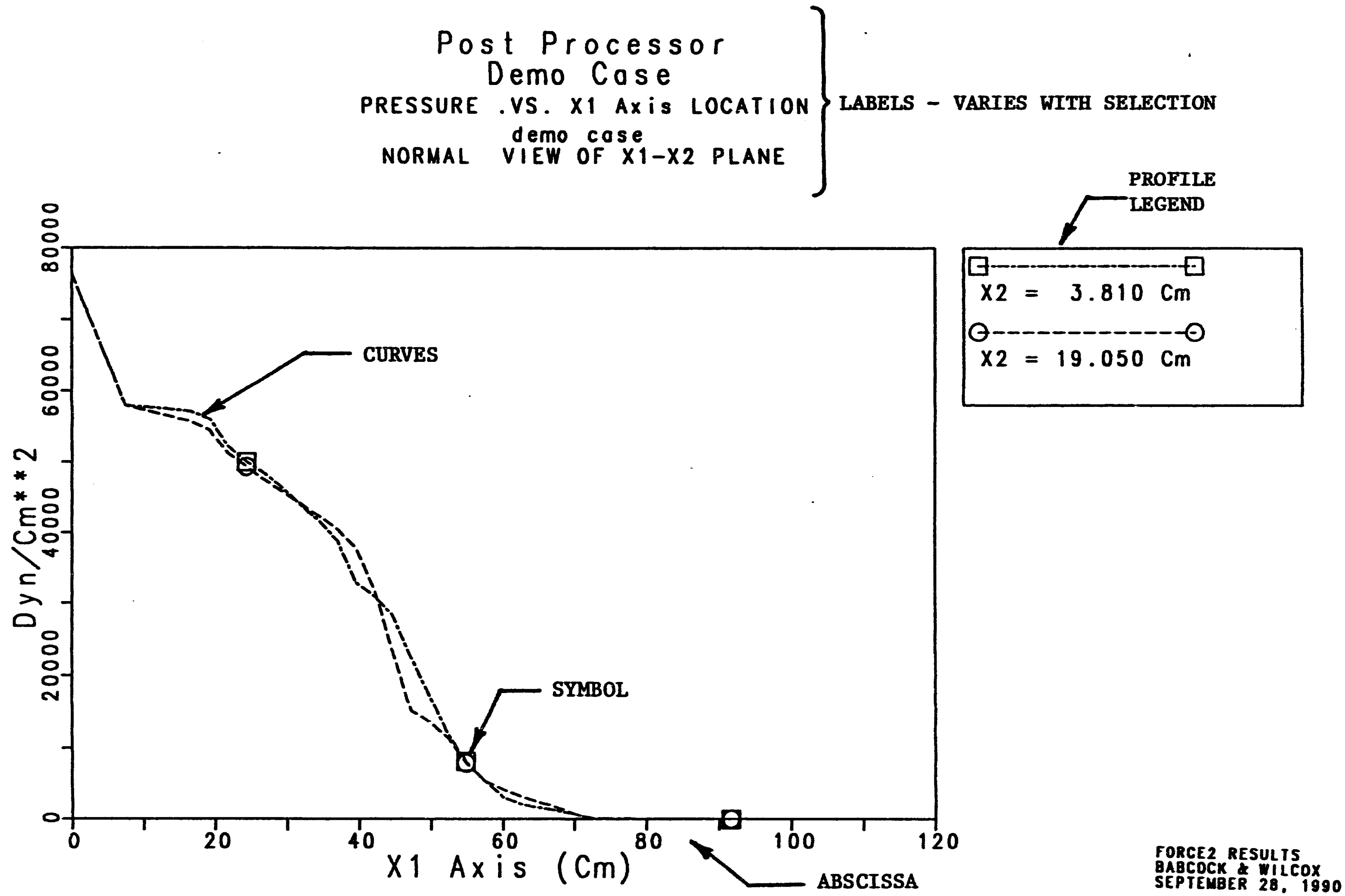


Figure 7-7. Profile plot with component parts labeled.

```
Post Processor
Demo Example
FORMAT = (I5, I5, 1X, A16, 3E11.3, 1X,A24 ,A16, E11.3)
 PLOT CURV  RESTART               X3         X2         X1      VARIABLE; NAME.          UNIT                  DATA
    1    1 demo case          1.524E+01  3.810E+00  0.000E+00 PRESSURE                        Dyn/Cm**2   7.620E+04
    1    1 demo case          1.524E+01  3.810E+00  7.500E+00 PRESSURE                        Dyn/Cm**2   5.797E+04
    1    1 demo case          1.524E+01  3.810E+00  1.650E+01 PRESSURE                        Dyn/Cm**2   5.718E+04
    1    1 demo case          1.524E+01  3.810E+00  1.927E+01 PRESSURE                        Dyn/Cm**2   5.603E+04
    1    1 demo case          1.524E+01  3.810E+00  2.181E+01 PRESSURE                        Dyn/Cm**2   5.224E+04
                                  •          •          •                                                   •
                                  •          •          •                                                   •
                                  •          •          •                                                   •
    1    2 demo case          1.524E+01  1.905E+01  0.000E+00 PRESSURE                        Dyn/Cm**2   7.620E+04
    1    2 demo case          1.524E+01  1.905E+01  7.500E+00 PRESSURE                        Dyn/Cm**2   5.796E+04
    1    2 demo case          1.524E+01  1.905E+01  1.650E+01 PRESSURE                        Dyn/Cm**2   5.572E+04
    1    2 demo case          1.524E+01  1.905E+01  1.927E+01 PRESSURE                        Dyn/Cm**2   5.445E+04
    1    2 demo case          1.524E+01  1.905E+01  2.181E+01 PRESSURE                        Dyn/Cm**2   5.119E+04
```

Figure 7-8. Profile output data file.

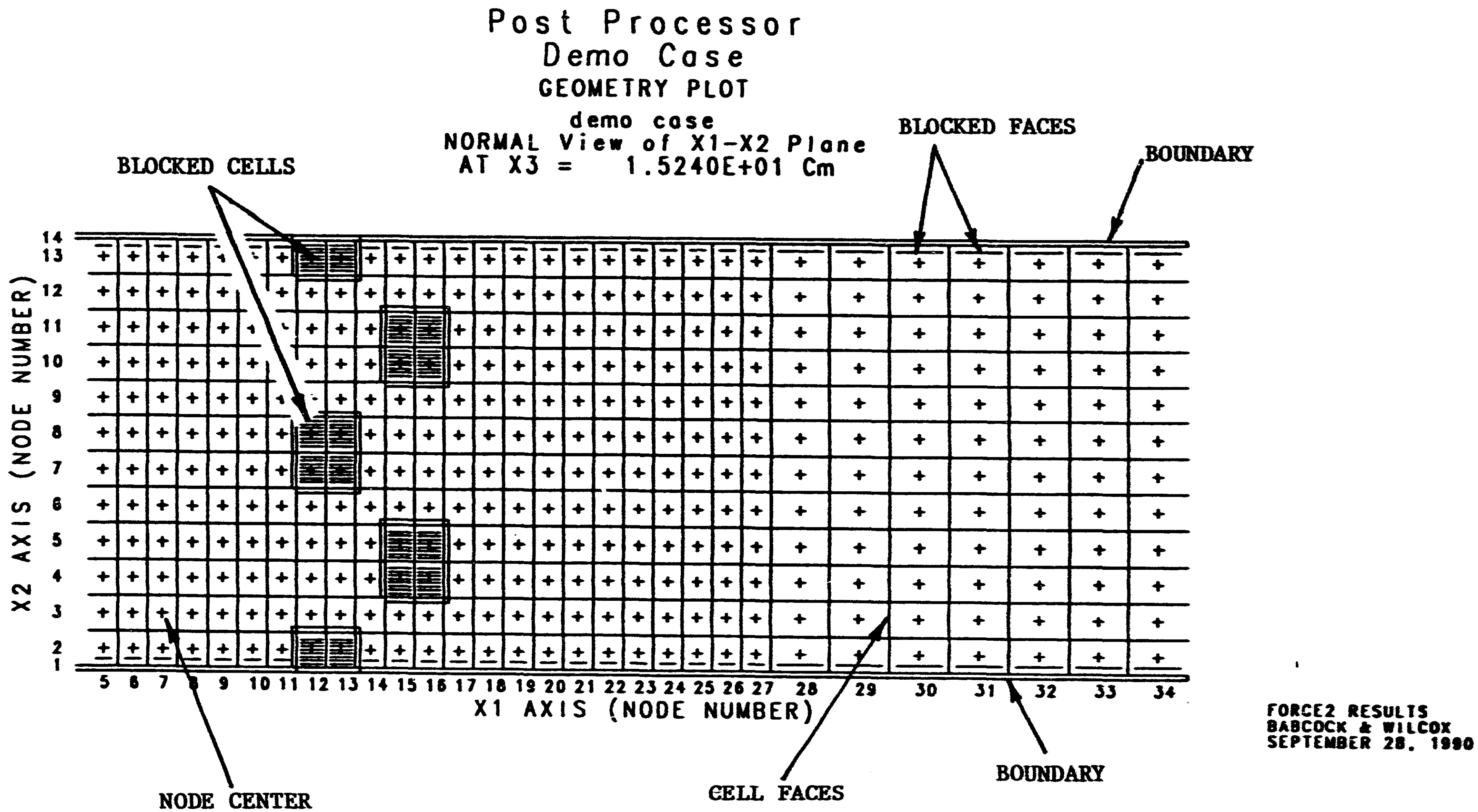


Figure 7-9. Geometry plot with component parts labeled.

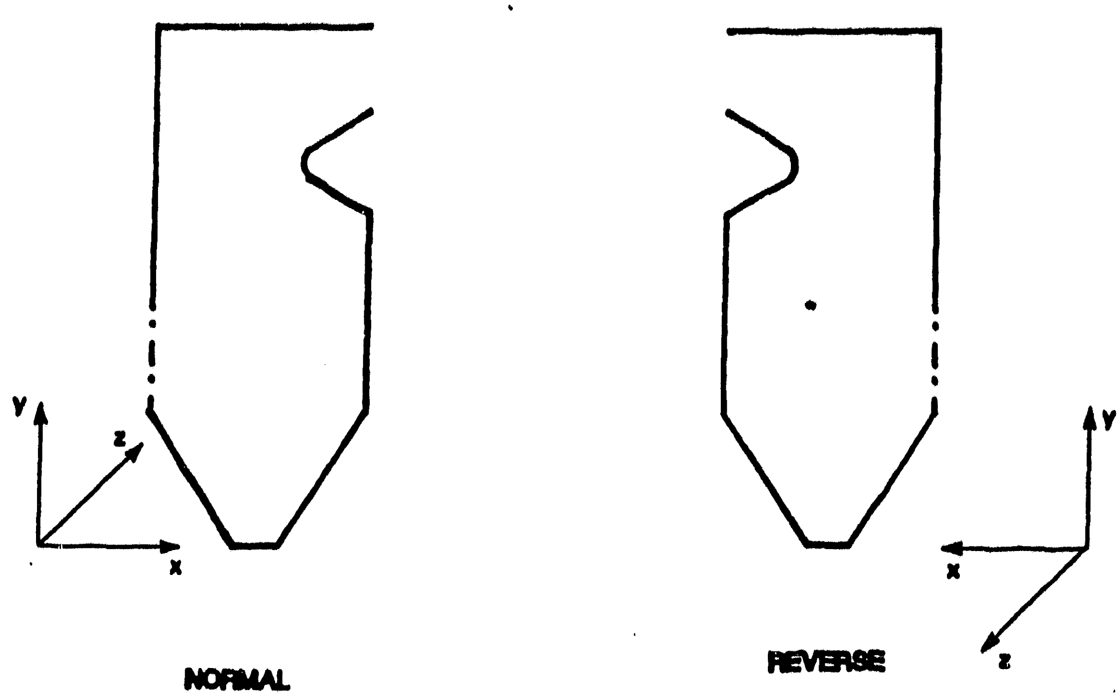


Figure 7-10. Invert X-axis option, normal and reverse views.

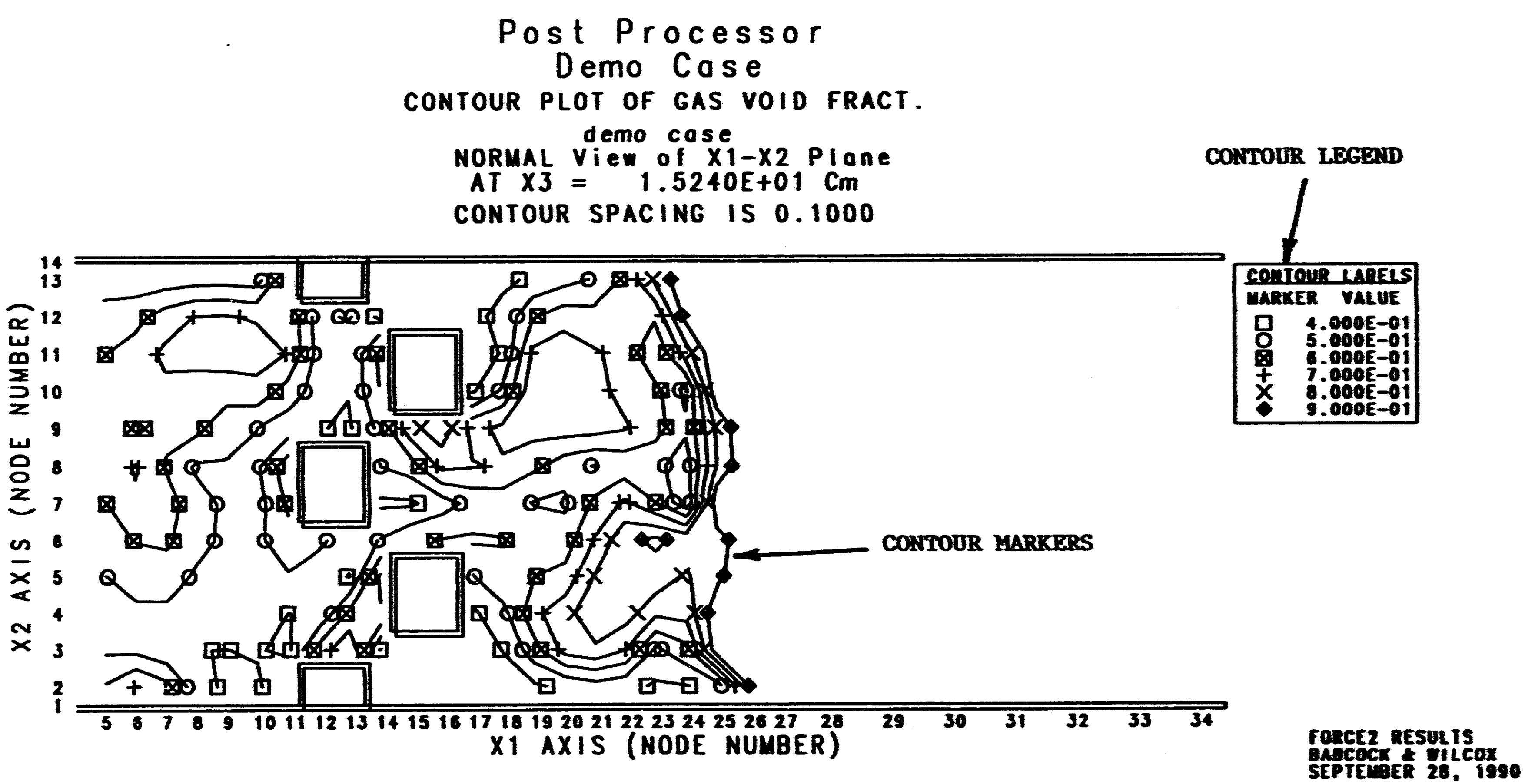


Figure 7-11. Contour plot with component parts labeled.

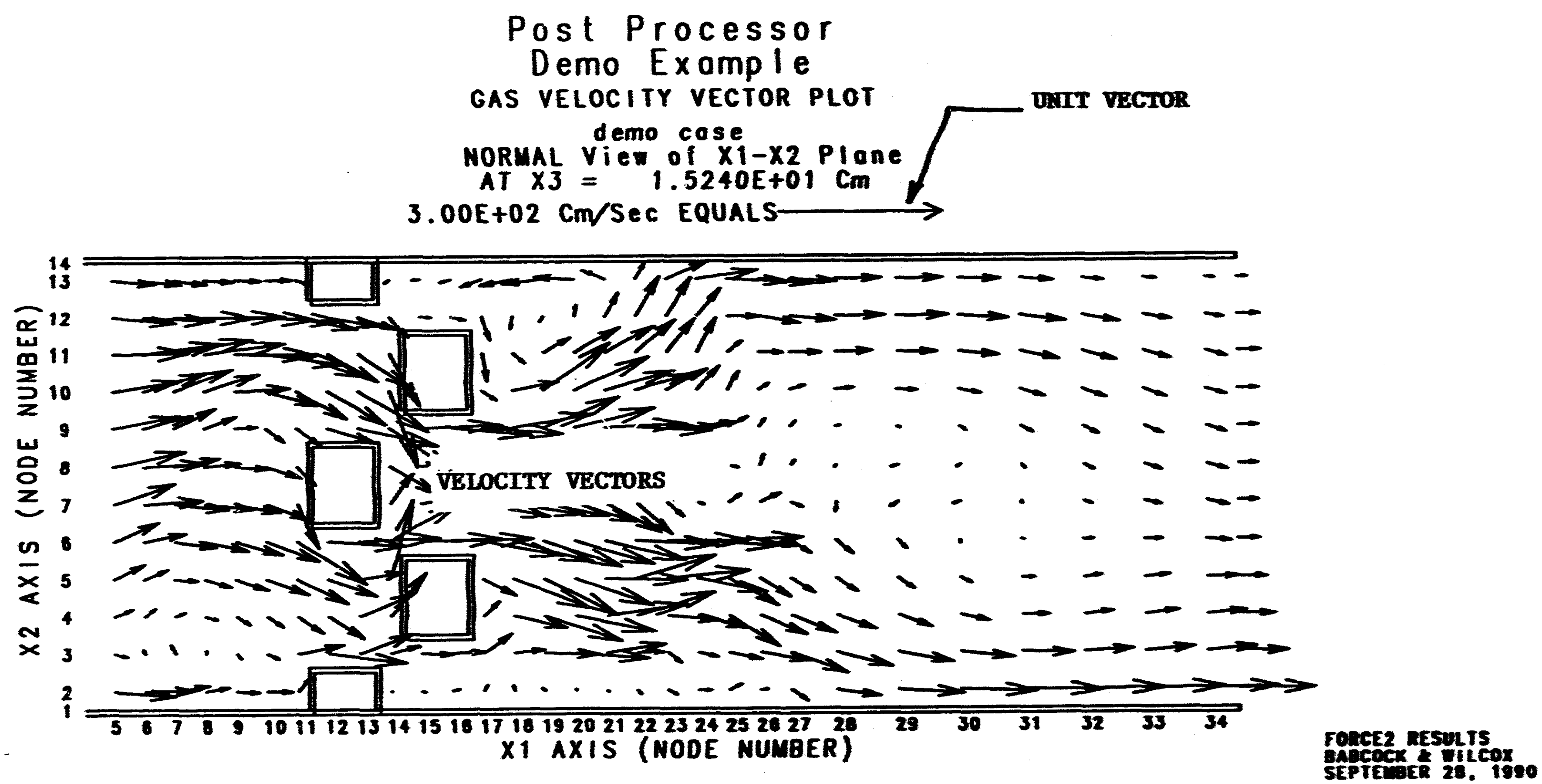


Figure 7-12. Velocity vector plot with component parts labeled.

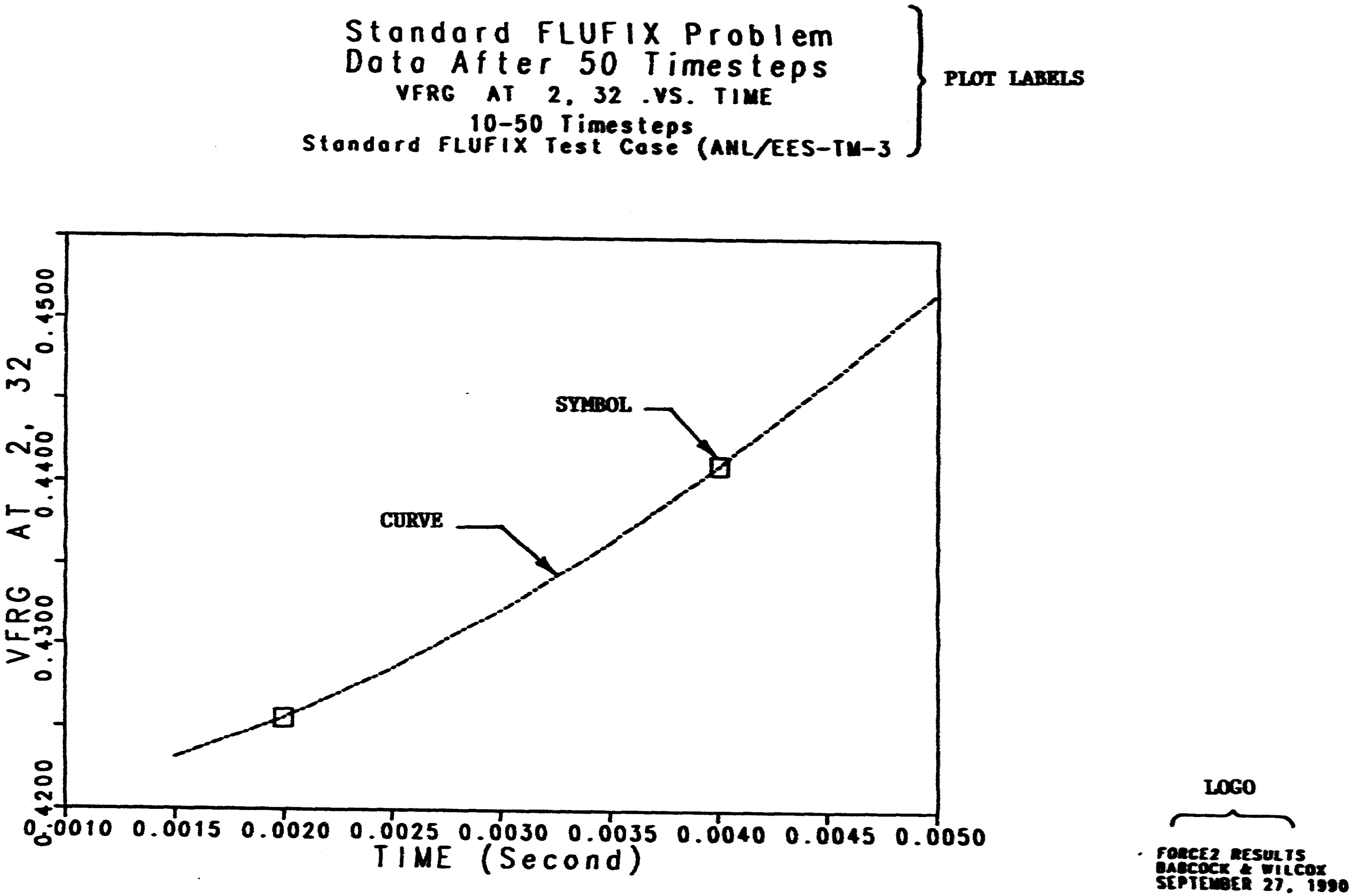


Figure 7-13. Variable vs time plot with the component parts labeled.

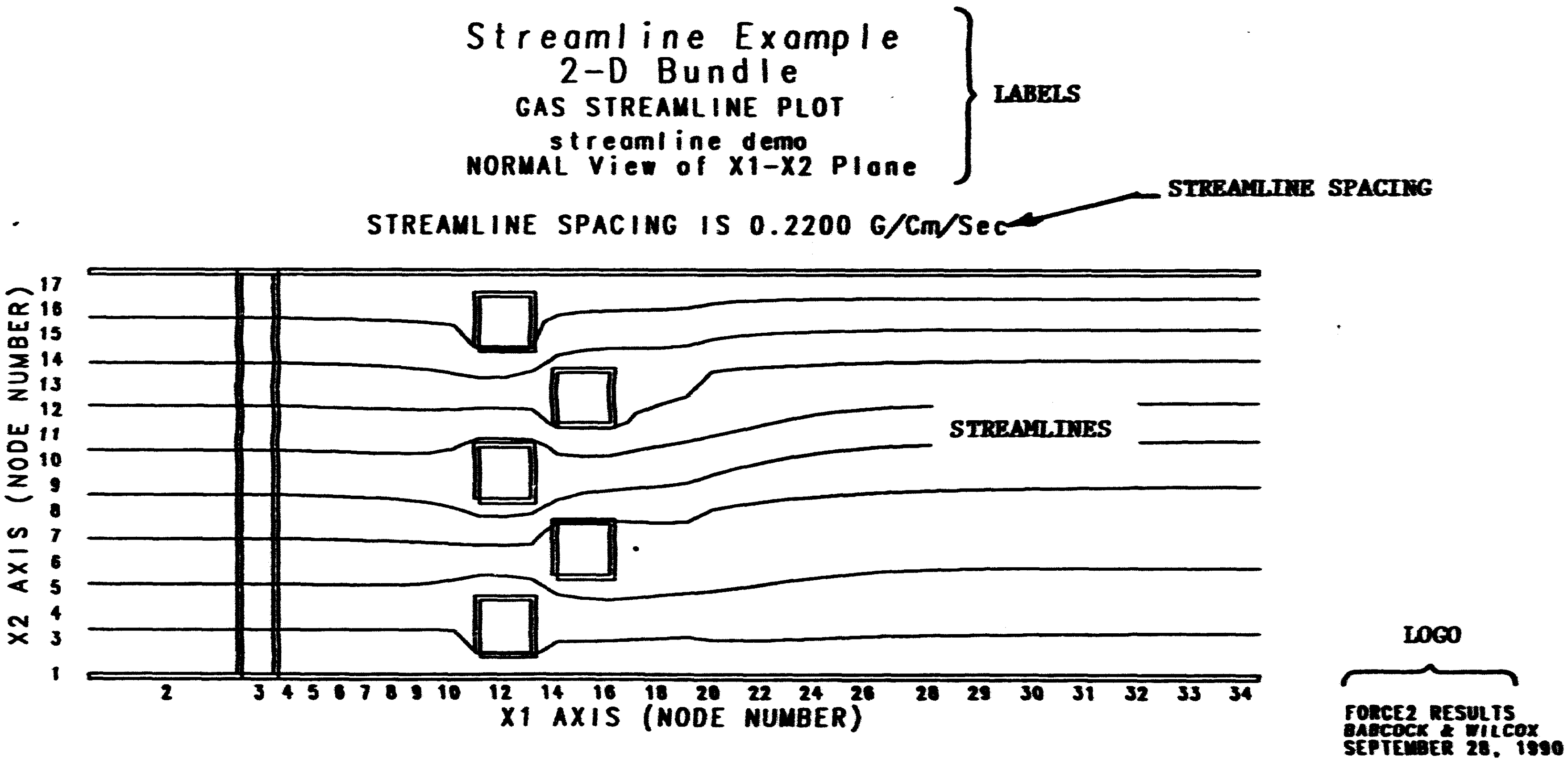


Figure 7-14. Streamline plot with component parts labeled.

8.0 REFERENCES

Lyczkowski, R.W., "Interim User's Manual for FLUFIX/MOD1: A Computer Program for Fluid-Solids Hydrodynamics," ANL/EES-TM-361, Argonne National Laboratory, October 1989.

Private communications with the Metal Wastage in Fluidized-Bed Combustors Steering Committee, June - September 1990.

APPENDIX A

TYPICAL COMMAND PROCEDURES TO EXECUTE FORCE2

Appendix A

Typical Command Procedures to Execute FORCE2

Command procedures for executing FORCE2 on a Sun 4/260 and VAX machines are listed in Tables A-1 and A-2. The user file names and corresponding FORCE2 files are listed in Table A-3.

A command procedure for executing the FORCE2 post-processor is listed in Table A-4.

Table A-1

A UNIX SCRIPT FOR EXECUTING FORCE2 ON A SUN 4/260 COMPUTER

```
# -------------------------------------------------------------
limit filesize 4000
#
# -------------------------------------------------------------
#  +++++++++  Assign FORCE2 Input Data File  +++++++
#
      ln -s   force2.in          in.dat
#
# -------------------------------------------------------------
#  +++++++++  Assign FORCE2  Input Restart File  +++
#
       ln -s    restart.in      fort.1
#
#  ++++++++++  Assign FORCE2  Output Restart File +++
#
       ln -s    restart.out    fort.2
#
# -------------------------------------------------------------
#  +++++++++++  Assign Runtime File  ++++++++++++++++
#
        ln -s     runtime       fort.11
#
# -------------------------------------------------------------
#  ++++++++    Assign  Log File  ++++++++++++++++++++
#
        ln -s      force2.log       fort.20
#
#  -------------------------------------------------------------
#  +++++++   Assign  Post Processor Files +++++++++++
#
#      Operating conditions throughout domain written to Unit #69
#
        ln -s         force2.ppf      fort.69
#
#      Selected operating conditions written to Unit #70
#
        ln -s         force2.ppt      fort.70
#
#  -------------------------------------------------------------
#  ++++   Assign  Hydrodynamic Data File for Erosion Model ++++
#
       ln -s          hydro.dat      fort.71
#
# -------------------------------------------------------------
#    Execute  FORCE2  , Name of  executable  assumed to be "FORCE2"
#
  (  FORCE2 < in.dat   > force2.out )>&! run.log
#
#    FORCE2 Output Listing  on  force2.out
#
  rm fort.*
  rm in.dat
exit
```

Table A-2

A COMMAND PROCEDURE FOR EXECUTING FORCE2 ON A VAX COMPUTER

```
$!
$! ------------------------------------------------
$!++++++++ Assign  FORCE2 Input Data File +++++++++
$!
$      ASSIGN  force2.in       FOR005
$! ------------------------------------------------
$! ++++++  Assign FORCE2  Output Listing  +++++++++
$!
$      ASSIGN  force2.out      FOR006
$! ------------------------------------------------
$! ++++++  Assign Input Restart File +++++++++++++
$!
$      ASSIGN  restart.in      FOR001
$!
$! ++++++  Assign Output Restart File +++++++++++++
$!
$      ASSIGN  restart.out     FOR002
$! -----------------------------------------------
$! ++++++  Assign  Runtime  File  +++++++++++++++++
$!
$      ASSIGN  runtime         FOR011
$! ------------------------------------------------
$! ++++++  Assign  Log  File  ++++++++++++++++++++
$!
$      ASSIGN  force2.log      FOR020
$!
$! ------------------------------------------------
$! ++++++  Assign  Post Processor Files +++++++++++
$!
$!    All Field Variables written to Unit #69
$!
$      ASSIGN  force2.ppf      FOR069
$!
$!    Time data  written to Unit #70
$!
$      ASSIGN  force2.ppt      FOR070
$!
! -----------------------------------------------
$! ++++++  Assign Erosion Model Data File ++++++++
$!
$      ASSIGN  hydro.dat      FOR071
$! ------------------------------------------------
$!
$! Execute FORCE2 , Name of executable assumed "FORCE2"
$!
$      RUN  FORCE2
$!
$ EXIT
```

Table A-3

USER FILE NAMES FOR THE FORCE2 INPUT AND OUTPUT FILES

FORCE2 FILE	USER FILE NAME
INPUT DATA	force2.in
OUTPUT LISTING	force2.out
RESTART INPUT	restart.in
RESTART OUTPUT	restart.out
RUNTIME	runtime
LOG	force2.log
POST-PROCESSOR	
Field Variables	force2.ppf
Time Data	force2.ppt
EROSION MODEL DATA	hydro.dat

Table A-4

A UNIX SCRIPT FOR EXECUTING THE POST-PROCESSOR ON A SUN 4/260 COMPUTER

```
#!  /bin/csh  -f
#       Setup files from FORCE2 for post processing with F2PLOT

onintr interupt

set f2dir = /mass/djb/force2
set ext = ppf
setenv gksdir "/usr/local/gks/grafpak"

echo " "
echo "FORCE2 Post Processing"

# field variable data file
get_restart:
    if (-e PPF)  rm -f PPF
    set f = $cwd:t
    echo -n "Field data file for plotting   [${f}.ppf] : "
    set ans = $<
    if ($ans == '')  then
        set ans = $f
    else
        set f = $ans:r
    endif
    set fr = ${f}.ppf
    if (! -e $fr)  then
        echo "Field data file $fr does not exist."
        goto get_restart
    endif
    ln -s $fr PPF

# Variable .vs. time data file
    if (-e PPT)  rm -f PPT
    if(-e $f.ppt) ln -s $f.ppt PPT

# Geometry data file
    if (-e PP_BOUND1)  rm -f PP_BOUND1
    if(-e $f.ppg) ln -s $f.ppg PP_BOUND1

# Units conversion file
if (-e CONVUNIT)  rm -f CONVUNIT
    ln convunit.dat CONVUNIT
```

Table A-4 (Cont'd)

A UNIX SCRIPT FOR EXECUTING THE POST-PROCESSOR ON A SUN 4/260 COMPUTER

```
echo -n "Use a Session file [n]? "
set ans = $<

if ($ans == 'y' | $ans == 'Y')  then
    get_session:
        if (-e PP_SESSION)  rm -f PP_SESSION
        ls -l *.ses
        echo
        echo -n "Session file name   [$f.ses]: "
        set ans = $<
        if ($ans == '')  then
                set fs = $f.ses
        else
                set fs = $ans:r.ses
        endif
        if (! -e $fs)  then
                echo "Session file does not exist"
            goto get_session
        endif
    ln $fs PP_SESSION
else
        if (-e PP_SESSION)  rm -f PP_SESSION
endif

if ($1 == 'debug')  then
        if (-e PP_SESSION)  rm -f PP_SESSION
        exit
endif

$f2dir/f2plot

if (-e PP_SESSION)  then
    ln PP_SESSION $f.ses.0
        @ ver = 0
        set nonomatch
        foreach s ($f.ses.[0-9]*)
                if ($s:e >= $ver)  @ ver = $s:e + 1
        end
        unset nonomatch
        mv -f $f.ses.0   $f.ses.$ver
        mv -f PP_SESSION $f.ses
endif

if (-e CONVUNIT)      rm -f CONVUNIT
if (-e PP_BOUND1)           rm -f PP_BOUND1
if (-e PP_SESSION)    rm -f PP_SESSION
if (-e PPF)           rm -f PPF
if (-e PPT)           rm -f PPT

exit
interupt:
        echo "Process interupted by Cntrl-C"
quit:
        exit
```

APPENDIX B

FORCE2 AND POST-PROCESSOR FILES FOR THE FLUFIX STANDARD PROBLEM

Appendix B

FORCE2 and Post-Processor Files for the FLUFIX Standard Problem

The FORCE2 input data for the initial and restart runs, the FORCE2 output listing and log file for the initial run, and the post-processor boundary data file are given in this appendix. These files were developed during runs of the FLUFIX standard problem.

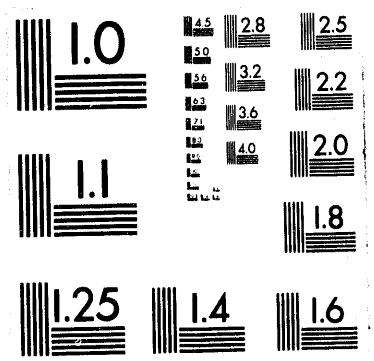
+++++++++++++++++ FORCE2 Input , Initial Run ++++++++++++++++

```
$       Bed  with Glass Beads
$       This Problem is the Standard FLUFIX Test Case
$
$       Note:  No  Gas or Solids Viscous Stress, MUG = MUS = 0.0
$              Gas Viscosity of .000182 Used to Evaluate Drag Coefficient
$ ---------------------------------------------------------------------------
SAVE
TITLE Standard FLUFIX Test Case (ANL/EES-TM-361)
GEOMETRY
$
$ -----   G e o m e t r y   P a r a g r a p h   -------
$
CARTESIAN  14   33
DX1    4.87
DX2     .635
$
$
$
 BOUNDARY  X2  LOWER  DIRICHLET
 BOUNDARY  X2  UPPER  DIRICHLET
 BLOCK   X2   0.0      0.0    58.44
 BLOCK   X2  19.685    0.0    58.44
BLOCK CELLS  9.74     19.48  18.4150  19.685

PROPERTIES
$
$ -----   P r o p e r t i e s   P a r a g r a p h   -------
$
$     Gas   Density   &    Viscosity
$
DENSITY    GAS    IDEAL
MOLECULAR  WEIGHT        1.0
GAS        CONSTANT     2.87E06
VISCOSITY  GAS          0.0
$
$  Solids   Density   &  Viscosity ( Specify Gas Viscosity )
$
DENSITY    SOLIDS    2.440 ,  0.0 ,  0.0 ,  0.0
VISCOSITY  SOLIDS       0.0
$
$     Gas  &  Solids  Reference  Densities  ( Used only for steady soln)
$
GAS     REFERENCE  DENSITY    .00117
SOLIDS  REFERENCE  DENSITY   2.440
$
$     Reference  Pressure (Atmospheric)
$
PRESSURE  1.013E06,  13, 2
$
$      Reference and  Absolute Temperatures
$
REFERENCE  TEMPERATURE    0.0
TEMPERATURE              298.0
```

```
FLOW
$
$ -----   F l o w   P a r a g r a p h   -------
$
TRANSIENT
GRAVITY   980.621 ,  0.0  , 0.0
$
$
DIAMETER   5.03E-02
SPHERISITY    1.0
VECTOR  DRAG
MODEL  B
$
$     Solid  Stress  Parameters
$
SOLIDS  STRESS  PARAMETER  GNORM    1.
SOLIDS  STRESS  PARAMETER  SLOPE   600.
SOLIDS  STRESS  PARAMETER  EPSTAR    .384

CONTROL
$
$ -----   C o n t r o l   P a r a g r a p h   -------
$
$       Print  & Data Output Control
$
PRINT  OFF
PRINT   P     INITIAL
PRINT   P     FINAL
PRINT   VFRG  FINAL
PRINT   U1G   FINAL
PRINT   U1S   FINAL
PRINT   U2G   FINAL
PRINT   U2S   FINAL
$
EROSION  MODEL  DATA    10000
$
$  Program  Execution  &  Timestep
$
TIME STEP   .0001
MAXIMUM  TIME STEPS    10
$
$   Iteration  &  Relaxation  Parameters
$
MAXIMUM   MICRO  ITERATIONS    5
MAXIMUM   MACRO  ITERATIONS    100
$
$     Convergence & Solution  Parameters
$
MASS  CONVERGENCE  CRITERIA    .0001
MAXIMUM   VOID                 .999
POST PROCESSOR DATA
     SAVE  ALL                  5

PERMEABILITIES
$
$ -----   P o r o s i t y / P e r m e a b i l i t y   -----------
```

```
$     Set   to   1.0
$
SET1
    SET2
       POROSITY          1.0
SET1
    SET2
       X1-PERMEABILITY   1.0

INITIAL
$
$  -----   I n i t i a l i z a t i o n    P a r a g r a p h   -------
$
$
$      Initialize Pressure Field Based on Min Fluidization Condition
$
MINIMUM  FLUIDIZATION   1
$
$     Set  velocities  & voids throughout  field
$
SET1
  SET2
         U1G     0.0
         U1S     0.0
         U2G     0.0
         U2S     0.0
         VFRG    1.0
         VFRS    0.0
          P      0.0
$
$     Set   Inlet   Pressure
$
SET1    1,  1
     SET2   1,  33
         P         41900.0
$
$     Set  Inflow  Gas  Velocities
$
$  ---  1.)  Secondary  Stream
$
SET1   1, 2
  SET2     2,  31
         U1G     26.0
$
$ ---   2.)  J e t
$
SET1   1, 2
  SET2     32,  32
         U1G    578.
$   ++++++++++++++++++++++++++++++++++++++
$
$     Set   Gas  Velocities  in Bed
$
SET1  3,  8
```

```
          U1G   61.9
$
$     Set   Gas  Velocities  in  Above Bed
$
SET1  9,  14
  SET2  1, 33
          U1G    26.
$
$      Set  Velocities  at Blocked Cells ( Just to be Safe)
$
SET1  4,  6
  SET2  31,  32
          U1G   0.0
$
$      Initialize  Void  Fractions in Bed
$
SET1  2,  7
  SET2  1,   33
          VFRG  .42
          VFRS  .58

FLOFLAGS
$
$  -----   C e l l   F l a g   P a r a g r a p h   -------
$
$    Set   all  to  free flow  cells  IFLO = 1
SET1
  SET2
           GAS     1
           SOLIDS  1
$
$  Inflow  Cells For Gas Phase   IFLO = 2
$
SET1  1, 1
  SET2  2, 32
           GAS    2
$
$   Constant Pressure Outflow Cells For Gas Phase IFLO = 4
$
SET1  14, 14
  SET2  2, 32
           GAS   4
$
$  No Flow Cells For Solids Phase at Non-Blocked Surfaces, IFLO = 0
$
SET1  1, 1
  SET2 2, 32
          SOLIDS   0
SET1  14, 14
  SET2 2, 32
          SOLIDS   0
$ +++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++
$  The Slip/No-Slip Conditions are Meaningless for This Run
$  Because Gas & Solids Viscosities are 0.0
$
```


1.0
1.1
1.25
1.4
1.6
1.8
2.0
2.2
2.5
2.8
3.2
3.6
4.0
4.5
5.0
5.6
6.3

3 of 3

```
$
$     Lower  X2 - Boundary
$            Gas    :  No-Slip  ,      IFLO = -3
$            Solids :  No-Slip  ,      IFLO = -3
$
SET1 1, 14
  SET2 1, 1
          GAS       -3
          SOLIDS    -3
$
$     Upper  X2 - Boundary
$            Gas    :  Free Slip ,     IFLO = -1
$            Solids :  Free Slip ,     IFLO = -1
$
SET1 1, 14
  SET2 33,33
          GAS       -1
          SOLIDS    -1
$
$     Upper  X2 - Boundary , At the Obstacle
$            Gas    :  No Slip ,       IFLO = -3
$            Solids :  No Slip ,       IFLO = -3
$
SET1  4, 5
  SET2  31, 32
          GAS       -3
          SOLIDS    -3
$
$    End of Data
$
```

++++++++++++++++++ FORCE2 Input , Restart Run ++++++++++++++++

```
$    2-D  SAMPLE PROBLEM , Units are in cm
$        Bed  with Glass Beads
$        This Problem is the Standard FLUFIX Test Case
$
$        Note:  No  Gas or Solids Viscous Stress, MUG = MUS = 0.0
$               Gas Viscosity of .000182 Used to Evaluate Drag Coefficient
$
$    +++++++++++    Restart  Run  ++++++++++++++++++++++++++++++
$ -------------------------------------------------------------------------------
RESTART SAVE
TITLE Restart  FLUFIX Test Case (ANL/EES-TM-361)

FLOW
$
$  -----    F l o w    P a r a g r a p h    -------
$
TRANSIENT

CONTROL
$
$  -----    C o n t r o l    P a r a g r a p h    -------
$
$       Print  & Data Output Control
$
PRINT  OFF
PRINT   P      FINAL
PRINT   VFRG   FINAL
PRINT   U1G    FINAL
PRINT   U1S    FINAL
PRINT   U2G    FINAL
PRINT   U2S    FINAL
$
$
$  Program  Execution  &  Timestep
$
TIME STEP    .0001
MAXIMUM  TIME STEPS     50
$
$
MASS  CONVERGENCE  CRITERIA     .0001
MAXIMUM   VOID                  .999
POST PROCESSOR DATA
     SAVE  ALL                   10
     SAVE  SELECTED VARAIABLES   5
STORE  VFRG  AT   2,  32
STORE  VFRG  AT   2,  31
STORE  P     AT   2,  32
STORE  P     AT   6,  30
STORE  U1G   AT   3,  32
STORE  U1S   AT   3,  32
STORE  U1G   AT   5,  30
STORE  U1S   AT   5,  30
$
$   End of Data
```

+++++++++++ FORCE2 Output Listing , Initial Run +++++++++

```
1

        * * * *   F  O  R  C  E  2 * * * *

        V E R S I O N    1.0    15-Feb-1990

        F L O W   I N   O R T H O G O N A L
      C O O R D I N A T E S   E U L E R I A N

        B A B C O C K   A N D   W I L C O X
   A L L I A N C E   R E S E A R C H   C E N T E R

1 LINE          INPUT DATA

      $
      $   2-D  SAMPLE PROBLEM , Units are in cm
      $       Bed  with Glass Beads
      $       This Problem is the Standard FLUFIX Test Case
      $
      $       Note:  No  Gas or Solids Viscous Stress, MUG = MUS = 0.0
      $              Gas Viscosity of .000182 Used to Evaluate Drag Coefficient
      $ ----------------------------------------------------------------------
      SAVE
      TITLE Standard FLUFIX Test Case (ANL/EES-TM-361)
      GEOMETRY
      $
      $ -----   G e o m e t r y   P a r a g r a p h  -------
      $
      CARTESIAN  14   33
      DX1     4.87
      DX2      .635
```

```
$
$
$
 BOUNDARY  X2  LOWER  DIRICHLET
 BOUNDARY  X2  UPPER  DIRICHLET
 BLOCK  X2   0.0        0.0   58.44
 BLOCK  X2  19.685      0.0   58.44
BLOCK CELLS  9.74     19.48  18.4150  19.685

PROPERTIES
$
$  -----   P r o p e r t i e s   P a r a g r a p h   -------
$
$    Gas   Density   &    Viscosity
$
DENSITY    GAS    IDEAL
MOLECULAR  WEIGHT          1.0
GAS        CONSTANT      2.87E06
VISCOSITY  GAS           0.0
$
$  Solids   Density   &  Viscosity ( Specify Gas Viscosity )
$
DENSITY    SOLIDS    2.440 ,  0.0 ,  0.0 ,  0.0
VISCOSITY  SOLIDS      0.0
$
$    Gas  &  Solids  Reference  Densities  ( Used only for steady soln)
$
GAS     REFERENCE  DENSITY    .00117
SOLIDS  REFERENCE  DENSITY   2.440
$
$    Reference  Pressure (Atmospheric)
$
PRESSURE  1.013E06,  13, 2
$
$     Reference and  Absolute Temperatures
$
REFERENCE  TEMPERATURE    0.0
TEMPERATURE              298.0

FLOW
$
$  -----   F l o w   P a r a g r a p h   -------
$
TRANSIENT
GRAVITY   980.621 ,  0.0  , 0.0
$
$
DIAMETER   5.03E-02
SPHERISITY    1.0
VECTOR  DRAG
MODEL  B
$
$    Solid  Stress  Parameters
$
SOLIDS  STRESS  PARAMETER  GNORM    1.
SOLIDS  STRESS  PARAMETER  SLOPE   600.
SOLIDS  STRESS  PARAMETER  EPSTAR    .384

```

```
CONTROL
$
$ -----   C o n t r o l   P a r a g r a p h   -------
$
$        Print  & Data Output Control
$
PRINT  OFF
PRINT   P      INITIAL
PRINT   P      FINAL
PRINT   VFRG   FINAL
PRINT   U1G    FINAL
PRINT   U1S    FINAL
PRINT   U2G    FINAL
PRINT   U2S    FINAL
$
EROSION  MODEL  DATA     10000
$
$  Program  Execution  &  Timestep
$
TIME STEP    .0001
MAXIMUM  TIME STEPS     10
$
$   Iteration  &  Relaxation  Parameters
$
MAXIMUM   MICRO  ITERATIONS     5
MAXIMUM   MACRO  ITERATIONS     100
$
$      Convergence & Solution  Parameters
$
MASS  CONVERGENCE  CRITERIA     .0001
MAXIMUM   VOID                  .999
POST PROCESSOR DATA
      SAVE  ALL                  5

PERMEABILITIES
$
$ -----   P o r o s i t y / P e r m e a b i l i t y   -----------
$                   P a r a g r a p h
$
$     Set  to  1.0
$
SET1
     SET2
        POROSITY          1.0
SET1
     SET2
        X1-PERMEABILITY  1.0

INITIAL
$
$ -----   I n i t i a l i z a t i o n   P a r a g r a p h   -------
$
$
$       Initialize Pressure Field Based on Min Fluidization Condition
$
MINIMUM  FLUIDIZATION   1
$
$      Set  velocities  & voids throughout  field
```

```
$
SET1
  SET2
         U1G     0.0
         U1S     0.0
         U2G     0.0
         U2S     0.0
         VFRG    1.0
         VFRS    0.0
          P      0.0
$
$     Set   Inlet   Pressure
$
SET1   1,  1
     SET2   1,  33
         P         41900.0
$
$     Set  Inflow  Gas  Velocities
$
$  ---  1.)  Secondary  Stream
$
SET1   1, 2
  SET2      2,  31
         U1G     26.0
$
$ ---   2.)  J e t
$
SET1   1, 2
  SET2     32,  32
         U1G    578.
$   ++++++++++++++++++++++++++++++++++++++
$
$     Set   Gas  Velocities  in Bed
$
SET1  3,  8
  SET2  1, 33
          U1G   61.9
$
$     Set   Gas  Velocities  in  Above Bed
$
SET1  9,  14
  SET2  1, 33
          U1G    26.
$
$      Set  Velocities  at Blocked Cells ( Just to be Safe)
$
SET1  4,  6
  SET2  31,  32
          U1G   0.0
$
$      Initialize  Void  Fractions in Bed
$
SET1  2,  7
  SET2  1,    33
          VFRG  .42
          VFRS  .58

FLOFLAGS
```

```
 Minimum  Fluidization Velocity :   5.832713E+01
  Gas  Void   :     0.4200

$
$  -----   C e l l   F l a g   P a r a g r a p h   -------
$
$    Set   all  to  free flow  cells  IFLO = 1
SET1
  SET2
          GAS     1
          SOLIDS  1
$
$  Inflow  Cells For Gas Phase  IFLO = 2
$
SET1  1, 1
  SET2  2, 32
          GAS    2
$
$   Constant Pressure Outflow Cells For Gas Phase IFLO = 4
$
SET1  14, 14
  SET2  2, 32
          GAS   4
$
$  No Flow Cells For Solids Phase at Non-Blocked Surfaces, IFLO = 0
$
SET1  1, 1
  SET2 2, 32
          SOLIDS   0
SET1  14, 14
  SET2 2, 32
          SOLIDS   0
$ ++++++++++++++++++++++++++++++++++++++++++++++++++++++++++++
$  The Slip/No-Slip Conditions are Meaningless for This Run
$  Because Gas & Solids Viscosities are 0.0
$
$     Gas & Solids Slip/ No-Slip Conditons Along Boundaries
$
$     Lower  X2 - Boundary
$           Gas    :  No-Slip  ,     IFLO = -3
$           Solids :  No-Slip  ,     IFLO = -3
$
SET1 1, 14
  SET2 1, 1
          GAS      -3
          SOLIDS   -3
$
$     Upper  X2 - Boundary
$           Gas    :  Free Slip ,     IFLO = -1
$           Solids :  Free Slip ,     IFLO = -1
$
SET1 1, 14
  SET2 33,33
          GAS      -1
          SOLIDS   -1
$
```

```
$      Upper  X2 - Boundary , At the Obstacle
$             Gas    :  No Slip ,        IFLO = -3
$             Solids :  No Slip ,        IFLO = -3
$
SET1  4, 5
  SET2  31, 32
           GAS       -3
           SOLIDS    -3
$
$     End of Data
$


                    *****  F O R C E 2 - Rev 1.0 -  15 Feb, 1990   *****

      TIME 16:49:25          DATE : 9/26/90

1ITERATION    0                        Standard FLUFIX Test Case (ANL/EES-TM-3

0
0GEOMETRY          2-D CARTESIAN
0   AXIS    LENGTH NODES   COORDINATES

    X1     58.440   14     0.      4.87     9.74     14.6     19.5     24.3     29.2     34.1     39.0
                          43.8     48.7     53.6     58.4
    X2     19.685   33     0.      0.635    1.27     1.90     2.54     3.17     3.81     4.44     5.08
                          5.72     6.35     6.99     7.62     8.26     8.89     9.53     10.2     10.8
                          11.4     12.1     12.7     13.3     14.0     14.6     15.2     15.9     16.5
                          17.1     17.8     18.4     19.1     19.7

0   X2 Lower Boundary Condition - DIRICHLET
    X2 Upper Boundary Condition - DIRICHLET

0   GEOMETRIC BLOCKAGES
0              X1 Limits              X2 Limits
    Type     Low         High       Low        High

     X2    0.0000E+00 5.8440E+01     0.0000E+00
     X2    0.0000E+00 5.8440E+01     1.9685E+01
    CELL   9.7400E+00 1.9480E+01 1.8415E+01 1.9685E+01
0
0FLUID PROPERTIES

                               GAS          SOLIDS
     NOMINAL DENSITY        1.184E-03   2.44
     NOMINAL VISCOSITY          0.          0.
     GRAVITATIONAL ACCELERATIONS    -981.        0.        0.
     SURFACE TENSION COEFFICIENT    7.000E-02
     REFERENCE PRESSURE     1.013E+06  ( 13, 2)
0     GRAVITY AND ANGLES     981.        0.        0.


                    *****  F O R C E 2 - Rev 1.0 -  15 Feb, 1990   *****

      TIME 16:49:25          DATE : 9/26/90
```

1ITERATION 0 Standard FLUFIX Test Case (ANL/EES-TM-3

0 PRESSURE

0 X1 (1) X2 (1)		1 0.00	2 0.24	3 0.73	4 1.22	5 1.70	6 2.19	7 2.68	8 3.17	9 3.65	10 4.14
33	1.97	4.190E+04	4.060E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
32	1.94	4.190E+04	3.722E+04	3.046E+04	0.000E+00	0.000E+00	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
31	1.87	4.190E+04	3.722E+04	3.046E+04	0.000E+00	0.000E+00	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
30	1.81	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
29	1.75	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
28	1.68	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
27	1.62	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
26	1.56	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
25	1.49	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
24	1.43	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
23	1.37	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
22	1.30	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
21	1.24	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
20	1.17	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
19	1.11	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
18	1.05	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
17	0.98	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
16	0.92	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
15	0.86	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
14	0.79	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
13	0.73	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
12	0.67	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
11	0.60	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
10	0.54	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
9	0.48	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
8	0.41	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
7	0.35	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
6	0.29	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
5	0.22	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
4	0.16	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
3	0.10	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
2	0.03	4.190E+04	3.722E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
1	0.00	4.190E+04	4.060E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01

0 X1 (1) X2 (1)		11 4.63	12 5.11	13 5.60	14 5.84
33	1.97	1.697E+01	1.131E+01	5.656E+00	0.000E+00
32	1.94	1.697E+01	1.131E+01	5.656E+00	0.000E+00
31	1.87	1.697E+01	1.131E+01	5.656E+00	0.000E+00
30	1.81	1.697E+01	1.131E+01	5.656E+00	0.000E+00
29	1.75	1.697E+01	1.131E+01	5.656E+00	0.000E+00
28	1.68	1.697E+01	1.131E+01	5.656E+00	0.000E+00
27	1.62	1.697E+01	1.131E+01	5.656E+00	0.000E+00
26	1.56	1.697E+01	1.131E+01	5.656E+00	0.000E+00
25	1.49	1.697E+01	1.131E+01	5.656E+00	0.000E+00
24	1.43	1.697E+01	1.131E+01	5.656E+00	0.000E+00
23	1.37	1.697E+01	1.131E+01	5.656E+00	0.000E+00
22	1.30	1.697E+01	1.131E+01	5.656E+00	0.000E+00
21	1.24	1.697E+01	1.131E+01	5.656E+00	0.000E+00
20	1.17	1.697E+01	1.131E+01	5.656E+00	0.000E+00

TIME 16:49:25 DATE : 9/26/90

1ITERATION 0 Standard FLUFIX Test Case (ANL/EES-TM-3

0 PRESSURE

0	X1 (1) / X2 (1)	11 4.63	12 5.11	13 5.60	14 5.84
19	1.11	1.697E+01	1.131E+01	5.656E+00	0.000E+00
18	1.05	1.697E+01	1.131E+01	5.656E+00	0.000E+00
17	0.98	1.697E+01	1.131E+01	5.656E+00	0.000E+00
16	0.92	1.697E+01	1.131E+01	5.656E+00	0.000E+00
15	0.86	1.697E+01	1.131E+01	5.656E+00	0.000E+00
14	0.79	1.697E+01	1.131E+01	5.656E+00	0.000E+00
13	0.73	1.697E+01	1.131E+01	5.656E+00	0.000E+00
12	0.67	1.697E+01	1.131E+01	5.656E+00	0.000E+00
11	0.60	1.697E+01	1.131E+01	5.656E+00	0.000E+00
10	0.54	1.697E+01	1.131E+01	5.656E+00	0.000E+00
9	0.48	1.697E+01	1.131E+01	5.656E+00	0.000E+00
8	0.41	1.697E+01	1.131E+01	5.656E+00	0.000E+00
7	0.35	1.697E+01	1.131E+01	5.656E+00	0.000E+00
6	0.29	1.697E+01	1.131E+01	5.656E+00	0.000E+00
5	0.22	1.697E+01	1.131E+01	5.656E+00	0.000E+00
4	0.16	1.697E+01	1.131E+01	5.656E+00	0.000E+00
3	0.10	1.697E+01	1.131E+01	5.656E+00	0.000E+00
2	0.03	1.697E+01	1.131E+01	5.656E+00	0.000E+00
1	0.00	1.697E+01	1.131E+01	5.656E+00	0.000E+00

***** F O R C E 2 - Rev 1.0 - 15 Feb, 1990 *****

TIME 16:52: 4 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0

0FLUID PROPERTIES

0 X1 GAS VEL.

0	X1 (1) / X2 (1)	2 0.00	3 0.49	4 0.97	5 1.46	6 1.95	7 2.43	8 2.92	9 3.41	10 3.90	11 4.38
33	1.97	0.000E+00	6.190E+01	6.190E+01	6.190E+01	6.190E+01	6.190E+01	6.190E+01	2.600E+01	2.600E+01	2.600E+01
32	1.94	5.780E+02	1.341E+02	0.000E+00	0.000E+00	0.000E+00	5.223E+01	5.620E+01	2.412E+01	2.481E+01	2.543E+01
31	1.87	2.600E+01	1.532E+02	0.000E+00	0.000E+00	0.000E+00	5.292E+01	5.622E+01	2.412E+01	2.481E+01	2.543E+01
30	1.81	2.600E+01	1.334E+02	9.407E+01	6.844E+01	6.606E+01	5.234E+01	5.636E+01	2.413E+01	2.481E+01	2.543E+01
29	1.75	2.600E+01	1.222E+02	9.147E+01	6.837E+01	6.488E+01	5.349E+01	5.647E+01	2.415E+01	2.481E+01	2.543E+01
28	1.68	2.600E+01	1.128E+02	8.879E+01	6.813E+01	6.390E+01	5.441E+01	5.662E+01	2.417E+01	2.482E+01	2.543E+01
27	1.62	2.600E+01	1.048E+02	8.612E+01	6.776E+01	6.310E+01	5.515E+01	5.680E+01	2.419E+01	2.483E+01	2.544E+01
26	1.56	2.600E+01	9.805E+01	8.353E+01	6.730E+01	6.242E+01	5.574E+01	5.702E+01	2.422E+01	2.483E+01	2.544E+01
25	1.49	2.600E+01	9.233E+01	8.106E+01	6.678E+01	6.184E+01	5.623E+01	5.724E+01	2.424E+01	2.484E+01	2.544E+01
24	1.43	2.600E+01	8.748E+01	7.873E+01	6.621E+01	6.136E+01	5.660E+01	5.745E+01	2.427E+01	2.485E+01	2.544E+01
23	1.37	2.600E+01	8.337E+01	7.657E+01	6.562E+01	6.095E+01	5.691E+01	5.765E+01	2.429E+01	2.485E+01	2.544E+01
22	1.30	2.600E+01	7.992E+01	7.454E+01	6.503E+01	6.059E+01	5.715E+01	5.783E+01	2.431E+01	2.486E+01	2.545E+01
21	1.24	2.600E+01	7.696E+01	7.272E+01	6.445E+01	6.027E+01	5.736E+01	5.799E+01	2.433E+01	2.487E+01	2.545E+01
20	1.17	2.600E+01	7.444E+01	7.107E+01	6.388E+01	5.998E+01	5.753E+01	5.813E+01	2.435E+01	2.487E+01	2.545E+01
19	1.11	2.600E+01	7.234E+01	6.956E+01	6.334E+01	5.973E+01	5.766E+01	5.824E+01	2.436E+01	2.488E+01	2.545E+01
18	1.05	2.600E+01	7.052E+01	6.823E+01	6.282E+01	5.952E+01	5.777E+01	5.833E+01	2.437E+01	2.488E+01	2.545E+01

17	0.98	2.600E+01	6.900E+01	6.702E+01	6.234E+01	5.933E+01	5.786E+01	5.840E+01	2.437E+01	2.488E+01	2.546E+01
16	0.92	2.600E+01	6.768E+01	6.598E+01	6.190E+01	5.917E+01	5.792E+01	5.844E+01	2.437E+01	2.489E+01	2.546E+01
15	0.86	2.600E+01	6.655E+01	6.506E+01	6.150E+01	5.903E+01	5.798E+01	5.845E+01	2.437E+01	2.489E+01	2.546E+01
14	0.79	2.600E+01	6.559E+01	6.426E+01	6.113E+01	5.892E+01	5.802E+01	5.845E+01	2.438E+01	2.489E+01	2.546E+01
13	0.73	2.600E+01	6.478E+01	6.357E+01	6.078E+01	5.883E+01	5.803E+01	5.845E+01	2.438E+01	2.490E+01	2.546E+01
12	0.67	2.600E+01	6.409E+01	6.296E+01	6.048E+01	5.875E+01	5.803E+01	5.844E+01	2.439E+01	2.490E+01	2.546E+01
11	0.60	2.600E+01	6.351E+01	6.243E+01	6.022E+01	5.868E+01	5.804E+01	5.844E+01	2.440E+01	2.490E+01	2.546E+01
10	0.54	2.600E+01	6.303E+01	6.198E+01	6.000E+01	5.860E+01	5.804E+01	5.844E+01	2.441E+01	2.491E+01	2.546E+01
9	0.48	2.600E+01	6.263E+01	6.160E+01	5.980E+01	5.853E+01	5.804E+01	5.844E+01	2.442E+01	2.491E+01	2.546E+01
8	0.41	2.600E+01	6.230E+01	6.129E+01	5.963E+01	5.847E+01	5.804E+01	5.844E+01	2.444E+01	2.491E+01	2.546E+01
7	0.35	2.600E+01	6.203E+01	6.103E+01	5.948E+01	5.841E+01	5.804E+01	5.844E+01	2.446E+01	2.492E+01	2.546E+01
6	0.29	2.600E+01	6.184E+01	6.082E+01	5.936E+01	5.835E+01	5.804E+01	5.844E+01	2.447E+01	2.492E+01	2.545E+01
5	0.22	2.600E+01	6.170E+01	6.065E+01	5.927E+01	5.831E+01	5.805E+01	5.844E+01	2.448E+01	2.493E+01	2.545E+01
4	0.16	2.600E+01	6.158E+01	6.052E+01	5.920E+01	5.828E+01	5.805E+01	5.843E+01	2.449E+01	2.494E+01	2.545E+01
3	0.10	2.600E+01	6.150E+01	6.044E+01	5.916E+01	5.825E+01	5.805E+01	5.843E+01	2.449E+01	2.494E+01	2.545E+01
2	0.03	2.600E+01	6.146E+01	6.041E+01	5.914E+01	5.824E+01	5.805E+01	5.843E+01	2.449E+01	2.494E+01	2.545E+01
1	0.00	0.000E+00	6.190E+01	6.190E+01	6.190E+01	6.190E+01	6.190E+01	6.190E+01	2.600E+01	2.600E+01	2.600E+01

0 X1 (1) 12 4.87 13 5.36 14 5.84

X2 (1)				
33	1.97	2.600E+01	2.600E+01	2.600E+01
32	1.94	2.600E+01	2.600E+01	2.600E+01
31	1.87	2.600E+01	2.600E+01	2.600E+01
30	1.81	2.600E+01	2.600E+01	2.600E+01
29	1.75	2.600E+01	2.600E+01	2.600E+01
28	1.68	2.600E+01	2.600E+01	2.600E+01
27	1.62	2.600E+01	2.600E+01	2.600E+01
26	1.56	2.600E+01	2.600E+01	2.600E+01
25	1.49	2.600E+01	2.600E+01	2.600E+01

***** F O R C E 2 - Rev 1.0 - 15 Feb. 1990 *****

TIME 16:52: 4 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0 X1 GAS VEL.
0 X1 (1) 12 4.87 13 5.36 14 5.84

X2 (1)				
24	1.43	2.600E+01	2.600E+01	2.600E+01
23	1.37	2.600E+01	2.600E+01	2.600E+01
22	1.30	2.600E+01	2.600E+01	2.600E+01
21	1.24	2.600E+01	2.600E+01	2.600E+01
20	1.17	2.600E+01	2.600E+01	2.600E+01
19	1.11	2.600E+01	2.600E+01	2.600E+01
18	1.05	2.600E+01	2.600E+01	2.600E+01
17	0.98	2.600E+01	2.600E+01	2.600E+01
16	0.92	2.600E+01	2.600E+01	2.600E+01
15	0.86	2.600E+01	2.600E+01	2.600E+01
14	0.79	2.600E+01	2.600E+01	2.600E+01
13	0.73	2.600E+01	2.600E+01	2.600E+01
12	0.67	2.600E+01	2.600E+01	2.600E+01
11	0.60	2.600E+01	2.600E+01	2.600E+01
10	0.54	2.600E+01	2.600E+01	2.600E+01
9	0.48	2.600E+01	2.600E+01	2.600E+01
8	0.41	2.600E+01	2.600E+01	2.600E+01
7	0.35	2.600E+01	2.600E+01	2.600E+01

6	0.29	2.600E+01	2.600E+01	2.600E+01
5	0.22	2.600E+01	2.600E+01	2.600E+01
4	0.16	2.600E+01	2.600E+01	2.600E+01
3	0.10	2.600E+01	2.600E+01	2.600E+01
2	0.03	2.600E+01	2.600E+01	2.600E+01
1	0.00	2.600E+01	2.600E+01	2.600E+01

***** F O R C E 2 - Rev 1.0 - 15 Feb, 1990 *****

TIME 16:52: 4 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0 X1 SOLIDS VEL.

0 X1 (1)	X2 (1)	2 0.00	3 0.49	4 0.97	5 1.46	6 1.95	7 2.43	8 2.92	9 3.41	10 3.90	11 4.38
33	1.97	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
32	1.94	0.000E+00	1.745E+00	0.000E+00	0.000E+00	0.000E+00	-1.172E-01	-1.933E-02	0.000E+00	0.000E+00	0.000E+00
31	1.87	0.000E+00	1.381E+00	0.000E+00	0.000E+00	0.000E+00	-1.059E-01	-1.849E-02	0.000E+00	0.000E+00	0.000E+00
30	1.81	0.000E+00	1.120E+00	4.606E-01	1.060E-01	1.037E-01	-8.195E-02	-1.724E-02	0.000E+00	0.000E+00	0.000E+00
29	1.75	0.000E+00	9.094E-01	4.131E-01	1.034E-01	8.483E-02	-6.385E-02	-1.497E-02	0.000E+00	0.000E+00	0.000E+00
28	1.68	0.000E+00	7.410E-01	3.674E-01	9.908E-02	6.965E-02	-4.954E-02	-1.224E-02	0.000E+00	0.000E+00	0.000E+00
27	1.62	0.000E+00	6.059E-01	3.247E-01	9.357E-02	5.734E-02	-3.823E-02	-9.306E-03	0.000E+00	0.000E+00	0.000E+00
26	1.56	0.000E+00	4.971E-01	2.855E-01	8.724E-02	4.731E-02	-2.928E-02	-6.298E-03	0.000E+00	0.000E+00	0.000E+00
25	1.49	0.000E+00	4.093E-01	2.499E-01	8.056E-02	3.902E-02	-2.216E-02	-3.443E-03	0.000E+00	0.000E+00	0.000E+00
24	1.43	0.000E+00	3.382E-01	2.181E-01	7.375E-02	3.223E-02	-1.662E-02	-7.567E-04	0.000E+00	0.000E+00	0.000E+00
23	1.37	0.000E+00	2.805E-01	1.897E-01	6.700E-02	2.666E-02	-1.229E-02	1.683E-03	0.000E+00	0.000E+00	0.000E+00
22	1.30	0.000E+00	2.333E-01	1.648E-01	6.045E-02	2.207E-02	-8.909E-03	3.855E-03	0.000E+00	0.000E+00	0.000E+00
21	1.24	0.000E+00	1.947E-01	1.429E-01	5.422E-02	1.827E-02	-6.227E-03	5.672E-03	0.000E+00	0.000E+00	0.000E+00
20	1.17	0.000E+00	1.630E-01	1.239E-01	4.837E-02	1.510E-02	-4.078E-03	7.088E-03	0.000E+00	0.000E+00	0.000E+00
19	1.11	0.000E+00	1.370E-01	1.075E-01	4.297E-02	1.251E-02	-2.393E-03	8.158E-03	0.000E+00	0.000E+00	0.000E+00
18	1.05	0.000E+00	1.156E-01	9.331E-02	3.802E-02	1.042E-02	-1.096E-03	8.924E-03	0.000E+00	0.000E+00	0.000E+00
17	0.98	0.000E+00	9.791E-02	8.108E-02	3.355E-02	8.758E-03	-1.303E-04	9.423E-03	0.000E+00	0.000E+00	0.000E+00
16	0.92	0.000E+00	8.329E-02	7.072E-02	2.947E-02	7.475E-03	5.418E-04	9.718E-03	0.000E+00	0.000E+00	0.000E+00
15	0.86	0.000E+00	7.135E-02	6.180E-02	2.595E-02	6.456E-03	9.876E-04	9.849E-03	0.000E+00	0.000E+00	0.000E+00
14	0.79	0.000E+00	6.143E-02	5.420E-02	2.285E-02	5.660E-03	1.270E-03	9.845E-03	0.000E+00	0.000E+00	0.000E+00
13	0.73	0.000E+00	5.334E-02	4.784E-02	2.005E-02	5.061E-03	1.367E-03	9.833E-03	0.000E+00	0.000E+00	0.000E+00
12	0.67	0.000E+00	4.672E-02	4.234E-02	1.770E-02	4.589E-03	1.367E-03	9.815E-03	0.000E+00	0.000E+00	0.000E+00
11	0.60	0.000E+00	4.133E-02	3.769E-02	1.564E-02	4.148E-03	1.366E-03	9.801E-03	0.000E+00	0.000E+00	0.000E+00
10	0.54	0.000E+00	3.691E-02	3.382E-02	1.384E-02	3.727E-03	1.365E-03	9.778E-03	0.000E+00	0.000E+00	0.000E+00
9	0.48	0.000E+00	3.333E-02	3.059E-02	1.229E-02	3.366E-03	1.365E-03	9.753E-03	0.000E+00	0.000E+00	0.000E+00
8	0.41	0.000E+00	3.047E-02	2.791E-02	1.095E-02	3.042E-03	1.364E-03	9.723E-03	0.000E+00	0.000E+00	0.000E+00
7	0.35	0.000E+00	2.818E-02	2.572E-02	9.833E-03	2.760E-03	1.364E-03	9.684E-03	0.000E+00	0.000E+00	0.000E+00
6	0.29	0.000E+00	2.640E-02	2.392E-02	8.933E-03	2.520E-03	1.363E-03	9.656E-03	0.000E+00	0.000E+00	0.000E+00
5	0.22	0.000E+00	2.506E-02	2.253E-02	8.222E-03	2.320E-03	1.363E-03	9.632E-03	0.000E+00	0.000E+00	0.000E+00
4	0.16	0.000E+00	2.407E-02	2.151E-02	7.676E-03	2.177E-03	1.362E-03	9.619E-03	0.000E+00	0.000E+00	0.000E+00
3	0.10	0.000E+00	2.344E-02	2.085E-02	7.305E-03	2.077E-03	1.362E-03	9.616E-03	0.000E+00	0.000E+00	0.000E+00
2	0.03	0.000E+00	2.314E-02	2.053E-02	7.125E-03	2.027E-03	1.362E-03	9.615E-03	0.000E+00	0.000E+00	0.000E+00
1	0.00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00

0 X1 (1)	X2 (1)	12 4.87	13 5.36	14 5.84
33	1.97	0.000E+00	0.000E+00	0.000E+00
32	1.94	0.000E+00	0.000E+00	0.000E+00
31	1.87	0.000E+00	0.000E+00	0.000E+00
30	1.81	0.000E+00	0.000E+00	0.000E+00
29	1.75	0.000E+00	0.000E+00	0.000E+00

28	1.68	0.000E+00	0.000E+00	0.000E+00
27	1.62	0.000E+00	0.000E+00	0.000E+00
26	1.56	0.000E+00	0.000E+00	0.000E+00
25	1.49	0.000E+00	0.000E+00	0.000E+00
24	1.43	0.000E+00	0.000E+00	0.000E+00
23	1.37	0.000E+00	0.000E+00	0.000E+00
22	1.30	0.000E+00	0.000E+00	0.000E+00
21	1.24	0.000E+00	0.000E+00	0.000E+00
20	1.17	0.000E+00	0.000E+00	0.000E+00

***** F O R C E 2 - Rev 1.0 - 15 Feb, 1990 *****

TIME 16:52: 5 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0 X1 SOLIDS VEL.

0 X1 (1) / X2 (1)		12 4.87	13 5.36	14 5.84
19	1.11	0.000E+00	0.000E+00	0.000E+00
18	1.05	0.000E+00	0.000E+00	0.000E+00
17	0.98	0.000E+00	0.000E+00	0.000E+00
16	0.92	0.000E+00	0.000E+00	0.000E+00
15	0.86	0.000E+00	0.000E+00	0.000E+00
14	0.79	0.000E+00	0.000E+00	0.000E+00
13	0.73	0.000E+00	0.000E+00	0.000E+00
12	0.67	0.000E+00	0.000E+00	0.000E+00
11	0.60	0.000E+00	0.000E+00	0.000E+00
10	0.54	0.000E+00	0.000E+00	0.000E+00
9	0.48	0.000E+00	0.000E+00	0.000E+00
8	0.41	0.000E+00	0.000E+00	0.000E+00
7	0.35	0.000E+00	0.000E+00	0.000E+00
6	0.29	0.000E+00	0.000E+00	0.000E+00
5	0.22	0.000E+00	0.000E+00	0.000E+00
4	0.16	0.000E+00	0.000E+00	0.000E+00
3	0.10	0.000E+00	0.000E+00	0.000E+00
2	0.03	0.000E+00	0.000E+00	0.000E+00
1	0.00	0.000E+00	0.000E+00	0.000E+00

***** F O R C E 2 - Rev 1.0 - 15 Feb, 1990 *****

TIME 16:52: 5 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0 X2 GAS VEL.

0 X1 (1) / X2 (1)		1 0.00	2 0.24	3 0.73	4 1.22	5 1.70	6 2.19	7 2.68	8 3.17	9 3.65	10 4.14
33	1.97	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
32	1.91	0.000E+00	-1.496E+02	-1.420E+01	0.000E+00	0.000E+00	6.490E+00	3.094E-01	5.989E-02	1.392E-02	2.982E-03
31	1.84	0.000E+00	-1.336E+02	-3.073E+01	0.000E+00	0.000E+00	1.307E+01	5.224E-01	1.089E-01	2.361E-02	3.540E-03
30	1.78	0.000E+00	-1.200E+02	-3.256E+01	-1.822E+00	2.403E-01	1.100E+01	8.355E-01	1.514E-01	3.537E-02	4.722E-03
29	1.71	0.000E+00	-1.077E+02	-3.331E+01	-3.286E+00	3.380E-01	9.253E+00	1.018E+00	1.886E-01	4.624E-02	7.655E-03
28	1.65	0.000E+00	-9.660E+01	-3.325E+01	-4.443E+00	3.297E-01	7.788E+00	1.105E+00	2.272E-01	5.567E-02	1.118E-02

27	1.59	0.000E+00	-8.662E+01	-3.256E+01	-5.323E+00	2.524E-01	6.551E+00	1.246E+00	2.461E-01	6.359E-02	1.324E-02
26	1.52	0.000E+00	-7.761E+01	-3.143E+01	-5.957E+00	1.332E-01	5.639E+00	1.213E+00	2.600E-01	6.829E-02	1.442E-02
25	1.46	0.000E+00	-6.949E+01	-2.998E+01	-6.375E+00	-3.710E-03	4.594E+00	1.153E+00	2.716E-01	7.094E-02	1.471E-02
24	1.40	0.000E+00	-6.215E+01	-2.832E+01	-6.607E+00	-1.471E-01	3.834E+00	1.073E+00	2.513E-01	7.148E-02	1.529E-02
23	1.33	0.000E+00	-5.552E+01	-2.678E+01	-6.688E+00	-2.839E-01	3.182E+00	9.434E-01	2.378E-01	6.852E-02	1.617E-02
22	1.27	0.000E+00	-4.952E+01	-2.461E+01	-6.644E+00	-4.077E-01	2.772E+00	8.380E-01	2.015E-01	6.437E-02	1.675E-02
21	1.21	0.000E+00	-4.411E+01	-2.270E+01	-6.498E+00	-5.161E-01	2.269E+00	7.306E-01	1.787E-01	5.847E-02	1.674E-02
20	1.14	0.000E+00	-3.922E+01	-2.108E+01	-6.273E+00	-6.085E-01	1.848E+00	6.174E-01	1.447E-01	5.315E-02	1.556E-02
19	1.08	0.000E+00	-3.480E+01	-1.896E+01	-5.990E+00	-6.832E-01	1.481E+00	4.602E-01	9.467E-02	4.693E-02	1.497E-02
18	1.02	0.000E+00	-3.081E+01	-1.745E+01	-5.664E+00	-7.434E-01	1.110E+00	3.425E-01	6.142E-02	4.132E-02	1.379E-02
17	0.95	0.000E+00	-2.721E+01	-1.547E+01	-5.308E+00	-7.876E-01	8.267E-01	2.325E-01	2.972E-02	3.776E-02	1.173E-02
16	0.89	0.000E+00	-2.397E+01	-1.388E+01	-4.928E+00	-8.168E-01	5.747E-01	1.074E-01	6.698E-02	3.598E-02	1.027E-02
15	0.83	0.000E+00	-2.105E+01	-1.239E+01	-4.537E+00	-8.209E-01	3.501E-01	1.002E-04	4.268E-02	3.306E-02	9.383E-03
14	0.76	0.000E+00	-1.839E+01	-1.100E+01	-4.283E+00	-8.125E-01	1.363E-01	3.871E-05	8.333E-02	3.218E-02	7.040E-03
13	0.70	0.000E+00	-1.599E+01	-9.707E+00	-3.753E+00	-7.865E-01	-1.669E-04	0.000E+00	1.016E-01	2.870E-02	3.240E-03
12	0.64	0.000E+00	-1.383E+01	-8.512E+00	-3.371E+00	-7.522E-01	-1.795E-04	0.000E+00	7.945E-02	2.521E-02	-5.624E-04
11	0.57	0.000E+00	-1.187E+01	-7.408E+00	-3.000E+00	-7.973E-01	-1.669E-04	0.000E+00	1.002E-01	1.949E-02	-5.244E-03
10	0.51	0.000E+00	-1.011E+01	-6.382E+00	-2.640E+00	-6.414E-01	-1.524E-04	0.000E+00	1.285E-01	1.482E-02	-7.878E-03
9	0.44	0.000E+00	-8.498E+00	-5.432E+00	-2.396E+00	-6.668E-01	-1.371E-04	0.000E+00	1.411E-01	9.026E-03	-1.139E-02
8	0.38	0.000E+00	-7.027E+00	-4.536E+00	-2.049E+00	-5.937E-01	-1.207E-04	0.000E+00	1.716E-01	3.272E-03	-1.404E-02
7	0.32	0.000E+00	-5.683E+00	-3.847E+00	-1.704E+00	-5.164E-01	-1.036E-04	0.000E+00	1.272E-01	4.999E-02	-1.723E-02
6	0.25	0.000E+00	-4.419E+00	-3.028E+00	-1.363E+00	-4.311E-01	-8.603E-05	0.000E+00	1.156E-01	4.051E-02	-5.193E-03
5	0.19	0.000E+00	-3.398E+00	-2.241E+00	-1.027E+00	-3.409E-01	-6.656E-05	0.000E+00	8.323E-02	3.342E-02	-8.429E-03
4	0.13	0.000E+00	-2.233E+00	-1.476E+00	-6.916E-01	-2.406E-01	-4.596E-05	0.000E+00	1.113E-02	2.256E-02	-2.660E-02
3	0.06	0.000E+00	-1.098E+00	-7.248E-01	-3.482E-01	-1.247E-01	-2.237E-05	0.000E+00	2.015E-03	1.017E-02	1.661E-02
2	0.00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00

0 X1 (1) 11 4.63 12 5.11 13 5.60 14 5.84

X2 (1)					
33	1.97	0.000E+00	0.000E+00	0.000E+00	0.000E+00
32	1.91	0.000E+00	0.000E+00	0.000E+00	0.000E+00
31	1.84	0.000E+00	0.000E+00	0.000E+00	0.000E+00
30	1.78	0.000E+00	0.000E+00	0.000E+00	0.000E+00
29	1.71	0.000E+00	0.000E+00	0.000E+00	0.000E+00
28	1.65	0.000E+00	0.000E+00	0.000E+00	0.000E+00
27	1.59	0.000E+00	0.000E+00	0.000E+00	0.000E+00
26	1.52	0.000E+00	0.000E+00	0.000E+00	0.000E+00
25	1.46	0.000E+00	0.000E+00	0.000E+00	0.000E+00
24	1.40	0.000E+00	0.000E+00	0.000E+00	0.000E+00
23	1.33	0.000E+00	0.000E+00	0.000E+00	0.000E+00
22	1.27	0.000E+00	0.000E+00	0.000E+00	0.000E+00
21	1.21	0.000E+00	0.000E+00	0.000E+00	0.000E+00
20	1.14	0.000E+00	0.000E+00	0.000E+00	0.000E+00
19	1.08	0.000E+00	0.000E+00	0.000E+00	0.000E+00

***** F O R C E 2 - Rev 1.0 - 15 Feb, 1990 *****

TIME 16:52: 5 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0 X2 GAS VEL.

0 X1 (1) 11 4.63 12 5.11 13 5.60 14 5.84

X2 (1)					
18	1.02	0.000E+00	0.000E+00	0.000E+00	0.000E+00
17	0.95	0.000E+00	0.000E+00	0.000E+00	0.000E+00
16	0.89	0.000E+00	0.000E+00	0.000E+00	0.000E+00

15	0.83	0.000E+00	0.000E+00	0.000E+00	0.000E+00
14	0.76	0.000E+00	0.000E+00	0.000E+00	0.000E+00
13	0.70	0.000E+00	0.000E+00	0.000E+00	0.000E+00
12	0.64	0.000E+00	0.000E+00	0.000E+00	0.000E+00
11	0.57	0.000E+00	0.000E+00	0.000E+00	0.000E+00
10	0.51	0.000E+00	0.000E+00	0.000E+00	0.000E+00
9	0.44	0.000E+00	0.000E+00	0.000E+00	0.000E+00
8	0.38	0.000E+00	0.000E+00	0.000E+00	0.000E+00
7	0.32	0.000E+00	0.000E+00	0.000E+00	0.000E+00
6	0.25	-2.439E-05	0.000E+00	0.000E+00	0.000E+00
5	0.19	-1.111E-05	0.000E+00	0.000E+00	0.000E+00
4	0.13	0.000E+00	0.000E+00	0.000E+00	0.000E+00
3	0.06	0.000E+00	0.000E+00	0.000E+00	0.000E+00
2	0.00	0.000E+00	0.000E+00	0.000E+00	0.000E+00

***** F O R C E 2 - Rev 1.0 - 15 Feb, 1990 *****

TIME 16:52: 5 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0 X2 SOLIDS VEL.

0	X1 (1)	1 0.00	2 0.24	3 0.73	4 1.22	5 1.70	6 2.19	7 2.68	8 3.17	9 3.65	10 4.14
	X2 (1)										
33	1.97	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
32	1.91	0.000E+00	-2.671E+00	-1.650E-01	0.000E+00	0.000E+00	9.011E-02	3.273E-03	0.000E+00	0.000E+00	0.000E+00
31	1.84	0.000E+00	-2.396E+00	-3.689E-01	0.000E+00	0.000E+00	1.815E-01	5.407E-03	0.000E+00	0.000E+00	0.000E+00
30	1.78	0.000E+00	-1.994E+00	-3.807E-01	-1.606E-02	3.074E-03	1.473E-01	9.031E-03	0.000E+00	0.000E+00	0.000E+00
29	1.71	0.000E+00	-1.666E+00	-3.751E-01	-2.826E-02	4.405E-03	1.203E-01	1.099E-02	0.000E+00	0.000E+00	0.000E+00
28	1.65	0.000E+00	-1.397E+00	-3.618E-01	-3.729E-02	4.317E-03	9.827E-02	1.188E-02	0.000E+00	0.000E+00	0.000E+00
27	1.59	0.000E+00	-1.174E+00	-3.415E-01	-4.376E-02	4.051E-03	8.060E-02	1.230E-02	0.000E+00	0.000E+00	0.000E+00
26	1.52	0.000E+00	-9.905E-01	-3.183E-01	-4.767E-02	2.861E-03	6.604E-02	1.174E-02	0.000E+00	0.000E+00	0.000E+00
25	1.46	0.000E+00	-8.362E-01	-2.921E-01	-4.990E-02	1.608E-03	5.343E-02	1.110E-02	0.000E+00	0.000E+00	0.000E+00
24	1.40	0.000E+00	-7.080E-01	-2.663E-01	-5.033E-02	7.123E-04	4.315E-02	1.010E-02	0.000E+00	0.000E+00	0.000E+00
23	1.33	0.000E+00	-5.998E-01	-2.392E-01	-4.968E-02	-1.399E-04	3.480E-02	9.019E-03	0.000E+00	0.000E+00	0.000E+00
22	1.27	0.000E+00	-5.094E-01	-2.144E-01	-4.808E-02	-8.900E-04	2.802E-02	7.572E-03	0.000E+00	0.000E+00	0.000E+00
21	1.21	0.000E+00	-4.327E-01	-1.902E-01	-4.598E-02	-1.724E-03	2.234E-02	5.953E-03	0.000E+00	0.000E+00	0.000E+00
20	1.14	0.000E+00	-3.665E-01	-1.680E-01	-4.315E-02	-2.279E-03	1.738E-02	4.541E-03	0.000E+00	0.000E+00	0.000E+00
19	1.08	0.000E+00	-3.112E-01	-1.478E-01	-4.024E-02	-2.738E-03	1.314E-02	3.253E-03	0.000E+00	0.000E+00	0.000E+00
18	1.02	0.000E+00	-2.648E-01	-1.298E-01	-3.690E-02	-3.077E-03	9.496E-03	2.137E-03	0.000E+00	0.000E+00	0.000E+00
17	0.95	0.000E+00	-2.243E-01	-1.128E-01	-3.422E-02	-3.319E-03	6.382E-03	1.266E-03	0.000E+00	0.000E+00	0.000E+00
16	0.89	0.000E+00	-1.890E-01	-9.805E-02	-3.036E-02	-3.728E-03	3.976E-03	5.844E-04	0.000E+00	0.000E+00	0.000E+00
15	0.83	0.000E+00	-1.605E-01	-8.499E-02	-2.726E-02	-3.864E-03	2.142E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00
14	0.76	0.000E+00	-1.347E-01	-7.318E-02	-2.497E-02	-3.764E-03	7.423E-04	0.000E+00	0.000E+00	0.000E+00	0.000E+00
13	0.70	0.000E+00	-1.134E-01	-6.305E-02	-2.134E-02	-3.558E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
12	0.64	0.000E+00	-9.513E-02	-5.415E-02	-1.889E-02	-3.313E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
11	0.57	0.000E+00	-7.955E-02	-4.603E-02	-1.674E-02	-3.158E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
10	0.51	0.000E+00	-6.615E-02	-3.897E-02	-1.448E-02	-2.718E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
9	0.44	0.000E+00	-5.455E-02	-3.281E-02	-1.254E-02	-2.425E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
8	0.38	0.000E+00	-4.456E-02	-2.720E-02	-1.057E-02	-2.115E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
7	0.32	0.000E+00	-3.575E-02	-2.222E-02	-8.606E-03	-1.804E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
6	0.25	0.000E+00	-2.758E-02	-1.741E-02	-6.867E-03	-1.496E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
5	0.19	0.000E+00	-2.035E-02	-1.289E-02	-5.202E-03	-1.071E-03	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
4	0.13	0.000E+00	-1.331E-02	-8.543E-03	-3.559E-03	-7.462E-04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
3	0.06	0.000E+00	-6.516E-03	-4.204E-03	-1.732E-03	-3.752E-04	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
2	0.00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00

```
0     X1 ( 1)  11   4.63    12   5.11    13   5.60    14   5.84
    X2 ( 1)
  33    1.97   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  32    1.91   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  31    1.84   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  30    1.78   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  29    1.71   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  28    1.65   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  27    1.59   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  26    1.52   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  25    1.46   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  24    1.40   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  23    1.33   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  22    1.27   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  21    1.21   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  20    1.14   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  19    1.08   0.000E+00  0.000E+00  0.000E+00  0.000E+00


                    *****   F O R C E 2 - Rev 1.0 -  15 Feb, 1990   *****

        TIME 16:52: 5          DATE : 9/26/90

1ITERATION    10                             Standard FLUFIX Test Case (ANL/EES-TM-3

0 X2 SOLIDS VEL.
0     X1 ( 1)  11   4.63    12   5.11    13   5.60    14   5.84
    X2 ( 1)
  18    1.02   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  17    0.95   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  16    0.89   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  15    0.83   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  14    0.76   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  13    0.70   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  12    0.64   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  11    0.57   0.000E+00  0.000E+00  0.000E+00  0.000E+00
  10    0.51   0.000E+00  0.000E+00  0.000E+00  0.000E+00
   9    0.44   0.000E+00  0.000E+00  0.000E+00  0.000E+00
   8    0.38   0.000E+00  0.000E+00  0.000E+00  0.000E+00
   7    0.32   0.000E+00  0.000E+00  0.000E+00  0.000E+00
   6    0.25   0.000E+00  0.000E+00  0.000E+00  0.000E+00
   5    0.19   0.000E+00  0.000E+00  0.000E+00  0.000E+00
   4    0.13   0.000E+00  0.000E+00  0.000E+00  0.000E+00
   3    0.06   0.000E+00  0.000E+00  0.000E+00  0.000E+00
   2    0.00   0.000E+00  0.000E+00  0.000E+00  0.000E+00


                    *****   F O R C E 2 - Rev 1.0 -  15 Feb, 1990   *****

        TIME 16:52: 5          DATE : 9/26/90

1ITERATION    10                             Standard FLUFIX Test Case (ANL/EES-TM-3

0 PRESSURE
0     X1 ( 1)   1   0.00     2   0.24     3   0.73     4   1.22     5   1.70     6   2.19     7   2.68     8   3.17     9   3.65    10   4.14
    X2 ( 1)
```

33	1.97	4.190E+04	4.060E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01
32	1.94	4.190E+04	5.923E+04	3.803E+04	0.000E+00	0.000E+00	9.087E+03	3.251E+03	8.656E+00	1.070E+01	1.143E+01
31	1.87	4.190E+04	5.671E+04	3.784E+04	0.000E+00	0.000E+00	9.172E+03	3.255E+03	8.708E+00	1.074E+01	1.147E+01
30	1.81	4.190E+04	5.434E+04	3.741E+04	2.542E+04	1.713E+04	9.346E+03	3.262E+03	8.801E+00	1.079E+01	1.151E+01
29	1.75	4.190E+04	5.226E+04	3.695E+04	2.539E+04	1.713E+04	9.492E+03	3.273E+03	8.930E+00	1.084E+01	1.154E+01
28	1.68	4.190E+04	5.045E+04	3.648E+04	2.535E+04	1.714E+04	9.615E+03	3.286E+03	9.089E+00	1.090E+01	1.157E+01
27	1.62	4.190E+04	4.886E+04	3.601E+04	2.529E+04	1.714E+04	9.717E+03	3.300E+03	9.284E+00	1.097E+01	1.161E+01
26	1.56	4.190E+04	4.747E+04	3.554E+04	2.522E+04	1.714E+04	9.804E+03	3.317E+03	9.497E+00	1.105E+01	1.164E+01
25	1.49	4.190E+04	4.624E+04	3.510E+04	2.514E+04	1.715E+04	9.878E+03	3.333E+03	9.722E+00	1.113E+01	1.168E+01
24	1.43	4.190E+04	4.516E+04	3.467E+04	2.505E+04	1.715E+04	9.938E+03	3.348E+03	9.964E+00	1.121E+01	1.171E+01
23	1.37	4.190E+04	4.422E+04	3.428E+04	2.497E+04	1.714E+04	9.988E+03	3.362E+03	1.019E+01	1.130E+01	1.175E+01
22	1.30	4.190E+04	4.338E+04	3.390E+04	2.488E+04	1.714E+04	1.003E+04	3.374E+03	1.040E+01	1.139E+01	1.179E+01
21	1.24	4.190E+04	4.265E+04	3.355E+04	2.479E+04	1.713E+04	1.007E+04	3.385E+03	1.058E+01	1.147E+01	1.183E+01
20	1.17	4.190E+04	4.200E+04	3.324E+04	2.470E+04	1.713E+04	1.010E+04	3.394E+03	1.075E+01	1.154E+01	1.187E+01
19	1.11	4.190E+04	4.143E+04	3.294E+04	2.461E+04	1.712E+04	1.012E+04	3.402E+03	1.089E+01	1.161E+01	1.191E+01
18	1.05	4.190E+04	4.093E+04	3.268E+04	2.453E+04	1.711E+04	1.014E+04	3.408E+03	1.098E+01	1.168E+01	1.196E+01
17	0.98	4.190E+04	4.049E+04	3.244E+04	2.446E+04	1.710E+04	1.015E+04	3.413E+03	1.103E+01	1.175E+01	1.199E+01
16	0.92	4.190E+04	4.011E+04	3.223E+04	2.438E+04	1.709E+04	1.016E+04	3.416E+03	1.106E+01	1.182E+01	1.202E+01
15	0.86	4.190E+04	3.977E+04	3.203E+04	2.432E+04	1.708E+04	1.017E+04	3.417E+03	1.147E+01	1.190E+01	1.204E+01
14	0.79	4.190E+04	3.948E+04	3.186E+04	2.425E+04	1.706E+04	1.018E+04	3.417E+03	1.143E+01	1.196E+01	1.206E+01
13	0.73	4.190E+04	3.922E+04	3.171E+04	2.420E+04	1.705E+04	1.018E+04	3.417E+03	1.180E+01	1.202E+01	1.206E+01
12	0.67	4.190E+04	3.900E+04	3.158E+04	2.415E+04	1.704E+04	1.018E+04	3.417E+03	1.201E+01	1.207E+01	1.206E+01
11	0.60	4.190E+04	3.881E+04	3.146E+04	2.410E+04	1.703E+04	1.018E+04	3.417E+03	1.217E+01	1.210E+01	1.205E+01
10	0.54	4.190E+04	3.865E+04	3.136E+04	2.406E+04	1.702E+04	1.018E+04	3.417E+03	1.202E+01	1.212E+01	1.202E+01
9	0.48	4.190E+04	3.851E+04	3.128E+04	2.402E+04	1.701E+04	1.018E+04	3.417E+03	1.217E+01	1.214E+01	1.201E+01
8	0.41	4.190E+04	3.839E+04	3.120E+04	2.399E+04	1.700E+04	1.018E+04	3.417E+03	1.219E+01	1.215E+01	1.199E+01
7	0.35	4.190E+04	3.830E+04	3.114E+04	2.396E+04	1.700E+04	1.018E+04	3.417E+03	1.213E+01	1.217E+01	1.197E+01
6	0.29	4.190E+04	3.822E+04	3.109E+04	2.394E+04	1.699E+04	1.018E+04	3.417E+03	1.204E+01	1.220E+01	1.196E+01
5	0.22	4.190E+04	3.816E+04	3.105E+04	2.392E+04	1.698E+04	1.018E+04	3.417E+03	1.202E+01	1.219E+01	1.209E+01
4	0.16	4.190E+04	3.812E+04	3.102E+04	2.391E+04	1.698E+04	1.018E+04	3.417E+03	1.241E+01	1.218E+01	1.183E+01
3	0.10	4.190E+04	3.809E+04	3.100E+04	2.390E+04	1.697E+04	1.018E+04	3.417E+03	1.239E+01	1.218E+01	1.150E+01
2	0.03	4.190E+04	3.807E+04	3.099E+04	2.389E+04	1.697E+04	1.018E+04	3.417E+03	1.238E+01	1.218E+01	1.157E+01
1	0.00	4.190E+04	4.060E+04	3.046E+04	2.370E+04	1.694E+04	1.018E+04	3.417E+03	3.394E+01	2.828E+01	2.262E+01

0 X1 (1) 11 4.63 12 5.11 13 5.60 14 5.84

X2 (1)

33	1.97	1.697E+01	1.131E+01	5.656E+00	0.000E+00
32	1.94	1.697E+01	1.131E+01	5.656E+00	0.000E+00
31	1.87	1.697E+01	1.131E+01	5.656E+00	0.000E+00
30	1.81	1.697E+01	1.131E+01	5.656E+00	0.000E+00
29	1.75	1.697E+01	1.131E+01	5.656E+00	0.000E+00
28	1.68	1.697E+01	1.131E+01	5.656E+00	0.000E+00
27	1.62	1.697E+01	1.131E+01	5.656E+00	0.000E+00
26	1.56	1.697E+01	1.131E+01	5.656E+00	0.000E+00
25	1.49	1.697E+01	1.131E+01	5.656E+00	0.000E+00
24	1.43	1.697E+01	1.131E+01	5.656E+00	0.000E+00
23	1.37	1.697E+01	1.131E+01	5.656E+00	0.000E+00
22	1.30	1.697E+01	1.131E+01	5.656E+00	0.000E+00
21	1.24	1.697E+01	1.131E+01	5.656E+00	0.000E+00
20	1.17	1.697E+01	1.131E+01	5.656E+00	0.000E+00

***** F O R C E 2 - Rev 1.0 - 15 Feb, 1990 *****

TIME 16:52; 5 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0 PRESSURE

0	X1 (1) X2 (1)	11 4.63	12 5.11	13 5.60	14 5.84
19	1.11	1.697E+01	1.131E+01	5.656E+00	0.000E+00
18	1.05	1.697E+01	1.131E+01	5.656E+00	0.000E+00
17	0.98	1.697E+01	1.131E+01	5.656E+00	0.000E+00
16	0.92	1.697E+01	1.131E+01	5.656E+00	0.000E+00
15	0.86	1.697E+01	1.131E+01	5.656E+00	0.000E+00
14	0.79	1.697E+01	1.131E+01	5.656E+00	0.000E+00
13	0.73	1.697E+01	1.131E+01	5.656E+00	0.000E+00
12	0.67	1.697E+01	1.131E+01	5.656E+00	0.000E+00
11	0.60	1.697E+01	1.131E+01	5.656E+00	0.000E+00
10	0.54	1.697E+01	1.131E+01	5.656E+00	0.000E+00
9	0.48	1.697E+01	1.131E+01	5.656E+00	0.000E+00
8	0.41	1.697E+01	1.131E+01	5.656E+00	0.000E+00
7	0.35	1.697E+01	1.131E+01	5.656E+00	0.000E+00
6	0.29	1.697E+01	1.131E+01	5.656E+00	0.000E+00
5	0.22	1.697E+01	1.131E+01	5.656E+00	0.000E+00
4	0.16	1.697E+01	1.131E+01	5.656E+00	0.000E+00
3	0.10	1.697E+01	1.131E+01	5.656E+00	0.000E+00
2	0.03	1.697E+01	1.131E+01	5.656E+00	0.000E+00
1	0.00	1.697E+01	1.131E+01	5.656E+00	0.000E+00

***** F O R C E 2 - Rev 1.0 - 15 Feb, 1990 *****

TIME 16:52: 5 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0 GAS VOID FRACT.

0	X1 (1) X2 (1)	1 0.00	2 0.24	3 0.73	4 1.22	5 1.70	6 2.19	7 2.68	8 3.17	9 3.65	10 4.14
33	1.97	1.000E+00	4.214E-01	4.200E-01	4.200E-01	4.200E-01	4.199E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
32	1.94	1.000E+00	4.214E-01	4.200E-01	4.200E-01	4.200E-01	4.199E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
31	1.87	1.000E+00	4.199E-01	4.200E-01	4.200E-01	4.200E-01	4.199E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
30	1.81	1.000E+00	4.198E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
29	1.75	1.000E+00	4.199E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
28	1.68	1.000E+00	4.199E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
27	1.62	1.000E+00	4.199E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
26	1.56	1.000E+00	4.199E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
25	1.49	1.000E+00	4.199E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
24	1.43	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
23	1.37	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
22	1.30	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
21	1.24	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
20	1.17	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
19	1.11	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
18	1.05	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
17	0.98	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
16	0.92	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
15	0.86	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
14	0.79	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
13	0.73	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
12	0.67	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
11	0.60	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
10	0.54	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00

9	0.48	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
8	0.41	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
7	0.35	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
6	0.29	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
5	0.22	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
4	0.16	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
3	0.10	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
2	0.03	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00
1	0.00	1.000E+00	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	4.200E-01	1.000E+00	1.000E+00	1.000E+00

0 X1 (1)		11 4.63	12 5.11	13 5.60	14 5.84
X2 (1)					
33	1.97	1.000E+00	1.000E+00	1.000E+00	1.000E+00
32	1.94	1.000E+00	1.000E+00	1.000E+00	1.000E+00
31	1.87	1.000E+00	1.000E+00	1.000E+00	1.000E+00
30	1.81	1.000E+00	1.000E+00	1.000E+00	1.000E+00
29	1.75	1.000E+00	1.000E+00	1.000E+00	1.000E+00
28	1.68	1.000E+00	1.000E+00	1.000E+00	1.000E+00
27	1.62	1.000E+00	1.000E+00	1.000E+00	1.000E+00
26	1.56	1.000E+00	1.000E+00	1.000E+00	1.000E+00
25	1.49	1.000E+00	1.000E+00	1.000E+00	1.000E+00
24	1.43	1.000E+00	1.000E+00	1.000E+00	1.000E+00
23	1.37	1.000E+00	1.000E+00	1.000E+00	1.000E+00
22	1.30	1.000E+00	1.000E+00	1.000E+00	1.000E+00
21	1.24	1.000E+00	1.000E+00	1.000E+00	1.000E+00
20	1.17	1.000E+00	1.000E+00	1.000E+00	1.000E+00

***** F O R C E 2 - Rev 1.0 - 15 Feb, 1990 *****

TIME 16:52: 6 DATE : 9/26/90

1ITERATION 10 Standard FLUFIX Test Case (ANL/EES-TM-3

0 GAS VOID FRACT.

0 X1 (1)		11 4.63	12 5.11	13 5.60	14 5.84
X2 (1)					
19	1.11	1.000E+00	1.000E+00	1.000E+00	1.000E+00
18	1.05	1.000E+00	1.000E+00	1.000E+00	1.000E+00
17	0.98	1.000E+00	1.000E+00	1.000E+00	1.000E+00
16	0.92	1.000E+00	1.000E+00	1.000E+00	1.000E+00
15	0.86	1.000E+00	1.000E+00	1.000E+00	1.000E+00
14	0.79	1.000E+00	1.000E+00	1.000E+00	1.000E+00
13	0.73	1.000E+00	1.000E+00	1.000E+00	1.000E+00
12	0.67	1.000E+00	1.000E+00	1.000E+00	1.000E+00
11	0.60	1.000E+00	1.000E+00	1.000E+00	1.000E+00
10	0.54	1.000E+00	1.000E+00	1.000E+00	1.000E+00
9	0.48	1.000E+00	1.000E+00	1.000E+00	1.000E+00
8	0.41	1.000E+00	1.000E+00	1.000E+00	1.000E+00
7	0.35	1.000E+00	1.000E+00	1.000E+00	1.000E+00
6	0.29	1.000E+00	1.000E+00	1.000E+00	1.000E+00
5	0.22	1.000E+00	1.000E+00	1.000E+00	1.000E+00
4	0.16	1.000E+00	1.000E+00	1.000E+00	1.000E+00
3	0.10	1.000E+00	1.000E+00	1.000E+00	1.000E+00
2	0.03	1.000E+00	1.000E+00	1.000E+00	1.000E+00
1	0.00	1.000E+00	1.000E+00	1.000E+00	1.000E+00

```
                         *****  F O R C E 2 - Rev 1.0 -  15 Feb, 1990   *****

         TIME 16:52: 6          DATE : 9/26/90

1ITERATION    10                           Standard FLUFIX Test Case (ANL/EES-TM-3

0
0                      ****  FILE I/O STATISTICS  ****
0          LOGICAL UNIT  4
           RECORD LENGTH   33          EFFICIENCY          1.00
           NO. RECORDS      0          HIT RATIO           1.00
           NO. BUFFERS  *****          NO. BUMPS              0
0          RECORD SLOTS     0          NO. FILE REQ           0
           RECORD SEARCH    1          NO. FILE READS         0
           NO. TEMP REQ        26      NO. FILE WRITES        0
           NO. TEMP LEFT        0
0PHASE:  1
0Inlet Flow    =  0.000000E+00
 Mass Sources =  0.000000E+00
 Outlet Flow   =  0.000000E+00
0PHASE:  2
0Inlet Flow    =  0.000000E+00
 Mass Sources =  0.000000E+00
 Outlet Flow   =  0.000000E+00
```

+++++++++++++ FORCE2 Log File , Initial Run +++++++++++++

```
   TIME,TOT.STEPS, TRYS :     0.00050        5      29
     MAX RESID, I,J,K   :    5.21022E-08    -2    32     1
 SOLIDS MASS, % ,GAS IN, GAS OUT:    7.96510E+02    2.45210E-04    1.06362E+00    6.06209E-01

   TIME,TOT.STEPS, TRYS :     0.00100       10      25
     MAX RESID, I,J,K   :    5.11577E-08    -2    27     1
 SOLIDS MASS, % ,GAS IN, GAS OUT:    7.96510E+02    2.52873E-04    1.06362E+00    6.06208E-01

   TIME,TOT.STEPS, TRYS :     0.00100       10      25
     MAX RESID, I,J,K   :    5.11577E-08    -2    27     1
 SOLIDS MASS, % ,GAS IN, GAS OUT:    7.96510E+02    2.52873E-04    1.06362E+00    6.06208E-01
```

+++++++++++++++ Post Processor Boundary Data File +++++++++++++++

```
3    1    2    5
     1    1    1    5    2     0.0       0.0
                           4     0.0      19.685
                           3    58.44     19.685
                           1    58.44      0.0
                           1     0.0       0.0
     1    1    2    5    1     9.74     18.415
                           2     9.74     19.685
                           1    19.48     19.685
                           1    19.48     18.415
                           1     9.74     18.415
```

Geometry File for 33X14 Standard FLUFIX Problem

APPENDIX C

FORCE2 AND POST-PROCESSOR FILES FOR THE CAPTF 3-D BUNDLE PROBLEM

Appendix C

FORCE2 and Post-Processor Files for the CAPTF 3-D Bundle Problem

The FORCE2 input data for the initial and restart runs and the post-processor boundary data file are given in this appendix. These files were developed during runs of the CAPTF 3-D bundle problem.

++++++++++++++++++++   FORCE2 Input , Initial Run +++++++++++++++++

```
$       3-D Model,  Tubes modelled with 2X2 Blocked cells
$         Bed Modelled with  Blocked Cells
$
$    Umf = 27 cm/s ( Approx)
$    Utest = 52.0 cm/s
$
$   Bed Material : .            Rho = 2.49, D= (425 +600)/2 Microns
$
$         Model Includes Windbox & Distributor Plate
$
$
SAVE
TITLE   3-D CAPTF Bed with Tubes
GEOMETRY
$
$  -----   G e o m e t r y   P a r a g r a p h   -------
$
CARTESIAN  35  , 14 , 5
DX1      15.0 , 3.0,
         2.54, 2.54, 2.54, 2.54, 2.54,
         2.54, 2.54, 2.54, 2.54, 2.54,
         2.54, 2.54, 2.54, 2.54, 2.54,
         2.54, 2.54, 2.54, 2.54, 2.54,
         2.54, 2.54, 2.54, 2.54,
         5.08, 5.08, 5.08, 5.08, 5.08,
         5.08, 5.08
DX2      2.54, 2.54, 2.54, 2.54, 2.54,
         2.54, 2.54, 2.54, 2.54, 2.54,
         2.54, 2.54
DX3      10.16, 10.16, 10.16
$
$
$
 BOUNDARY  X2  LOWER  DIRICHLET
 BOUNDARY  X2  UPPER  DIRICHLET
 BOUNDARY  X3  LOWER  DIRICHLET
 BOUNDARY  X3  UPPER  DIRICHLET
 BLOCK  X2   0.0       0.0   114.52  0.0   30.48
 BLOCK  X2  30.48      0.0   114.52  0.0   30.48
 BLOCK  X3   0.0       0.0   114.52  0.0   30.48
 BLOCK  X3  30.48      0.0   114.52  0.0   30.48
$
$      Tube # 1 ( Half Tube on Right Wall)
$
BLOCK CELLS   38.32   43.4      0.0     2.54   0.0   30.48
$
$      Tube # 2 ( Lower Row, Center of Bed)
$
BLOCK CELLS   38.32    43.4    12.7    17.78   0.0   30.48
$
$      Tube # 3 (Half Tube on Left Wall )
$
BLOCK CELLS   38.32    43.4    27.94   30.48   0.0   30.48
$
```

```
$      Tube # 4 ( Upper Row )
$
BLOCK CELLS   45.94    51.02   5.08    10.16     0.0    30.48
$
$      Tube # 5 (Upper Row )
$
BLOCK CELLS   45.94    51.02   20.32   25.40   0.0    30.48
$ --------------------------------------------

PROPERTIES
$
$  -----   P r o p e r t i e s   P a r a g r a p h   -------
$
$    Gas   Density   &   Viscosity
$
DENSITY    GAS    IDEAL
MOLECULAR  WEIGHT        1.0
GAS        CONSTANT     2.87E06
VISCOSITY  GAS          1.82E-04
$
$  Solids   Density   &  Viscosity ( Specify Gas Viscosity )
$
DENSITY    SOLIDS    2.490 ,  0.0 ,  0.0 ,  0.0
VISCOSITY  SOLIDS    1.0
$
$    Gas  &  Solids  Reference  Densities  ( Used only for steady soln)
$
GAS     REFERENCE  DENSITY    .00117
SOLIDS  REFERENCE  DENSITY    2.49
$
$    Reference  Pressure (Atmospheric)
$
PRESSURE  1.013E06,  34, 2
$
$     Reference and  Absolute Temperatures
$
REFERENCE  TEMPERATURE    0.0
TEMPERATURE              300.0
$
SECOND VISCOSITY COEFFICIENT GAS     .6667
SECOND VISCOSITY COEFFICIENT SOLIDS  .6667

FLOW
$
$  -----   F l o w   P a r a g r a p h   -------
$
TRANSIENT
GRAVITY   980.621 ,  0.0  , 0.0
$
$
DIAMETER    .0513
SPHERISITY    1.0
SCALAR  DRAG
MODEL  B
$
$    Solid  Stress  Parameters
```

```
SOLIDS  STRESS  PARAMETER  SLOPE   600.
SOLIDS  STRESS  PARAMETER  EPSTAR    .40
$
$  ----------------------------------------
$
$    Dieributed Resistance Model for Distributor Plate
$
$       X1-Direction
$
DISTRIBUTED RESISTANCE GAS
     LOSS FACTOR   906.0
     X1   RANGE    15.1    15.5  0.0    30.48    0.0    30.48
$
$       X2-Direction
$
DISTRIBUTED RESISTANCE GAS
     LOSS FACTOR   100000.0
     X2   RANGE    15.1     17.9  0.0    30.48  0.0    30.48
$
DISTRIBUTED RESISTANCE GAS
     LOSS FACTOR   100000.0
     X3   RANGE    15.1     17.9  0.0    30.48  0.0    30.48
$
$
$        Windbox Resistance = Ergun's Correlation
$          with : i) stationary solids,ii) Dp = .3 cm,
$               iii) Void = .4
$
$        X1-Direction
$
USER DEFINED RESISTANCE GAS
   RESISTANCE NUMBER  3
   X1  RANGE   10.0     14.    0.0     30.48  0.0    30.48
$
$        X2-Direction
$
USER DEFINED RESISTANCE GAS
   RESISTANCE NUMBER  3
   X2  RANGE   0.0     14.5    0.0     30.48  0.0    30.48
$
$        X3-Direction
$
USER DEFINED RESISTANCE GAS
   RESISTANCE NUMBER  3
   X3  RANGE   0.0     14.5    0.0     30.48  0.0    30.48

CONTROL
$
$  -----   C o n t r o l   P a r a g r a p h   -------
$
$       Print  & Data Output Control
$
PRINT  OFF
PRINT   P     FINAL
```

```
PRINT   U1G   FINAL
PRINT   U1S   FINAL
PRINT   U2G   FINAL
PRINT   U2S   FINAL
PRINT   U3G   FINAL
PRINT   U3S   FINAL
$
$
$  Program  Execution  &  Timestep
$
TIME STEP   .0001
MAXIMUM  TIME STEPS    10
$
$   Iteration  &  Relaxation  Parameters
$
MAXIMUM   MICRO  ITERATIONS    10
MAXIMUM   MACRO  ITERATIONS    100
$
$
$     Convergence & Solution  Parameters
$
MASS  CONVERGENCE  CRITERIA    .00005
MAXIMUM   VOID                 .999
POST PROCESSOR DATA
     SAVE  ALL                  2000

PERMEABILITIES
$
$  -----   P o r o s i t y / P e r m e a b i l i t y   -----------
$                  P a r a g r a p h
$
$    Set  to  1.0
$
SET1
    SET2
      SET3
        POROSITY        1.0
SET1
    SET2
      SET3
          X1-PERMEABILITY  1.0
          X2-PERMEABILITY  1.0
          X3-PERMEABILITY  1.0
$
$    Set Distributor Plate Perm very low in X2 & X3 Direction
$
SET1 3, 3
    SET2 2, 13
       SET3   2, 5
          X2-PERMEABILITY  0.001
          X3-PERMEABILITY  0.001
SET1  2, 2
   SET2 2, 13
      SET3 2, 5
          POROSITY   0.4
```

```
$
$  -----   I n i t i a l i z a t i o n   P a r a g r a p h   -------
$
$
$       Initialize Pressure Field Based on Min Fluidization Condition
$
MINIMUM  FLUIDIZATION   1
$
$     Set  velocities  & voids throughout  field
$
SET1
  SET2
    SET3
         U1G      0.0
         U1S      0.0
         U2G      0.0
         U2S      0.0
         U3G      0.0
         U3S      0.0
         VFRG     1.0
         VFRS     0.0
$
$     Set   Inlet   Pressure
$
SET1    1,  1
     SET2   1,  14
        SET3   1,  5
            P        76200.
$
$     Set  Inflow  Gas  Velocity to Plenum
$
$
SET1   1, 2
  SET2     2,  13
     SET3     2,  4
         U1G    27.0
$ ----------------------------------------
$
$
$     Set  Inflow  Gas  Velocities in Plenum & Plate
$
SET1   3, 4
  SET2    2, 13
     SET3    2, 4
         U1G    27.0
$   +++++++++++++++++++++++++++++++++++++
$
$     Set   Gas  Velocities  in Bed
$
SET1  5, 20
  SET2  2, 13
     SET3  2, 4
         U1G  62.0
$
```

```
$       Set    Gas    Velocities    in    Above Bed
$
SET1  21, 35
  SET2  2, 13
     SET3  2, 4
          U1G     27.0
$
$
$       Initialize  Void  Fractions in Bed
$
SET1  4,  19
  SET2  1,   14
     SET3  1,   5
          VFRG  .404
          VFRS  .596
$
$      Set Velocities & Voids at Blocked cells
$
SET1  12, 14
  SET2 2,  2
     SET3 2,  4
         U1G   0.0
$
SET1  12, 14
  SET2 7, 8
     SET3 2,  4
         U1G   0.0
$
SET1  12, 14
  SET2 13, 13
     SET3 2,  4
         U1G   0.0
$
SET1  15, 17
  SET2 4,  5
     SET3 2,  4
         U1G   0.0
$
SET1  15, 17
  SET2 10, 11
     SET3 2,  4
         U1G   0.0
$
$
SET1  12, 13
  SET2 2,  2
     SET3 2,  4
         VFRG  0.0
$
SET1  12, 13
  SET2 7,  8
     SET3 2,  4
         VFRG  0.0
$
SET1  12, 13
  SET2 13,  13
```

```
    VFRG  0.0
$
$
SET1  15, 16
  SET2 4 ,  5
      SET3 2,  4
        VFRG  0.0
$
SET1  15, 16
  SET2 10 ,  11
     SET3 2,  4
          VFRG  0.0
$
$ -------------------------------------------------

FLOFLAGS
$
$ -----   C e l l   F l a g   P a r a g r a p h   -------
$
$    Set   all  to  free flow  cells  IFLO = 1
SET1
  SET2
    SET3
           GAS      1
           SOLIDS   1
$
$  Inflow  Cells For Gas Phase  IFLO = 2
$
SET1  1, 1
  SET2  2, 13
    SET3 2, 4
           GAS     2
$
$   Constant Pressure Outflow Cells For Gas Phase IFLO = 4
$
SET1  35 , 35
  SET2  2, 13
    SET3 2, 4
           GAS   4
$
$  No Flow Solids Cells in Windbox & Distributor Plate
$
SET1  2, 3
  SET2 2, 13
    SET3 2, 4
          SOLIDS  0
$
$  No Flow Cells For Solids Phase at Non-Blocked Surfaces, IFLO = 0
$
SET1  1, 1
  SET2 2, 13
    SET3 2, 4
          SOLIDS   0
SET1  35 , 35
  SET2 2, 13
```

```
           SOLIDS    0
 $
 $      Gas & Solids Slip/ No-Slip Conditons Along Boundaries
 $
 $      Lower  X2 - Boundary
 $             Gas    :  No-Slip  ,      IFLO = -3
 $             Solids :  Partial-Slip    IFLO = -2
 $
 SET1 1, 35
   SET2 1, 1
     SET3 2, 4
           GAS       -3
           SOLIDS    -2
 $
 $      Upper  X2 - Boundary
 $             Gas    :  No-Slip ,            IFLO = -3
 $             Solids :  Partial Slip ,       IFLO = -2
 $
 SET1 1, 35
   SET2  14, 14
     SET3 2, 4
           GAS       -3
           SOLIDS    -2
 $
 $      Upper  X3 - Boundary
 $             Gas    :  No-Slip ,            IFLO = -3
 $             Solids :  Partial Slip ,       IFLO = -2
 $
 SET1 1, 35
   SET2  1, 14
     SET3 5, 5
           GAS       -3
           SOLIDS    -2
 $
 $      Lower  X3 - Boundary
 $             Gas    :  No-Slip ,            IFLO = -3
 $             Solids :  Partial Slip ,       IFLO = -2
 $
 SET1 1, 35
   SET2  1, 14
     SET3 1, 1
           GAS       -3
           SOLIDS    -2
 $
 $      Boundary Conditions at Tubes
 $
 $             Gas    :  No-Slip ,            IFLO = -3
 $             Solids :  Partial-Slip ,       IFLO = -2
 $
 $        Tube # 1
 SET1 12, 13
   SET2 2, 2
      SET3 2, 4
         GAS      -3
         SOLIDS   -2
```

```
$         Tube # 2
SET1 12, 13
  SET2 7 , 8
    SET3 2, 4
        GAS      -3
        SOLIDS   -2
$
$
$         Tube # 3
SET1 12, 13
  SET2  13, 13
    SET3 2, 4
        GAS      -3
        SOLIDS   -2
$
$
$         Tube # 4
SET1 15, 16
  SET2  4, 5
    SET3 2. 4
        GAS      -3
        SOLIDS   -2
$
$
$         Tube # 5
SET1 15. 16
  SET2   10, 11
    SET3 2, 4
        GAS      -3
        SOLIDS   -2
$
$      End    Data Set
```

++++++++++++++++++++ FORCE2 Input , Restart Run ++++++++++++++++

```
$      3-D Model,  Tubes modelled with 2X2 Blocked cells
$        Bed Modelled with  Blocked Cells
$
$   Umf = 27 cm/s ( Approx)
$   Utest = 52:0 cm/s
$
$  Bed Material :     Rho = 2.49, D= (425 +600)/2 Microns
$
$        Model Includes Windbox & Distributor Plate
$
$
RESTART SAVE
TITLE   3-D CAPTF Bed with Tubes

FLOW
$
$  -----   F l o w   P a r a g r a p h   -------
$
TRANSIENT
$

CONTROL
$
$  -----   C o n t r o l   P a r a g r a p h   -------
$
$        Print  & Data Output Control
$
PRINT  OFF
PRINT   U1G   FINAL
PRINT   U1S   FINAL
PRINT   U2G   FINAL
PRINT   U2S   FINAL
PRINT   U3G   FINAL
PRINT   U3S   FINAL
PRINT   VFRG   FINAL
$
$
$  Program  Execution  &  Timestep
$
TIME STEP   .0001
MAXIMUM  TIME STEPS     30700
POST PROCESSOR DATA
     SAVE  ALL                    3
     SAVE  SELECTED VARAIABLES    2
 STORE  P      AT   2, 9, 3
 STORE  P      AT  17, 9, 3
 STORE  U1G    AT   4, 9, 3
 STORE  U1G    AT   7, 6, 2
 STORE  U1S    AT   7, 6, 2
 STORE  VFRG   AT  11, 8, 4
 STORE  VFRG   AT  14, 9, 2
 STORE  U3S    AT   8, 8, 4

INITIAL
```

```
$
$ -----   I n i t i a l i z a t i o n   P a r a g r a p h   -------
$
$
$       Initialize Pressure Field Based on Min Fluidization Condition
$
$
$
$      Set   Inlet   Pressure
$
SET1   1,  1
     SET2   1,  14
        SET3   1,  5
            P        76200.
$
$      Set  Inflow  Gas  Velocity to Plenum
$
$
SET1   1, 2
  SET2      2,  13
     SET3      2,  4
         U1G     52.0
$ ------------------------------------------
$
$
$     End of Data
```

+++++++++++++++ Post Processor Boundary Data File +++++++++++++++

	1	11	1	5	1	0.0	0.0	1	0.0	30.48
					1	30.48	30.48	1	30.48	0.0
					1	0.0	0.0			
	12	13	1	5	1	0.0	0.0	1	0.0	30.48
					1	2.54	30.48	1	2.54	0.0
					1	0.0	0.0			
	12	13	2	5	1	12.7	0.0	1	12.7	30.48
					1	17.78	30.48	1	17.78	0.0
					1	12.7	0.0			
	12	13	3	5	1	27.94	0.0	1	27.94	30.48
					1	30.48	30.48	1	30.48	0.0
					1	27.94	0.0			
	12	13	4	5	1	0.0	0.0	1	0.0	30.48
					1	30.48	30.48	1	30.48	0.0
					1	0.0	0.0			
	14	14	1	5	1	0.0	0.0	1	0.0	30.48
					1	30.48	30.48	1	30.48	0.0
					1	0.0	0.0			
	14	14	2	5	1	0.0	0.0	1	0.0	30.48
					1	30.48	30.48	1	30.48	0.0
					1	0.0	0.0			
	15	16	1	5	1	5.08	0.0	1	5.08	30.48
					1	10.16	30.48	1	10.16	0.0
					1	5.08	0.0			
	15	16	2	5	1	20.32	0.0	1	20.32	30.48
					1	25.4	30.48	1	25.4	0.0
					1	20.32	0.0			
	15	16	3	5	1	0.0	0.0	1	0.0	30.48
					1	30.48	30.48	1	30.48	0.0
					1	0.0	0.0			
	17	35	1	5	1	0.0	0.0	1	0.0	30.48
					1	30.48	30.48	1	30.48	0.0
					1	0.0	0.0			
2	9	3	5							
	2	2	1	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			
	2	2	2	5	1	15.0	0.0	1	15.0	30.48
					1	18.0	30.48	1	18.0	0.0
					1	15.0	0.0			
	2	2	3	5	1	38.32	0.0	1	38.32	30.48
					1	43.4	30.48	1	43.4	0.0
					1	38.32	0.0			
	3	3	1	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			
	3	3	2	5	1	15.0	0.0	1	15.0	30.48
					1	18.0	30.48	1	18.0	0.0
					1	15.0	0.0			
	4	5	1	5	1	45.94	0.0	1	45.94	30.48
					1	51.02	30.48	1	51.02	0.0
					1	45.94	0.0			
	4	5	2	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			

					1	15.0	0.0			
	6	6	1	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			
	6	6	2	5	1	15.0	0.0	1	15.0	30.48
					1	18.0	30.48	1	18.0	0.0
					1	15.0	0.0			
	7	8	1	5	1	38.32	0.0	1	38.32	30.48
					1	43.4	30.48	1	43.4	0.0
					1	38.32	0.0			
	7	8	2	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			
	7	8	3	5	1	15.0	0.0	1	15.0	30.48
					1	18.0	30.48	1	18.0	0.0
					1	15.0	0.0			
	9	9	1	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			
	9	9	2	5	1	15.0	0.0	1	15.0	30.48
					1	18.0	30.48	1	18.0	0.0
					1	15.0	0.0			
	10	11	1	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			
	10	11	2	5	1	15.0	0.0	1	15.0	30.48
					1	18.0	30.48	1	18.0	0.0
					1	15.0	0.0			
	10	11	3	5	1	45.94	0.0	1	45.94	30.48
					1	51.02	30.48	1	51.02	0.0
					1	45.94	0.0			
	12	12	1	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			
	12	12	2	5	1	15.0	0.0	1	15.0	30.48
					1	18.0	30.48	1	18.0	0.0
					1	15.0	0.0			
	13	13	1	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			
	13	13	2	5	1	15.0	0.0	1	15.0	30.48
					1	18.0	30.48	1	18.0	0.0
					1	15.0	0.0			
	13	13	3	5	1	38.32	0.0	1	38.32	30.48
					1	43.4	30.48	1	43.4	0.0
					1	38.32	0.0			
3	1	7	5							
	1	5	1	5	2	0.0	0.0	1	0.0	30.48
					3	114.52	30.48	1	114.52	0.0
					2	0.0	0.0			
	1	5	2	5	1	15.0	0.0	1	15.0	30.48
					1	18.0	30.48	1	18.0	0.0
					1	15.0	0.0			
	1	5	3	5	1	38.32	0.0	1	38.32	2.54

				1	43.4	2.54	1	43.4	0.0
				1	38.32	0.0			
1	5	4	5	1	38.32	12.7	1	38.32	17.78
				1	43.4	17.78	1	43.4	12.7
				1	38.32	12.7			
1	5	5	5	1	38.32	27.94	1	38.32	30.48
				1	43.4	30.48	1	43.4	27.94
				1	38.32	27.94			
1	5	6	5	1	45.94	5.08	1	45.94	10.16
				1	51.02	10.16	1	51.02	5.08
				1	45.94	5.08			
1	5	7	5	1	45.94	20.32	1	45.94	25.4
				1	51.02	25.4	1	51.02	20.32
				1	45.94	20.32			

CAPTF 3-D Bundle Geometry File

Notice

The FORCE2 computer program may be obtained from:

Energy Science and Technology Software Center
P. O. Box 1020
Oak Ridge, TN 37833

Telephone (615) 576-2606
FAX (615) 576-2865
E-Mail:ESTSC@ADONIS.OSTI.GOV

DATE FILMED

6/2/94

END