# PHOENICS 培 训 教 程

## CHAM-上海办事处

# Tutorials

## General


1. Flow through a simple labyrinth 4
2. Duplication of objects 13
3. Cylindrical obstacles 22
4. Array and group settings 29
5. Fans and inlets 37
6. A jet pump 48
7. Flow Simulation within a Pipe 60
8. Flow Around Buildings 70
9. Flow Over Heated Bricks 82
10. Conjugate heat transfer 90
11. Introducing a contaminant 93
12. Modifying domain properties 103


## PARSOL and Fine-Grid Embedding


1. PARSOL 114
2. Fine-grid embedding 121


## Transient


1. Heat conduction in a bar 126
2. Rupture of a diaphragm 135


## Body-Fitted Co-ordinates


1. BFC 1 - Points and Lines 144
2. BFC 2 - Making Frames 155
3. BFC 3 - Frame matching 163
4. BFC 4 - Copy/Transfer 173
5. BFC 5 - Boundary Conditions 183
6. BFC 6 - A Simple Case 190
7. Multi-blocking 1 T-Junction, natural link 202
8. Multi-blocking 2 3 Blocks, unnatural link 215


## Polar Co-ordinates


1. Flow in a Cylindrical Jet Pump 229
2. 3D Swirling Flow Through an Orifice Plate 238

## Radiation Models - IMMERSOL


1. IMMERSOL: 2 bodies in a duct 249
2. IMMERSOL in an electronics box 257


## Turbulence Models


1. Turbulence modelling 266
2. The LVEL turbulence model 275


## Multi-phase


1. IPSA workshop 1 Two-Phase Flow (Solids in Air) 279
2. IPSA workshop 2 Introducing non-standard interphase transport correlations using PLANT 284
3. SEM workshop 1 Water pouring into a bund 295
4. SEM workshop 2 A lighter fluid penetrating heavier liquid 302
5. Shallow-water workshop 1 Local depth in an open U-bend 309
6. Shallow-water workshop 2 A whirlpool in a shallow pond: viscous flow 313
7. GENTRA: Particle-tracking Flow of solid particles over a backward facing step 319


## Moving Bodies - MOFOR


1. Overview of the BVH file 337
2. MOFOR Example - Agitated tank 354


## FORTRAN Coding


1. Using PLANT 366
2. Using GROUND 377


## Miscellaneous

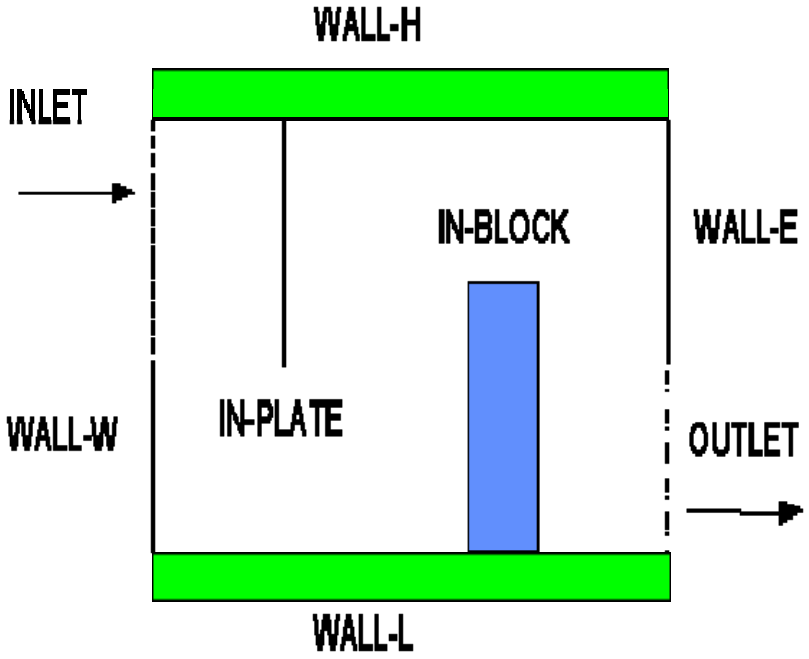

1. Boundary conditions and sources 382
2. Using PHOTON 393


## Combustion


1. Simple Chemical Reaction Scheme 402

# WORKSHOP - Simulation of a Labyrinth Flow.

This workshop example shows how to set up and run the simulation of a simple flow in PHOENICS-VR.
The example is a 2-dimensional labyrinth flow. The fluid is a gas and the flow is laminar with heat exchange.
Its geometry is shown in the figure below.




WALL-W is an adiabatic boundary wall

IN-PLATE is an internal plate

IN-BLOCK is an adiabatic internal blockage of a heat conducting material

WALL-H is an adiabatic boundary wall of a heat conducting material

WALL-E is an adiabatic boundary wall

WALL-L is a boundary wall of a heat conducting material, which is also a heat source.

## Accessing PHOENICS-VR

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation .

## Within VR Editor.

**Set the domain size, and activate solution of the required variables**:

Click on 'Main Menu' and then on 'Geometry'.

Change the X-Domain Size to 2.0 m.

Click 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Set the 'solution for velocities and pressure' ON.

Click on 'energy equation', and select 'Temperature' as energy formulation. Click on 'OK'.

Click on 'Total' whereby it changes to 'Static'.

Click on 'Top menu', then on 'OK' to exit the Main Menu.

**Create the WALL-H object**:

Click on 'New Object'.

Change name to WALL-H (which stands for high wall).

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 2.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.95 Zsize: 0.05

Select Type: Blockage (default).

To change the material, click on 'Attributes', click on ‘Types’,

click on SOLIDS then 'OK';, and then on 'ALUMINIUM at 27 deg C' and 'OK';.

Click on 'OK' to exit the Attributes menu,

and on 'OK' to close the Object Dialogue Box.

**Create the WALL-L object**:

Click on 'New Object'.

Change name to WALL-L (which stands for low wall).

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 2.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 0.05

Define Type: Blockage (default).

Click on 'Attributes', select material as ALUMINIUM as above.

Set a HEAT SOURCE of 100 W. Click on ‘Adiabatic’, and select ‘Fixed heat Flux’. Click on Value and enter 100.

Click on 'OK' to return to the Object Dialogue Box, and on 'OK' to close the Object Dialogue Box.

**Create the WALL-E object**:

Click on 'New Object'.

Change name to WALL-E (which stands for east wall).

Set POSITION and SIZE of object as:

Xpos: 2.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.5 Zsize: 0.5

Define Type: Plate.

Click on 'OK' to close the Object Dialogue Box.

**Create the WALL-W object**:

Click on 'New Object'.

Change name to WALL-W (which stands for west wall).

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 0.5

Define Type: Plate.

Click on 'OK' to close the Object Dialogue Box.

**Create the IN-PLATE object**:

Click on 'New Object'.

Change name to IN-PLATE (which stands for interior plate).

Set POSITION and SIZE of object as:

Xpos: 0.5 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.3 Zsize: 0.65

Define Type: Plate.

Click on 'OK' to close the Object Dialogue Box.

**Create the IN-BLOCK object**:

Click on 'New Object'.

Change name to IN-BLOCK (which stands for interior block).

Set POSITION and SIZE of object as:

Xpos: 1.0 Xsize: 0.4

Ypos: 0.0 Ysize: 1.0

Zpos: 0.05 Zsize: 0.65

Define Type: Blockage (default).

Click on 'Attributes', then on ‘Other materials’, select ‘Solids’ then COPPER.

Click on 'OK' to return to the Object Dialogue Box,

and on 'OK' to close the Object Dialogue Box.

**Create the INLET object**:

Click on 'New Object'.

Change name to INLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.5 Zsize: 0.45

Define Type: Inlet.

Click on 'Attributes' and set velocity in X-direction at 0.05 m/s.

Set temperature at 20 degrees C for the inflowing air.

Click on 'OK' to close the Attributes menu,

and on 'OK' in the Object Dialogue Box.

**Create the OUTLET object**:

Click on 'New Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 2.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.05 Zsize: 0.45

Define Type: Outlet.

Leave the default values in the Attributes dialogue box.

Click on 'OK' to exit the Object Dialogue Box.

**Set the grid**:

Click on the 'Mesh toggle' button. The default mesh (shown as orange lines) will appear on the screen. Click anywhere on the image, and the ‘Gridmesh settings’ dialog box will appear. Change the Number of cells to 20 cells in X and 15 cells in Z. Click on 'OK'; to close the dialog box. The mesh will now be shown as blue and orange lines. The orange lines are ‘regions’, defined by the bounding boxes of the objects. Click on ‘Mesh toggle’ again to turn off the mesh display.

**Set the remaining solution-control parameters**:

Click on 'Main Menu' and then on 'Numerics'.

Set the number of iterations to 200.

Click on 'Output'.

Change the monitoring point by setting

IXMON to 8,

IYMON to 1,

IZMON to 8.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

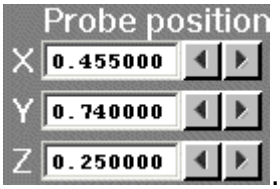
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' 

To change the direction of the plotting plane, set the slice direction to X, Y or Z 


To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:



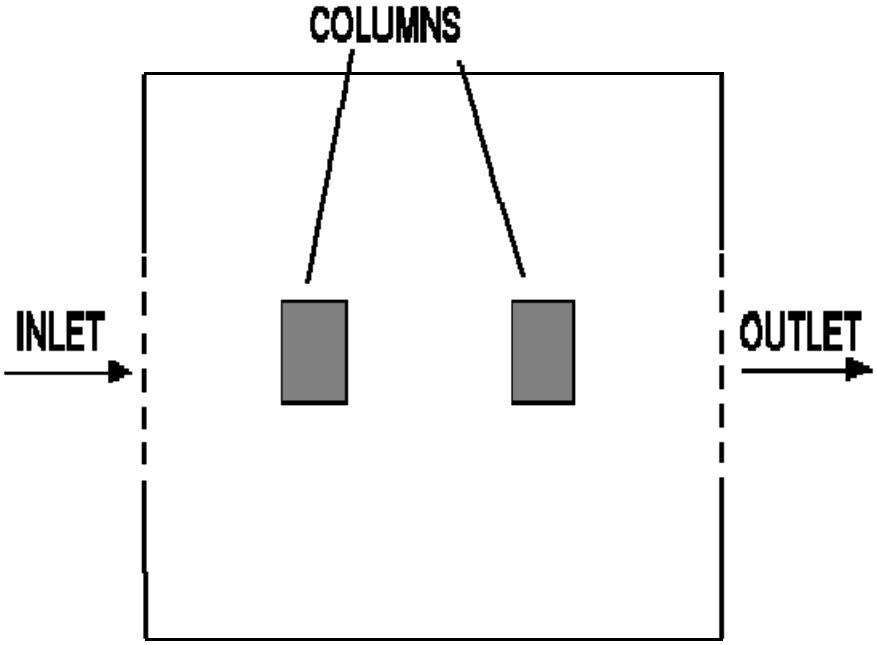

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'LABYRNTH ' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Duplication in PHOENICS-VR.

This workshop example demonstrates how to use the Object Duplication facility of the VR Editor in setting up a geometry. The simulation is run and the results are viewed.
The flow in the example is two-dimensional and its geometry is shown in the figure below.




## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core’, then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*

This is the default mode of operation.

## Within VR Editor.

**Set the domain size, and activate solution of the required variables**:

Click on 'Main Menu' and then on 'Geometry'.

Change the X-Domain Size to 2.00 m.

Click on 'OK' to close th Grid mesh dialog.

Click on 'Models'.

Set the 'solution for velocities and pressure' ON.

Select the LVEL Turbulence model in 'Turbulence Models'.

Click on 'Top menu', and on 'OK' to exit the Main Menu.

**Create lower surface of the duct**:

Click on 'New Object'.

Change name to L-WALL (which stands for low wall).

Set position and size of object as:

Xpos: 0.0 Xsize: 2.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 0.0

Define Type as Plate.

Click on 'OK' to close the Object Dialogue Box.

**Create the upper surface of the duct** by duplicating L-WALL:

Click on the ‘Duplicate object or group’ button on the handset. The duplicated object always appears at the origin. It can then be moved to the required position.

Click on the ‘Z position up’ button until the duplicate object (auto-named B2) has moved to the top of the duct (at Z=1.0)

Click on the image of the copy. This opens the Object Dialogue Box of the object copy. Change name to H-WALL (which stands for high-wall).

Click on 'OK' to exit from the Object Dialogue Box.

**Create the lower-left boundary wall**:

Click on 'New Object'.

Change name to W-WALL (which stands for west wall).

Set position and size of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 0.4

Define Type as Plate.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper-left boundary wall** by duplicating W-WALL:

Click on the ‘Duplicate object or group’ button.

Click on the ‘Z position up’ button until the duplicate object has moved as far in the Z direction as possible (Z position 0.6m)

Click on the image of the copy. Change name to W-WALL2 (which stands for west-wall2)

Click on 'OK' to exit from the Object Dialogue Box.

**Create the lower-right boundary wall** by duplicating W-WALL2:

Click on the ‘Duplicate object or group’ button.

Click on the ‘X position up’ button until the duplicate object has moved as far in the X direction as possible (X position 2.0m)

Click on the image of the copy. Change name to E-WALL (which stands for east wall).

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper-right boundary wall** by duplicating E-WALL:

Click on the ‘Duplicate object or group’ button.

Click on the ‘X position up’ button until the duplicate object has moved as far in the X direction as possible (X position 2.0m)

Click on the ‘Z position up’ button until the duplicate object has moved as far in the Z direction as possible (Z position 0.6m)

Click on the image of the copy. Change name to E-WALL2 (which stands for east-wall2).

Click on 'OK' to exit from the Object Dialogue Box.

**Create the first column**:

Click on 'New Object'.

Change name to BLOCK1 (which stands for first blockage).

Set position and size of object as:

Xpos: 0.5 Xsize: 0.2

Ypos: 0.0 Ysize: 1.0

Zpos: 0.4 Zsize: 0.2

Define Type as Blockage (default).

Click on 'OK' to exit from the Object Dialogue Box.

**Create the second column by duplicating BLOCK1**:

Click on the ‘Duplicate object or group’ button.

Click on the object copy which is located in position (0,0,0).

Change name to BLOCK2 (which stands for second block).

Change position to:

Xpos: 1.3

Ypos: 0.0

Zpos: 0.4

Click on 'OK' to exit from the Object Dialogue Box.

**Create the fluid inlet**:

Click on 'New Object'.

Change name to INLET.

Set position and size of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.4 Zsize: 0.2

Define Type as Inlet.

Click on 'Attributes'.

Set the velocity to 2.00 m/s in the X-direction.

Click on 'OK' twice.

**Create the fluid outlet**:

Click on 'New Object'.

Change name to OUTLET.

Set position and size of object as:

Xpos: 2.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.4 Zsize: 0.2

Define Type as Outlet.

Click on 'OK'.

**Set the grid**:

Click on the 'Mesh toggle' button. The default mesh (shown as orange lines) will appear on the screen. Click anywhere on the image, and the ‘Gridmesh settings’ dialog box will appear. Change the Number of cells to 40 cells in X and 20 cells in Z. Click on 'OK'; to close the dialog box. The mesh will now be

shown as blue and orange lines. The orange lines are ‘regions’, defined by the bounding boxes of the objects. Click on ‘Mesh toggle’ again to turn off the mesh display.

**Set the remaining solution control parameters**:

Click on 'Main Menu', and then on 'Numerics'.

Set the number of iterations to 300.

Click on 'Relaxation Control'.

Set DTFALS (false time step relaxation) for U1 and W1 to 0.1.

Click on 'Previous Panel' to return to 'Numerics'.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

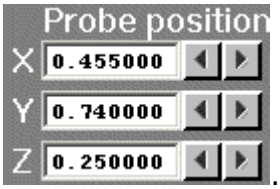
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' 

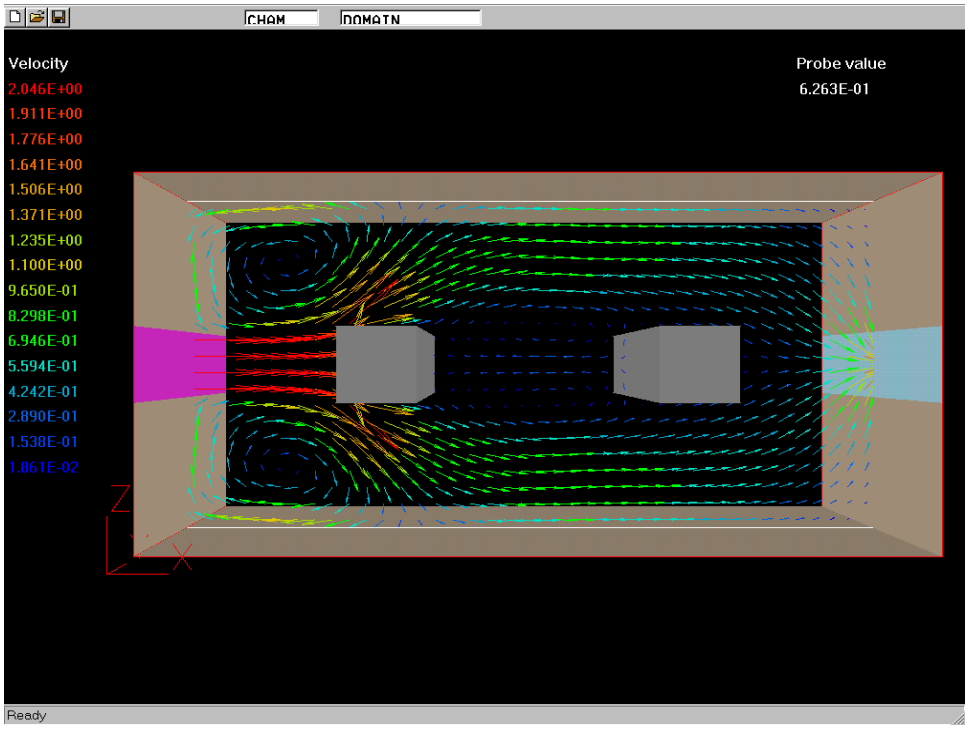
To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons



.

A typical plot from this case is:




## Saving the results.

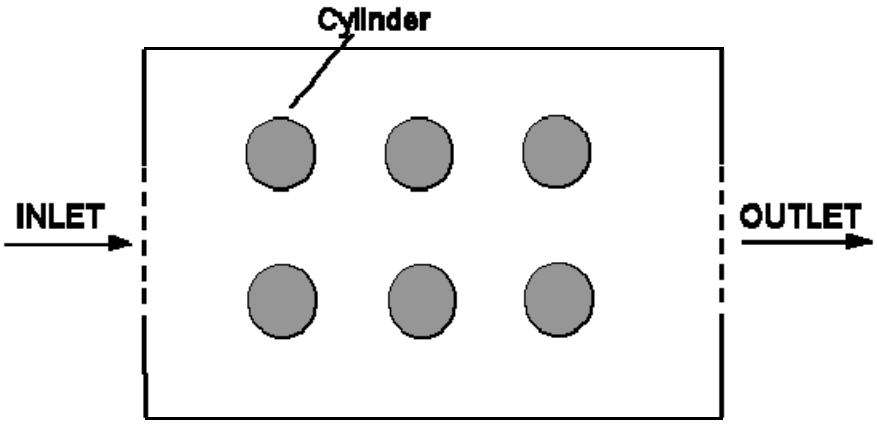
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'DUPLICAT' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Simulation of a Flow between cylindrical obstacles.

## Object Arraying

This workshop example shows how to use the Object Arraying facility of the VR Editor. The flow is two-dimensional and its geometry is shown in the figure below. The shaded circles represent cylinders. Velocity and pressure will be solved.




## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

**Set the domain size and activate solution of required variables**:

Click on 'Main Menu' and then on 'Geometry'.

Change the X-domain Size to 2.0 m.

Click 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Set the 'solution for velocities and pressure' ON.

Click on 'Top menu', then on 'OK'.

**Create the fluid inlet**:

Click on 'New Object'.

Change name to INLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.25 Zsize: 0.5

Define Type: Inlet

Click on 'Attributes' and set velocity in X-direction at 0.1 m/s.

Click on 'OK' to close the Attribute menu, and on 'OK' in the Object Dialogue Box.

**Create the fluid outlet**:

Click on 'New Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 2.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.25 Zsize: 0.5

Define Type: Outlet.

Leave the default values in the 'Attributes' dialogue box.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the first cylinder**:

Click on 'New Object'.

Change name to CYL-1 (which stands for cylinder 1).

Set POSITION and SIZE of object as:

Xpos: 0.4 Xsize: 0.15

Ypos: 0.0 Ysize: 1.0

Zpos: 0.25 Zsize: 0.15

In 'Geometry', select SHAPES\CYLINDER.DAT.

Click on ROTATION and change the value in the box beside 'Clip-art' to 5/24 instead of 1/24 (click on the > button to increase the value).

Define Type: Blockage (default).

Leave the default values in the Attributes menu.

Click on 'OK' to close the Object Dialogue Box.

**Create the remaining cylinders** by copying from CYL-1. We want 3 cylinders in the X direction, with a 0.5m space between them, and 2 in Z, spaced at 0.35m.

Check that the original cylinder is still selected and appears with white contours, if not click on it, then on OK.

Now click on the ’Duplicate using array’ button on the handset. It has this icon: 

In the new dialogue box, set

| Direction | Dimension | Pitch |
|---|---|---|
| X | 3 | 0.5 |
| Y | 1 | 0.0 |
| Z | 2 | 0.35 |

Click on 'OK' to exit the dialogue box.

An array of cylinders appears in the domain.

**Set the grid**:

Click on the 'Mesh toggle' button. The default mesh (shown as orange lines) will appear on the screen. Click anywhere on the image, and the ‘Gridmesh settings’ dialog box will appear. Change the Number of cells to 20 cells in X and 15 cells in Z. Click on 'OK'; to close the dialog box. The mesh will now be shown as blue and orange lines. The orange lines are ‘regions’, defined by the bounding

boxes of the objects. Click on ‘Mesh toggle’ again to turn off the mesh display.

**Set the remaining solution control parameters:**

Click on 'Main Menu' and then on 'Numerics'.

Set the number of iterations to 100.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

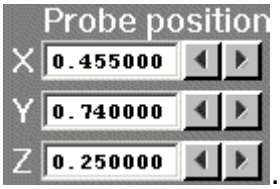
To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .

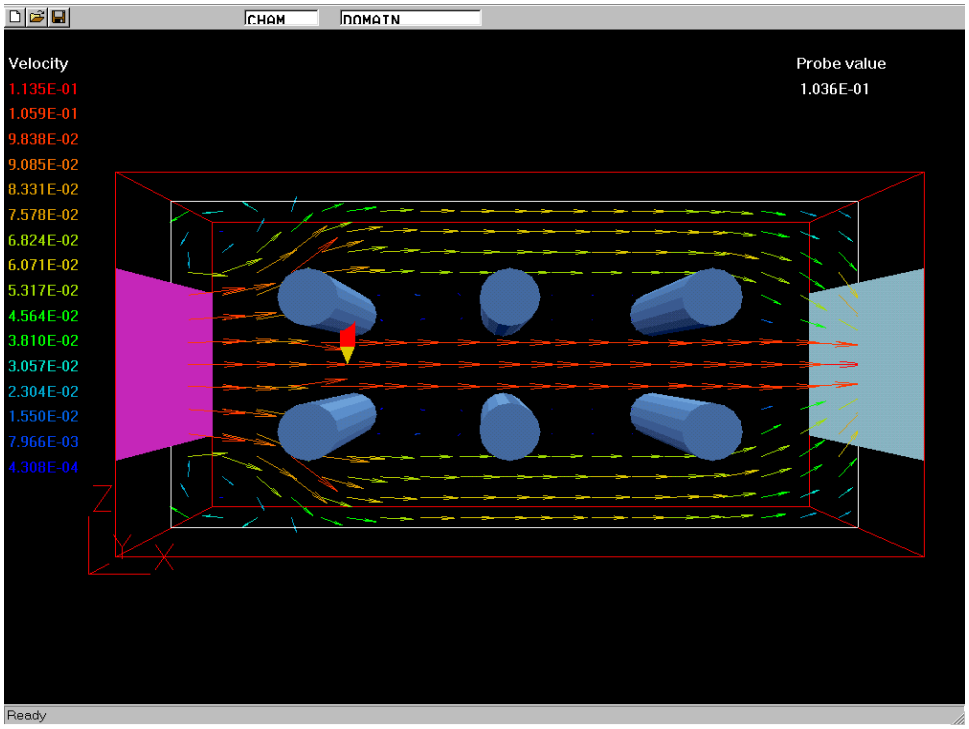
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' 

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:




## Saving the results.

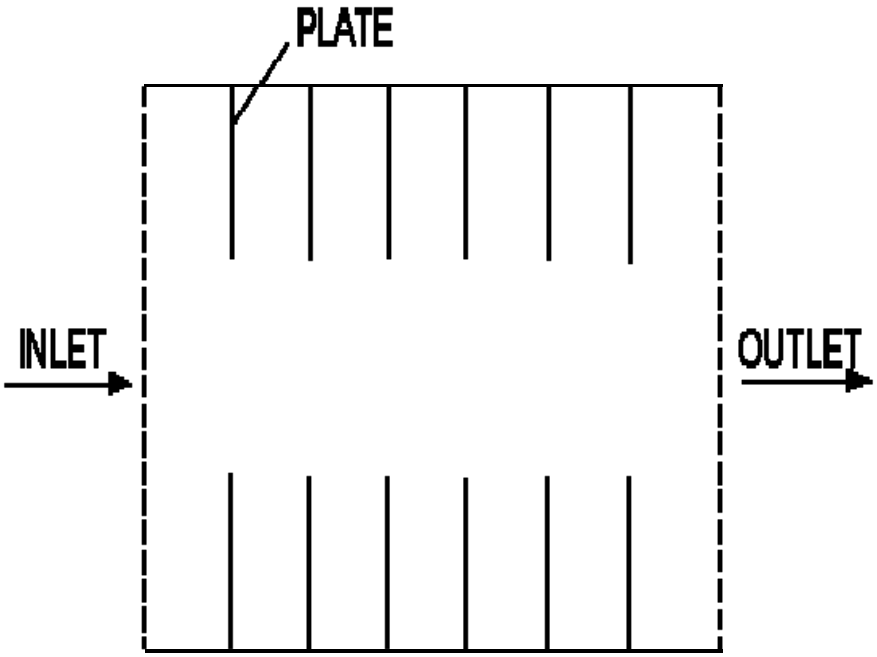
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'COLUMNS' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Array and Group Settings in PHOENICS-VR.

## Array and Group Settings in PHOENICS-VR.

This workshop demonstrates how to set up a geometry with the Array and Group facility of the VR Editor, and to run the simulation. The flow is two-dimensional and its geometry is shown in the figure below.
Velocity and pressure will be solved.




## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*

Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

**Set the domain size and activate solution of required variables**:

Click on 'Main Menu' and then on 'Geometry'.

Change the X-Domain Size to 2.00 m.

Click 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Set the 'solution for velocities and pressure' ON.

Select the LVEL Turbulence model in 'Turbulence Models'.

Click on 'Top menu', on 'OK'

**Create the upper boundary wall**:

Click on 'New Object'.

Change name to H-WALL (which stands for high-wall).

Set position and size of object as:

Xpos: 0.0 Xsize: 2.0

Ypos: 0.0 Ysize: 1.0

Zpos: 1.0 Zsize: 0.0

Define Type as Plate.

Click on 'OK' to exit from the Object Dialogue Box.

**Create lower boundary wall** by duplicating H-WALL:

Click on the ‘Duplicate object or group’ button.

Click on the image of the object copy (located at (0,0,0)).

Change name to L-WALL (which stands for low-wall).

Click on 'OK' to exit from the Object Dialogue Box.

**Create the fluid inlet**:

Click on 'New Object'.

Change name to INLET.

Set position and size of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 1.0

Define Type as Inlet.

Click on 'Attributes'.

Set the velocity to 2.00 m/s in the X-Direction.

Click on 'OK' twice to exit from the Object Dialogue Box.

**Create the fluid outlet** by duplicating INLET:

Click on the ‘Duplicate object or group button.

Click on the ‘X position up’ button until the copy has moved as far as possible in the X direction (X position 2.0m)

Click on the image of the object copy.

Change Type to Outlet.

Change name to OUTLET.

Click on 'OK' to exit the Object Dialogue Box.

**Create the first plate blockage object**:

Click on 'New Object'.

Define Type as Blockage (default).

Change name to BLOCK1 (which stands for first blockage).

Set position and size of object as:

Xpos: 0.36 Xsize: 0.05

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 0.25

Click on 'OK' to exit from the Object Dialogue Box.

**Create 5 copies of BLOCK1 along the lower surface** using the arraying feature:

Click on the ‘Duplicate using array’ button on the handset.

Set Dimension and Pitch as:

| Direction | Dimension | Pitch |
|---|---|---|
| X: | 6 | 0.24 |
| Y: | 1 | 0.0 |
| Z: | 1 | 0.0 |

Click on 'OK'.

**We now turn the 6 blocks (original + 5 copies) into a group, and copy the group to make the upper six blocks**.

Click on the ‘Group objects’ button on the handset

Click on 'Switch on group mode' then click 'OK'.

BLOCK1 has now been added to the group.

For BLOCK2 (B.6), click twice on the object, and 'Add it to group'.

Do the same for BLOCK3 (B.7), BLOCK4 (B.8),...BLOCK6 (B.10).

Click on the ‘Group objects’ button to check that all 6 blocks have been added to the group.

Click on 'OK' to exit.

Click on the ‘Duplicate object or group’ button.

Click on the ‘Group objects’ button.

Set position as:

Xpos: 0.36

Ypos: 0.0

Zpos: 0.75

Click on 'OK'.

Click on the ‘Group objects’ button.

Click on 'Switch off group mode', and click on 'OK'.

**Set the grid**:

Click on the 'Mesh toggle' button. The default mesh (shown as orange lines) will appear on the screen. Click anywhere on the image, and the ‘Gridmesh settings’ dialog box will appear. Change the Number of cells to 40 cells in X and 20 cells in Z. Click on 'OK'; to close the dialog box. The mesh will now be shown as blue and orange lines. The orange lines are ‘regions’, defined by the bounding boxes of the objects. Click on ‘Mesh toggle’ again to turn off the mesh display.

**Set the remaining solution control parameters**:

Click on 'Main Menu' and then on 'Numerics'.

Set the number of iteration to 300.

Click on 'Output'.

Set the monitor location to XMON=1.0 and ZMON=0.5.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

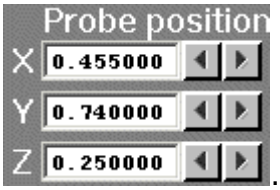
To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 

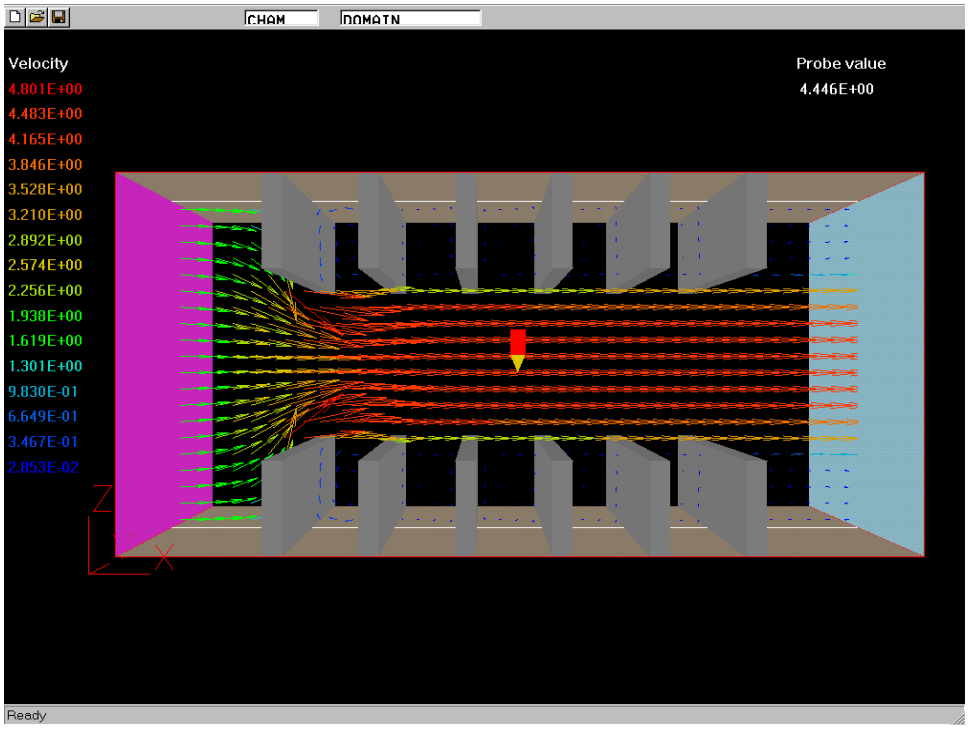
- To select any other variable - click on the 'Select a Variable button' 


To change the direction of the plotting plane, set the slice direction to X, Y or Z 


To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:




## Saving the results.

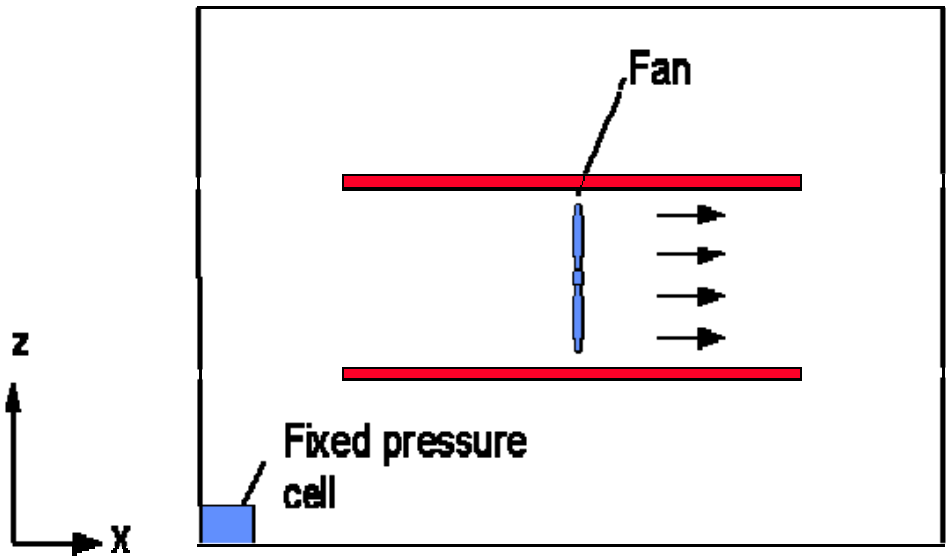
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'ARRAYS' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Fans and Inlets internal to the Domain.

## Fans and Inlets within the flow domain.

In this workshop it is shown how to specify an internal momentum source in PHOENICS-VR as a fan object or, alternatively, as an internal mass source-sink, represented as an inlet-outlet pair.
With the fan-type object, the user can fix the velocity at that location. This just circulates the air already in the domain. The fan-method is demonstrated in the first section.
A similar flow can be set up by replacing the fan object by an inlet-outlet pair, separated by a small blocked region. This continually injects fresh air, which then leaves the domain. The inlet-outlet pair method is demonstrated in the second section.
Note that this demonstration is interesting because the fan is internal; if the fan object were at a boundary, then the fan object would work like an ordinary inlet.)



## Within PHOENICS-VR environment:

Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor:

**Set the domain size, and activate solution of the required variables**:

Click on ‘Main Menu’ and then on ‘Geometry’.

Change the z-domain size to 0.6.

Click 'OK' to close the Grid mesh settings dialog.

Click on ‘Models’.

Turn ‘Solution for velocities and pressure’ to ON. Click on LAMINAR and choose the KEMODL.

Click on ‘Top Menu’ and then on 'OK';.

**Create the lower internal wall**:

Click on ‘New Object’ button.

Change Name to ‘WALL-1’.

Change the following to:

XPos: 0.2 XSize: 0.6

YPos: 0.0 YSize: 1.0

ZPos: 0.2 ZSize: 0.02

Click on 'OK';.

**Create the upper internal wall** by duplicating WALL-1:

Click on the ‘Duplicate object or group’ button and then on the resulting object at the origin.

Change Name to ‘WALL-2’.

Change the following to:

XPos: 0.2

YPos: 0.0

ZPos: 0.4

Click on 'OK';.

**Create the fan object**:

Click on ‘New Object’.

Change Name to ,’Fan’.

Change the following to:

XPos: 0.5 XSize: 0.0

YPos: 0.0 YSize: 1.0

ZPos: 0.22 ZSize: 0.18

Click on Type dialogue box and change Type to FAN. Click on OK.

Click on Attributes and change the x-direction velocity to 1.0 m/s.

Click on OK.

Click on 'OK';.

**Create a fixed pressure point** – the domain is closed, and the fluid is incompressible. Without a fixed pressure point the pressure level is undefined.

Click on ‘New Object’.

Change Name to ‘Fixedp’.

Change the following to:

XSize: 0.03

YSize: 1.0

ZSize: 0.03

Click on ‘Type’ dialogue box and choose Pressure_relief. Click on 'OK';.

Click on 'OK';.

**Set the grid**:

Click on the 'Mesh toggle' button. The default mesh (shown as orange lines) will appear on the screen. Click anywhere on the image, and the ‘Gridmesh settings’ dialog box will appear. Change the Number of cells to 20 cells in X and 15 cells in Z.

Click on ‘Modify region’ for Z, and enter 1. Click 'Apply'.

Click on ‘Cells in region’ for Z and enter 6.

Click on ‘Modify region’ for Z, and enter 2. Click 'Apply'.

Click on ‘Cells in region’ for Z and enter 3.

Click on ‘Modify region’ for Z, and enter 3.
Click 'Apply'.

Click on ‘Cells in region’ for Z and enter 15.

Click on ‘Power/ratio’ for Z, and enter 2.

Click on Symmetric for Z and toggle to Yes

Click on ‘Modify region’ for Z, and enter 4.
Click 'Apply'.

Click on ‘Cells in region’ for Z and enter 3.

Click on ‘Modify region’ for Z, and enter 5.
Click 'Apply'.

Click on ‘Cells in region’ for Z and enter 6.

Click on 'OK'; to close the dialog box.

The mesh will now be shown as blue and orange lines. The orange lines are ‘regions’, defined by the bounding boxes of the objects. Click on ‘Mesh toggle’ again to turn off the mesh display.

**Set the remaining solution control parameters**:

Click on ‘Main Menu’, then on click on Numerics and enter 200 in the ‘Total number of iterations’ dialogue box.

Click on ‘Top Menu’ and 'OK';.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

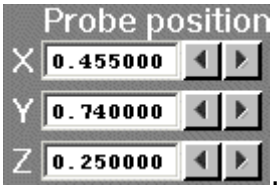
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

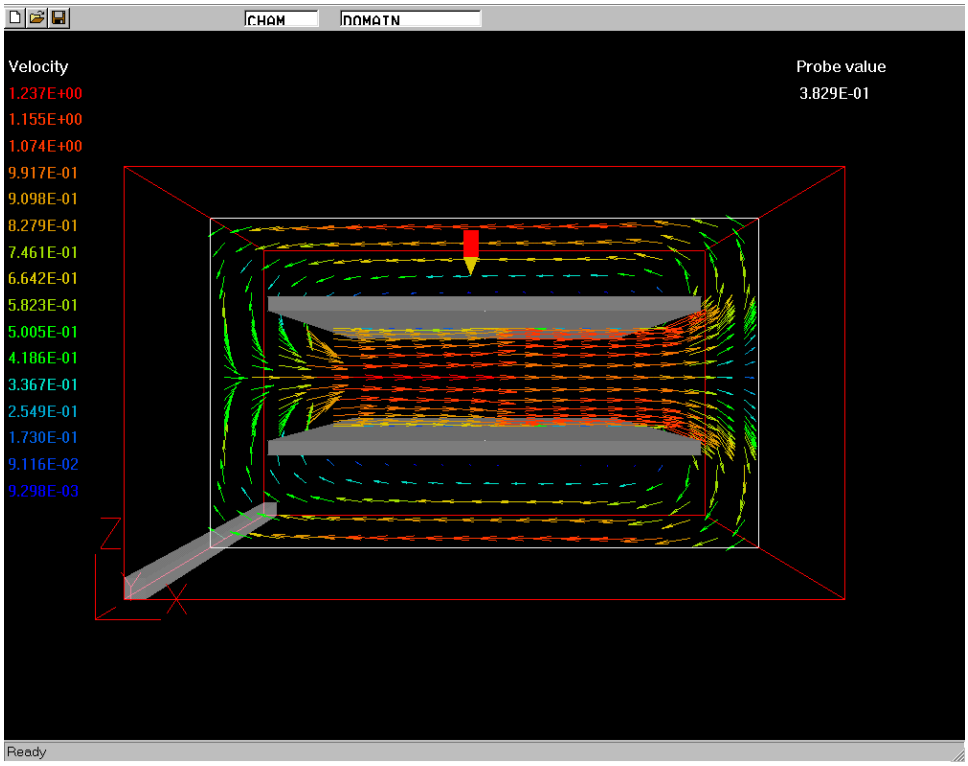
- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' 

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons



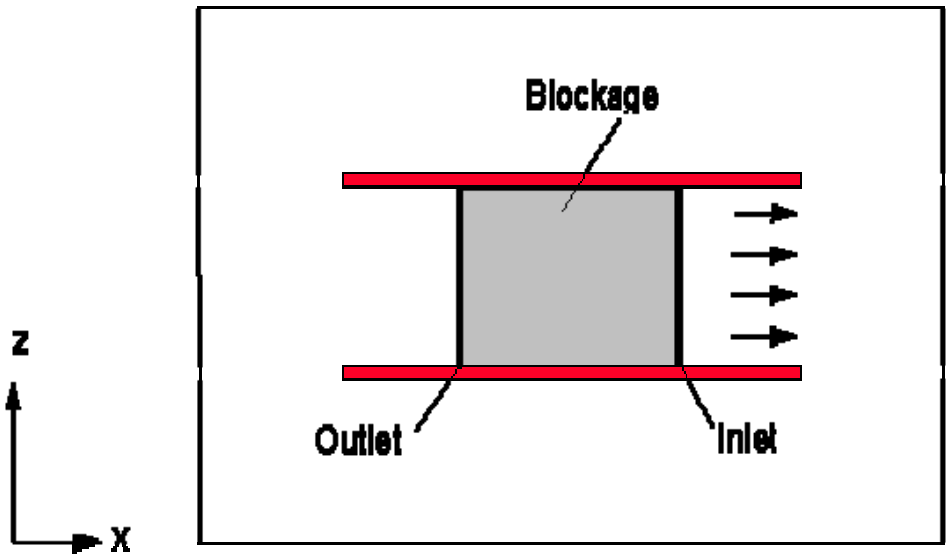
A typical plot from this case is:




## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'INLETS' (e.g.) and save as 'CASE1' (e.g.).

## Alternative method: Using Inlet-Outlet pair within the Domain.




**Change the fan (fixed velocity) into an inlet (fixed mass flow)**

Click on the fixed cell object and then on Delete.

Click on OK.

Click on the FAN object.

Click on the Type dialogue box.

Click on Inlet/OK.

Click on Attributes and enter 1.0 in the x-direction dialogue box.

Change Object side to High.

Click on 'OK';.

Change the position and size to:

XPos: 0.7

YPos: 0.0

ZPos: 0.22

Click on 'OK';

**Create the blockage separating the inlet and outlet**:

Click on ‘New Object’

Change the following to:

XPos: 0.3 XSize: 0.4

YPos: 0.0 YSize: 1.0

ZPos: 0.22 YSize: 0.18

Click on 'OK';

**Create the outlet**:

Click on ‘New Object’.

Change the Name to ‘OUT’.

Change the following to:

XPos: 0.3 XSize: 0.0

YPos: 0.0 YSize: 1.0

ZPos: 0.22 ZSize: 0.18

Click on ‘Type’ and select OUTLET.

Click on ‘Attributes’ and check that the Object side is set to ‘Low’.

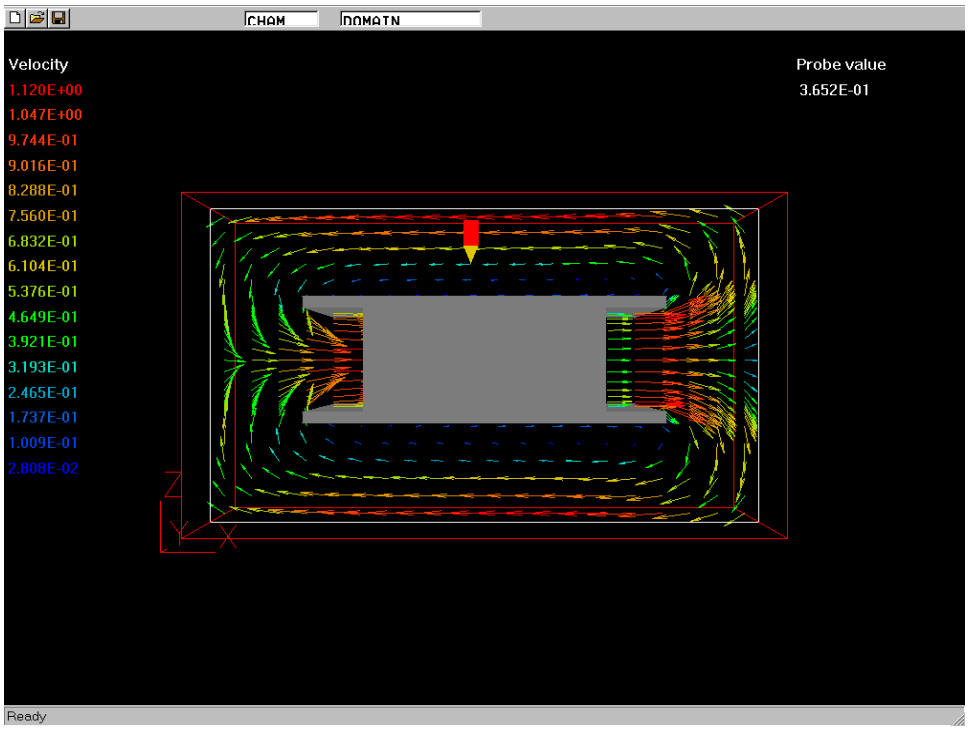
Click on OK/OK.

-

## Run EARTH.

## Use VR Viewer to view the results.

The object FAN should be acting as a ‘source’ and the object OUT should be acting as a ‘sink’.




Investigate the following combination of Attributes for the FAN and OUT objects

| Object name | Fan | Out |
|---|---|---|
| | | |
| Object type | Inlet | Outlet |
| Attributes | Velocity = -1.0 m/s | pressure condition |
| Object side | High | Low |
| | | |
| Object type | Outlet | Inlet |
| Attributes | pressure condition | Velocity = -1.0 m/s |
| Object side | High | Low |
| | | |
| Object type | Outlet | Inlet |
| Attributes | pressure condition | Velocity = +1.0 m/s |
| Object side | High | Low |

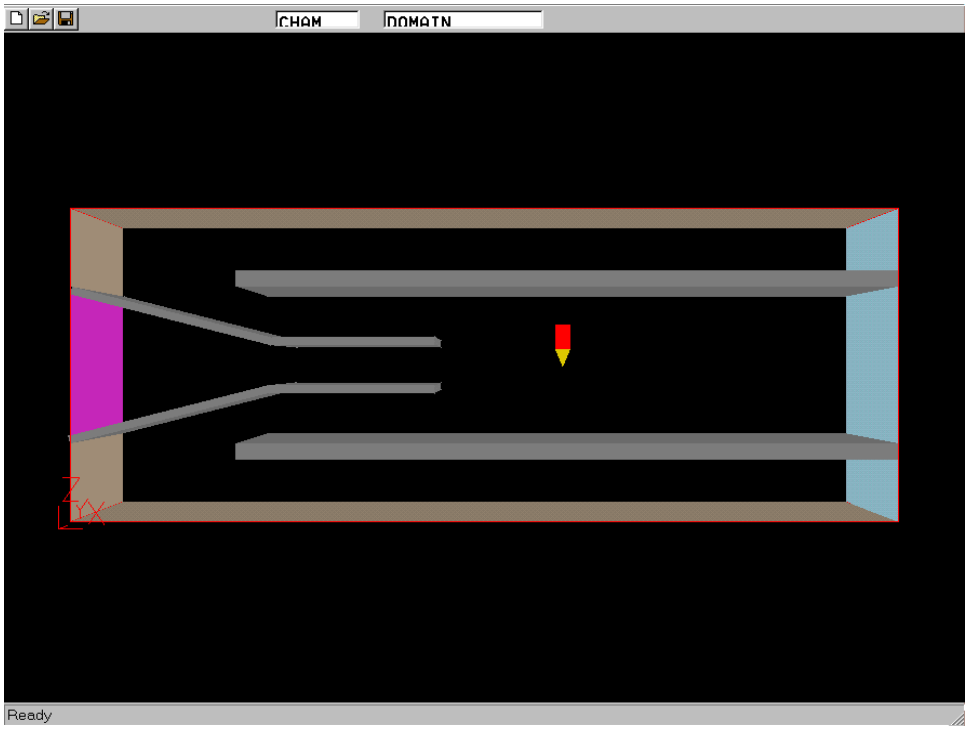
Each run can be saved as a different case in the INLETS folder.

Home Page

# WORKSHOP - Jet Pump

## Object Rotation

This example shows how to set up and run the simulation of a Jet Pump using VR. It demonstrates the use of the duplication facility and how to rotate objects. The flow is two-dimensional and its geometry is shown below. Velocity and pressure will be solved.




## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.

*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

**Set the domain size, and activate solution of variables**:

Click on 'Main Menu' and then on 'Geometry'.

Change the X-Domain Size to 5.0 m.

Change the Z-Domain Size to 2.0 m.

Set the number of cells:

- 60 cells in the X-direction,
- 1 cell in the Y-direction,
- 60 cells in the Z-direction.

Click 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Click on the 'solution for velocities and pressure' to set it to ON.

Select the KECHEN turbulence model in 'Turbulence models'.

Click on 'Top menu' and then on 'OK'.

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

**Create the fluid inlet:**

Click on 'New Object'.

Change name to INLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.55 Zsize: 0.9

Define Type: Inlet.

Click on 'Attributes' and set velocity in X-direction at 20.0 m/s.

Click on 'OK' twice to exit from the Object Dialogue Box.

**Create the fluid outlet:**

Click on 'New Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 5.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.5 Zsize: 1.0

Define Type: Outlet.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper secondary inlet** (as an opening):

Click on 'New Object'.

Change name to OPEN1 (which stands for first opening).

Set POSITION and SIZE of object as:

Xpos: 5.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 1.6 Zsize: 0.4

Define Type: Outlet.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the lower secondary inlet**, by duplication from OPEN1:

Click on the 'Duplicate object or group' button on the handset.

Click on the image of the object copy, located in (0,0,0).

Change name to OPEN2 (which stands for second opening).

Set position as:

Xpos: 5.0

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper-left boundary wall:**

Click on 'New Object'.

Change name to WALL1.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 1.45 Zsize: 0.55

Define Type: Plate.

Click on 'OK' twice to exit from the Object Dialogue Box.

**Create the lower-left boundary wall**, by duplication from WALL1:

Click on the 'Duplicate object or group' button.

Click on the image of the object copy, located in (0,0,0).

Change name to WALL2.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper inner horizontal wall:**

Click on 'New Object'.

Change name to WALL3.

Set POSITION and SIZE of object as:

Xpos: 1.0 Xsize: 4.0

Ypos: 0.0 Ysize: 1.0

Zpos: 1.5 Zsize: 0.1

Define Type: Blockage.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the lower inner horizontal wall** by duplication from WALL3**:**

Click on the 'Duplicate object or group' button.

Click on the object copy, located in (0,0,0).

Change name to WALL4.

Change position to:

Xpos: 1.0

Ypos: 0.0

Zpos: 0.4

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper angled wall:**

Click on 'New Object'.

Change name to BAFFLE1.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.25

Ypos: 0.0 Ysize: 1.0

Zpos: 1.45 Zsize: 0.05

Define Type: Blockage

Click on ROTATION, set the 'Angle about Y' angle to 15 degrees.

Click on 'OK' twice to exit from the Object Dialogue Box.

**Create the lower angled wall** by duplication from BAFFLE1:

Click on the Duplication button.

Click on the image of the object copy, located in (0,0,0).

Change name to BAFFLE2.

Change position to:

Xpos: 0.01295 (to allow for the object rotation)

Ypos: 0.0

Zpos: 0.5

Click on ROTATION, change the 'Angle about Y' rotation angle to 345 degrees.

Click on 'OK' twice to exit from the Object Dialogue Box.

**Create the upper horizontal baffle:**

Click on 'New Object'.

Change name to BAFFLE3.

Set POSITION and SIZE of object as:

Xpos: 1.2 Xsize: 1.0

Ypos: 0.0 Ysize: 1.0

Zpos: 1.13 Zsize: 0.05

Define Type: Blockage (default, its default attribute is

'Solid with smooth-wall friction').

Click on 'OK' to exit from the Object Dialogue Box.

**Create the lower horizontal baffle** by duplication from BAFFLE3:

Click on the 'Duplicate object or group' button.

Click on the image of the object copy, located in (0,0,0).

Change name to BAFFLE4.

Change position to:

Xpos: 1.2

Ypos: 0.0

Zpos: 0.82

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper boundary wall:**

Click on 'New Object'.

Change name to T-WALL (which stands for top wall).

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 5.0

Ypos: 0.0 Ysize: 1.0

Zpos: 2.0 Zsize: 0.0

Define Type: Plate.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the lower boundary wall** by duplication from T-WALL;

Click on the 'Duplicate object or group' button.

Click on the image of the object copy, located in (0,0,0).

Change name to L-WALL (which stands for low wall).

Click on 'OK' to exit from the Object Dialogue Box.

**Set the remaining solution control parameters:**

Click on 'Output'.

Change the monitoring point by setting

XMON to 3.0,

YMON to 0.5,

ZMON to 0.5

Click on 'Numerics'.

Set the number of iterations to 300 (this is not sufficient for complete convergence, but it gives a reasonable flow field).

Click on 'Relaxation Control'.

Set the Auto calculation of false-time step factor, SARAH, to 0.005.

Click on 'Previous Panel' to return to 'Numerics'.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.
To view:

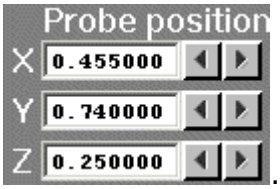
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

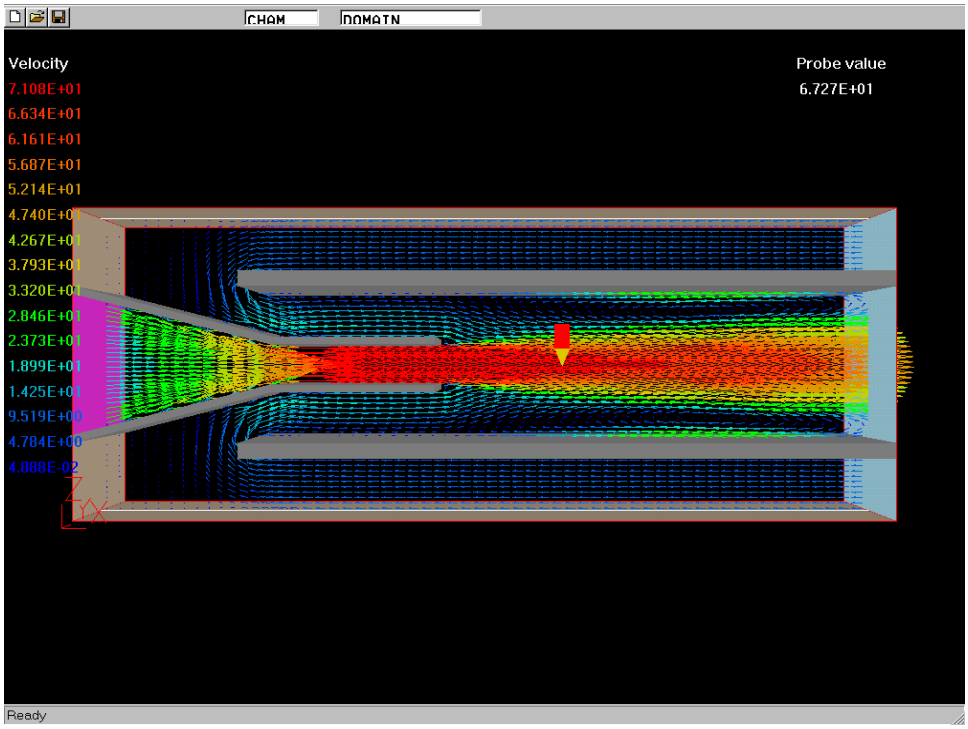
- To select Pressure - click on the 'Select Pressure button' P.
- To select Velocity - click on the 'Select Velocity button' V
- To select Temperature - click on the 'Select Temperature button' T
- To select any other variable - click on the 'Select a Variable button' C

To change the direction of the plotting plane, set the slice direction to X, Y or Z X Y Z

To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:



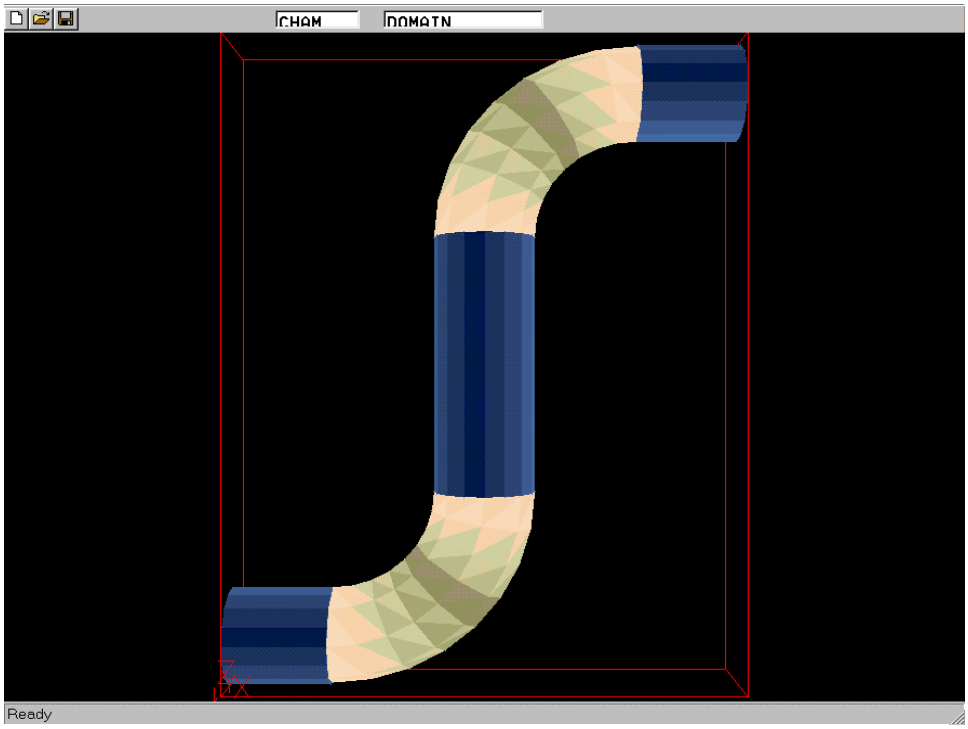

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'JETPUMP' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Flow Simulation within a pipe.

The following instructions will allow the user to set up and run the simulation of gas in a pipe using VR. Initially, the domain is filled with a solid, and the pipe is then 'carved' out by making the pipe sections from a fluid material. The flow is three-dimensional and its geometry is shown below:




## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core', then click on 'OK'; to confirm the resetting.

*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

### Set the domain size, activate solution of variables and set the domain material:

Click on 'Main Menu' and then on 'Geometry'.

Change the X-domain Size to 1.5 m.

Change the Y-domain size to 0.3 m.

Change the Z-domain size to 2.0 m.

Set the number of cells:

- 20 in the X-direction,
- 5 in the Y-direction,
- 30 in the Z-direction.

Click 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Set the 'solution for velocities and pressure' ON.

Select the LVEL Turbulence model in 'Turbulence Models'.

Click on 'Properties'.

Click on the button next to 'The current domain material is'. Select 'Gases', then 'Carbon Dioxide at 27 deg C, 1 atm'.

Click on 'Top menu' then on 'OK'.

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

**Create a solid to fill the domain:**

Click on 'New Object'.

Change name to SOLID.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.5

Ypos: 0.0 Ysize: 0.3

Zpos: 0.0 Zsize: 2.0

Click on 'OK' to close the Object Dialogue box.

Click on the SOLID object, to bring up the Object dialog again, then click on 'Hide'. The solid will no longer obscure the rest of the geometry.

**Create the first horizontal pipe segment:**

Click on 'New Object'.

Change name to PIPE-1.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.3

Ypos: 0.0 Ysize: 0.3

Zpos: 0.0 Zsize: 0.3

In 'Geometry', select SHAPES\CYLINDER.DAT.

Define Type: Blockage (default).

Click on 'Attributes', click on 'Other materials', 'Domain material'.

Click on 'OK' to exit the attributes menu, and click on 'OK' to close the Object Dialogue box.

Click on the 'Rotate object up' button (middle-right of the second row of buttons) to give it the right orientation: the axis of the cylinder is in X-direction.

**Create the second horizontal pipe section** by duplication from PIPE-1:

Click on 'Duplicate object or group'.

Click on 'X position up' until the copy object has moved as far as possible in X (Xpos=1.2)

Click on 'Z position up' until the copy object has moved as far as possible in Z (Zpos=1.7)

Click on the copy object and change name to PIPE-2.

**Create the vertical pipe section:**

Click on 'New Object'.

Change name to PIPE-3.

Set POSITION and SIZE of object as:

Xpos: 0.6 Xsize: 0.3

Ypos: 0.0 Ysize: 0.3

Zpos: 0.6 Zsize: 0.8

In 'Geometry', select SHAPES\CYLINDER.DAT.

Define Type: Blockage (default).

Click on 'Attributes', then on 'Other materials', 'Domain material'.

Click on 'OK' to exit the attributes menu, and click on 'OK' to close the Object Dialogue box.

The cylinder already has its axis in the Z-direction as desired.

**Create the lower pipe bend:**

Click on 'New Object'.

Change name to BPIPE-1.

Set POSITION and SIZE of object as:

Xpos: 0.3 Xsize: 0.6

Ypos: 0.0 Ysize: 0.3

Zpos: 0.0 Zsize: 0.6

In 'Geometry', select SHAPES\PIPEBEND.DAT.

Define Type: Blockage (default).

Click on 'Attributes', then on 'Other materials', 'Domain material'.

Click on 'OK' to exit the Attributes menu, and click on 'OK' to close the Object Dialogue box.

Rotate the pipe to give it the right direction (clip art rotation 9/24).

**Create the second pipe bend** by duplication from BPIPE-1:

Click on 'Duplicate object or group'.

Click on the copy object (at 0,0,0)

Change name to BPIPE-2.

Set POSITION of object as:

Xpos: 0.6

Ypos: 0.0

Zpos: 1.4

Rotate the pipe to give it the right direction (clip art rotation 3/24).

**Create the fluid inlet:**

Click on 'New Object'.

Change name to INLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 0.3

Zpos: 0.0 Zsize: 0.3

In 'Geometry', select SHAPES\CYLINDER.DAT.

Click on 'Rotation' and toggle clip art rotation to 9/24.

Define Type: Inlet.

Click on Attributes to define the inlet conditions.

Set the velocity of the inflow in X-direction at 1.0 m/s.

Click on 'OK' to close the Attributes menu, and on 'OK' in the Object Dialogue Box.

**Create the fluid outlet:**

Click on 'New Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 1.5 Xsize: 0.0

Ypos: 0.0 Ysize: 0.3

Zpos: 1.7 Zsize: 0.3

In 'Geometry', select SHAPES\CYLINDER.DAT.

Click on 'Rotation' and toggle clip art rotation to 9/24.

Define Type: outlet.

Leave the default values in the Attributes Dialogue box.

Click on 'OK'.

**Set the remaining solution control parameters:**

In the Main Menu, click on 'OUTPUT'.

Place the probe in the middle of the vertical pipe section by setting

XMON to 0.75,

YMON to 0.15,

ZMON to 1.5.

Click on 'Numerics'.

Set the number of iterations to 100.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.
To view:

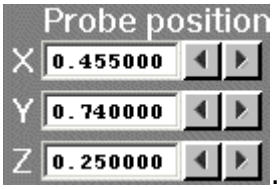
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

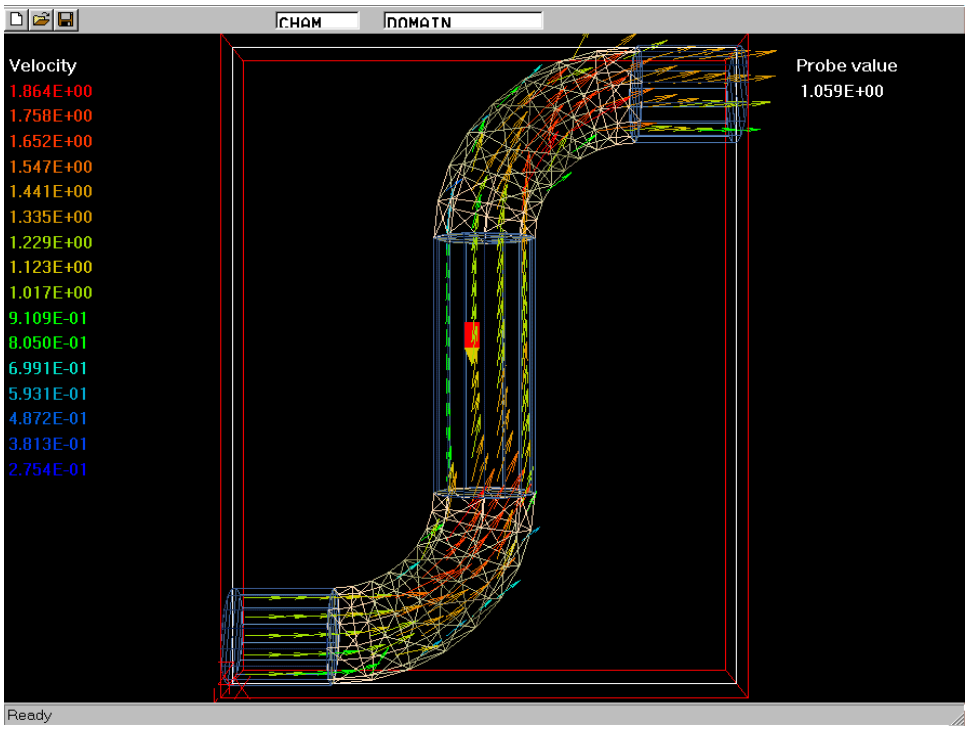
- To select Pressure - click on the 'Select Pressure button' P.
- To select Velocity - click on the 'Select Velocity button' V
- To select Temperature - click on the 'Select Temperature button' T
- To select any other variable - click on the 'Select a Variable button' C

To change the direction of the plotting plane, set the slice direction to X, Y or Z X Y Z

To change the position of the plotting plane, move the probe using the probe position buttons




To make the objects see-through, click the 'Wire-frame' toggle (to the right of the Macro button on the second row of buttons).
A typical plot from this case is:



## Saving the results.

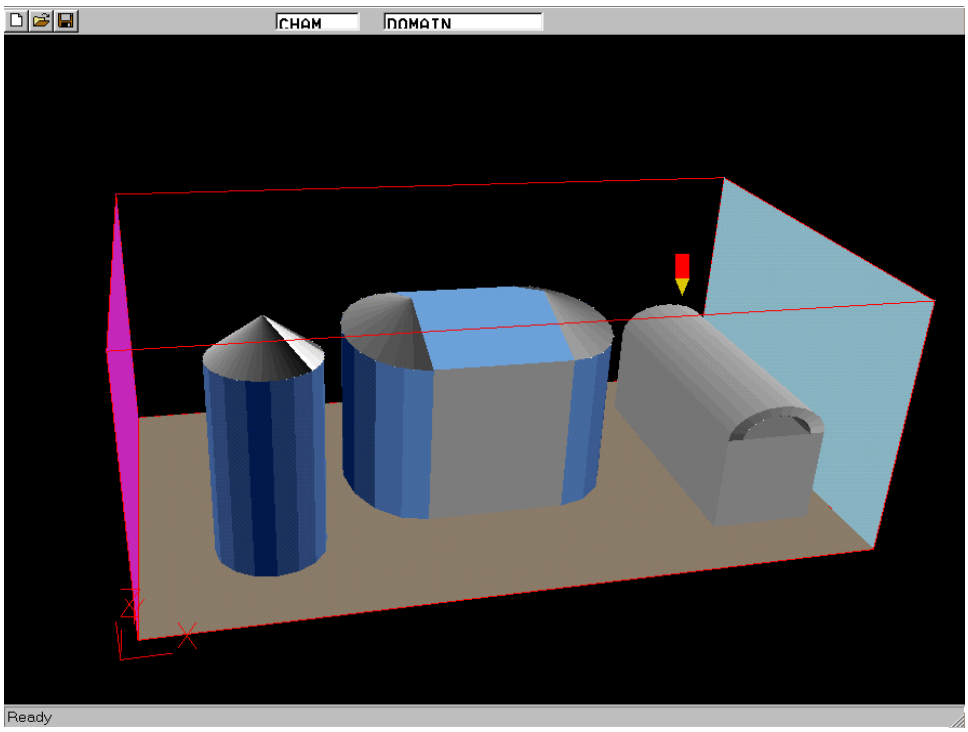
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'PIPE' (e.g.) and save as 'CASE1' (e.g.)

Home Page

# WORKSHOP - Flow Around Buildings.

## External Flow

The following instructions will allow the user to set up and run the simulation of a flow around three buildings created with some current ASAP objects in PHOENICS VR Editor. The flow is three-dimensional and its geometry is shown below.




The calculations are performed using the PHOENICS solver, EARTH, and the results are viewed using VR Viewer. Velocity and pressure will be simulated.

## Accessing PHOENICS-VR.

*From the system level:*

To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

### Set the domain size, and activate solution of variables:

Click on 'Main Menu' and then on 'Geometry'.

Change the X-Domain Size to 100 m.

Change the Y-Domain Size to 60 m.

Change the Z-Domain Size to 40 m.

Click on 'Main Menu' and then on 'Geometry'.

Set the Number of cells:

- 20 cells in the X-direction,
- 20 cells in the Y-direction,
- 10 cells in the Z-direction.

Click OK to close the Grid mesh settings dialog.

Click on 'Models'.

Set the 'Solution for velocities and pressure' to ON.

Click on 'Top menu', then on 'OK'.

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

**Create the fluid inlet:**

Click on 'New Object'.

Change name to INLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 60.0

Zpos: 0.0 Zsize: 40.0

Define Type: Inlet.

Click on 'Attributes' and set velocity in X-direction at 25 m/s.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the fluid outlet:**

Click on 'New Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 100.0 Xsize: 0.0

Ypos: 0.0 Ysize: 60.0

Zpos: 0.0 Zsize: 40.0

Define Type: Outlet.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the ground plane:**

Click on 'New Object'.

Change name to GROUND

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 100.0

Ypos: 0.0 Ysize: 60.0

Zpos: 0.0 Zsize: 0.0

Define Type: Plate.

Click on 'OK' to exit from the Object Dialogue Box.

**Create body of cylindrical building:**

Click on 'New Object'.

Change name to BUILD_1.

Set POSITION and SIZE of object as:

Xpos: 10.0 Xsize: 15.0

Ypos: 10.0 Ysize: 15.0

Zpos: 0.0 Zsize: 30.0

In 'Geometry', select SHAPES\CYLINDER.DAT.

Define Type: Blockage (default, with default attribute 'solid with smooth-wall friction).

Click on 'OK' to exit from the Object Dialogue Box.

**Create conical roof of cylindrical building:**

Click on 'New Object'.

Change name to ROOF_1.

Set POSITION and SIZE of object as:

Xpos: 10.0 Xsize: 15.0

Ypos: 10.0 Ysize: 15.0

Zpos: 30.0 Zsize: 5.0

In 'Geometry', select SHAPES\CONE.DAT.

Define Type: set same type and property as the previous object (which is the default).

Click on 'OK' to exit from the Object Dialogue Box.

**Create core of body of central building:**

Click on 'New Object'.

Change name to BUILD_21.

Set POSITION and SIZE of object as:

Xpos: 40 Xsize: 20

Ypos: 20 Ysize: 30

Zpos: 0 Zsize: 25

Click on 'OK' to exit from the Object Dialogue Box.

**Create half-cylindrical left end of central building:**

Click on 'New Object'.

Change name to BUILD_22.

Set POSITION and SIZE of object as:

Xpos: 30 Xsize: 10

Ypos: 20 Ysize: 30

Zpos: 0 Zsize: 25

In 'Geometry', select SHAPES\HFCYL.DAT (half cylinder).

Click on 'OK' to exit from the Object Dialogue Box.

Use the 'Rotate object up/down' buttons  to turn the shape in the right position. (clip art rotation 2/24).

**Create half-cylindrical right end of central building:**

Click on the 'Duplicate object or group' button.

Click on the image of the object copy, located in (0,0,0).

Change name to BUILD_23.

Change position to:

Xpos: 60

Ypos: 20

Zpos: 0

Click on 'OK' to exit from the Object Dialogue Box.

Adjust the orientation with the rotation arrow buttons (clip art rotation 4/24).

**Create half-cone left end of roof of central building:**

Click on 'New Object'.

Change name to ROOF_21.

Set POSITION and SIZE of object as:

Xpos: 30 Xsize: 10

Ypos: 20 Ysize: 30

Zpos: 25 Zsize: 5

In 'Geometry', select SHAPES\HFCONE.DAT (half cone).

Click on 'OK' to exit from the Object Dialogue Box.

Adjust the orientation with the rotation arrow buttons (clip art rotation 2/24).

**Create half-cone right end of roof of central building:**

Click on the 'Duplicate object or group' button.

Click on the image of the object copy, located in (0,0,0).

Change name to ROOF_22.

Change position to:

Xpos: 60

Ypos: 20

Zpos: 25

Click on 'OK' to exit from the Object Dialogue Box.

Adjust the orientation with the rotation arrow buttons (clip art rotation 4/24).

**Create front half of central wedge roof of central building:**

Click on 'New Object'.

Change name to ROOF_23

Set POSITION and SIZE of object as:

Xpos: 40 Xsize: 20

Ypos: 20 Ysize: 15

Zpos: 25 Zsize: 5

In 'Geometry', select SHAPES\WEDGE1.DAT.

Click on 'OK' to exit from the Object Dialogue Box.

Adjust the orientation with the rotation arrow buttons (clip art rotation 3/24).

**Create back half of central wedge roof of central building:**

Click on the DUPLICATION button.

Click on the object copy positioned in (0,0,0).

Change name to ROOF_24.

Change position to:

Xpos: 40

Ypos: 35

Zpos: 25

Click on 'OK' to exit from the Object Dialogue Box.

Adjust the orientation with the rotation arrow buttons (clip art rotation 1/24).

**Create body of right-hand building:**

Click on 'New Object'.

Change name to BUILD_3.

Set POSITION and SIZE of object as:

Xpos: 80 Xsize: 15

Ypos: 10 Ysize: 40

Zpos: 0 Zsize: 15

Click on 'OK' to exit from the Object Dialogue Box.

**Create arched roof of right-hand building:**

Click on 'New Object'.

Change name to ROOF_3.

Set POSITION and SIZE of object as:

Xpos: 80 Xsize: 15

Ypos: 10 Ysize: 40

Zpos: 15 Zsize: 5

In 'Geometry', select SHAPES\HFPIPE.DAT (half pipe).

Click on 'OK' to exit from the Object Dialogue Box.

Adjust the orientation with the rotation arrow buttons (clip art rotation 5/24).

**Set remaining solution control parameters:**

In the Main Menu, click on 'Output'.

Change the monitoring point by setting

IXMON to 5,

IYMON to 12,

IZMON to 8

Click on 'Numerics'.

Set the number of iterations to 100.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

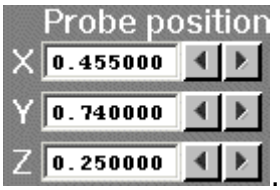
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

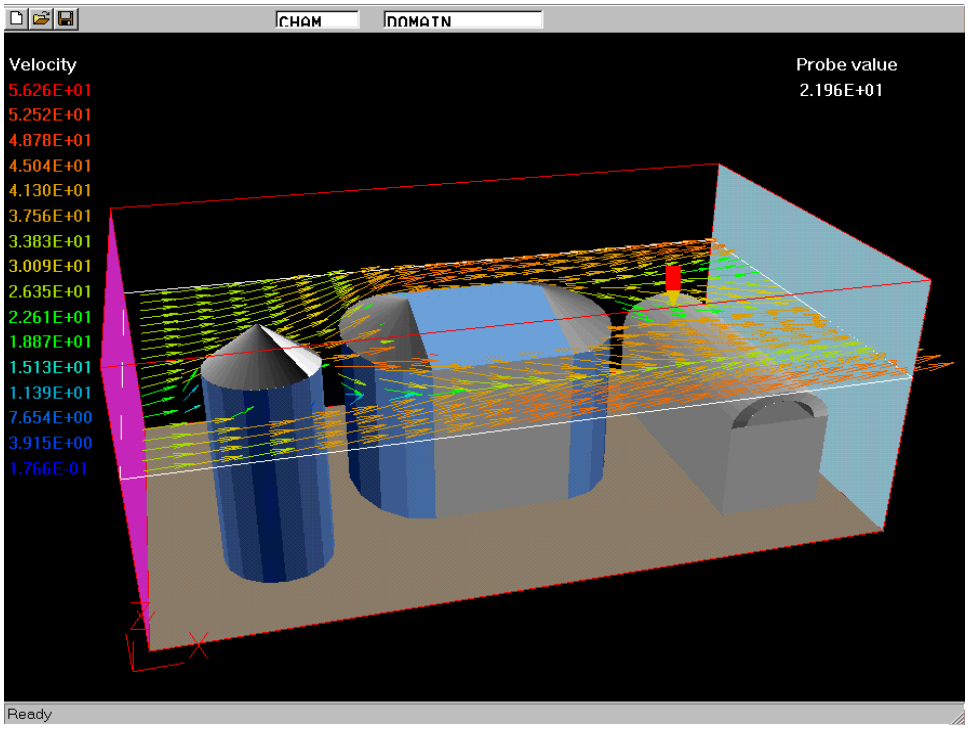
- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' 

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:




## Saving the results.

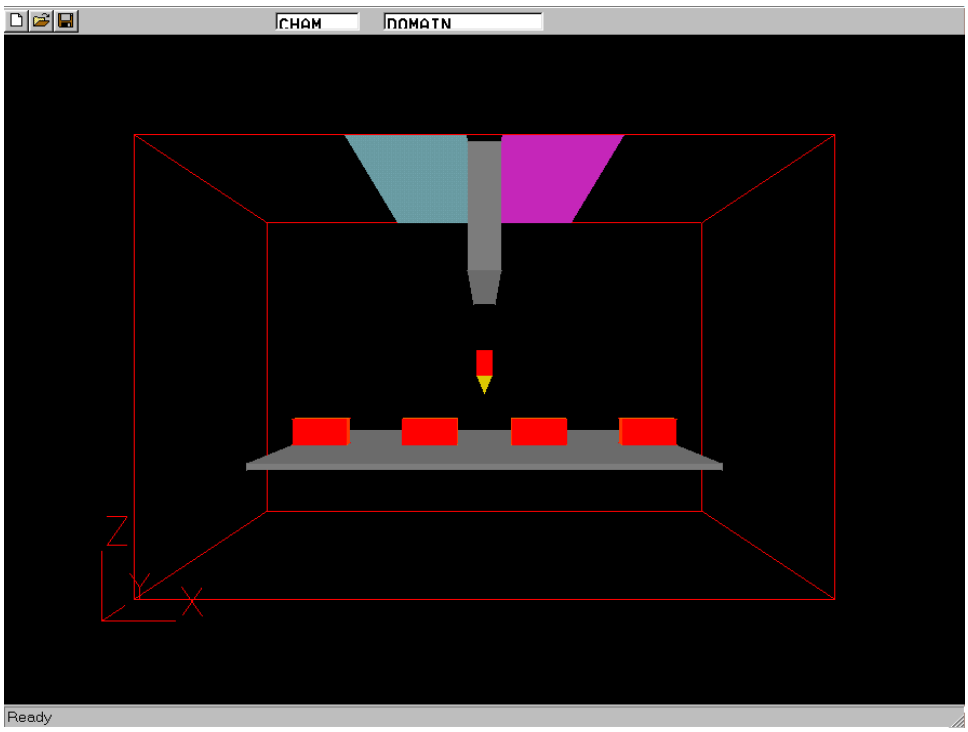
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'BUILDING' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Flow Over Heated Bricks

## Heat sources, Arraying

The following instructions will allow the user to set up and run the simulation of air in a cavity with heated bricks in VR.
The flow is three-dimensional and its geometry is shown below.




The geometry and simulation options are set up using VR Editor, the calculations are performed using the PHOENICS solver, EARTH, and the results are viewed using VR Viewer. Velocity, pressure and temperature will be simulated.

## Accessing PHOENICS-VR.

*From the system level:*

To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

### Set the domain size, and activate solution of variables:

Click on 'Main Menu' and then on 'Geometry'.

Change the Z-Domain Size to 0.7 m.

Set the Number of cells:

- 30 cells in the X-direction ,
- 22 cells in the Y-direction,
- 25 cells in the Z-direction.

Click 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Set the 'solution for velocities and pressure' ON.

Click on 'energy equation' and

select 'Temperature' as energy formulation.

Click on 'Total' thus changing it to 'Static'.

Click on 'Top menu' and then on 'OK'.

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

**Create the horizontal plate:**

Click on 'New Object'.

Change name to PLATE.

Set POSITION and SIZE of object as:

Xpos: 0.15 Xsize: 0.7

Ypos: 0.05 Ysize: 0.9

Zpos: 0.19 Zsize: 0.01

The Type is Blockage (default). Click on 'Attributes', then on 'Other materials', 'Solids' and select the material as STEEL.

Click on 'OK' to exit the Attributes menu and on 'OK' to exit the Object Dialogue Box.

**Create the vertical plate:**

Click on 'New Object'.

Change name to CUBE.

Set POSITION and SIZE of object as:

Xpos: 0.475 Xsize: 0.05

Ypos: 0.05 Ysize: 0.9

Zpos: 0.5 Zsize: 0.2

The Type is Blockage (default).

Click on 'OK' to exit from the Object Dialogue Box.

**Create the first brick:**

Click on 'New Object'.

Change name to BRICK1.

Set POSITION and SIZE of object as:

Xpos: 0.15 Xsize: 0.1

Ypos: 0.45 Ysize: 0.05

Zpos: 0.2 Zsize: 0.05

The Type is Blockage. Click on 'Attributes' and define the material as BRICK (go to OTHER MATERIALS, SOLID then to BRICK).

Click on 'Adiabatic', and select 'Fixed temperature'

Set the temperature value to 500 Celsius degrees.

Click on 'OK' to close the Attributes menu.

Click on 'OK' to close the Object Dialogue Box.

**Create the remaining bricks by arraying:**

Check that the first brick built is still selected and appears with white contours, if not click on it, then on 'OK'.

Now click on the 'Duplicate using array button'. In the new dialogue box, set

| Direction | Dimension | Pitch |
|---|---|---|

| | | |
|---|---|---|
| X | 4 | 0.2 |
| Y | 1 | 0.0 |
| Z | 1 | 0.0 |

Click on 'OK' to exit from the Array Dialogue Box.

A rank of bricks with the same attributes appears in the domain.

**Create the fluid inlet;**

Click on 'New Object'.

Change name to INLET.

Set POSITION and SIZE of object as:

Xpos: 0.525 Xsize: 0.175

Ypos: 0.0 Ysize: 1.0

Zpos: 0.7 Zsize: 0.0

Define Type: Inlet.

Click on 'Attributes' to define the inlet conditions.

Set the velocity of the inflow rate in Z-direction at -0.5 m/s.

Set temperature at 20 degrees C for the inflow.

Click on 'OK' to close the Attributes menu, and on 'OK' in the Object Dialogue Box.

**Create the fluid outlet**:

Click on 'New Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 0.3 Xsize: 0.175

Ypos: 0.0 Ysize: 1.0

Zpos: 0.7 Zsize: 0.0

Define Type: Outlet.

Leave the default values in the Attributes dialogue box.

Click on 'OK' to exit from the Object Dialogue Box.

**Set remaining solution control parameters:**

In the Main Menu, click on 'Output'.

Change the monitoring point by setting

XMON to 0.5

YMON to 0.5,

ZMON to 0.3.

Click on 'Numerics'.

Set the number of iterations to 200.

Click on 'Relaxation Control', set DTFALS (false time step) for U1, V1 and W1 to 10.

Click on 'Previous panel'.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

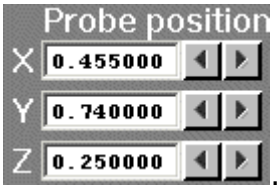
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

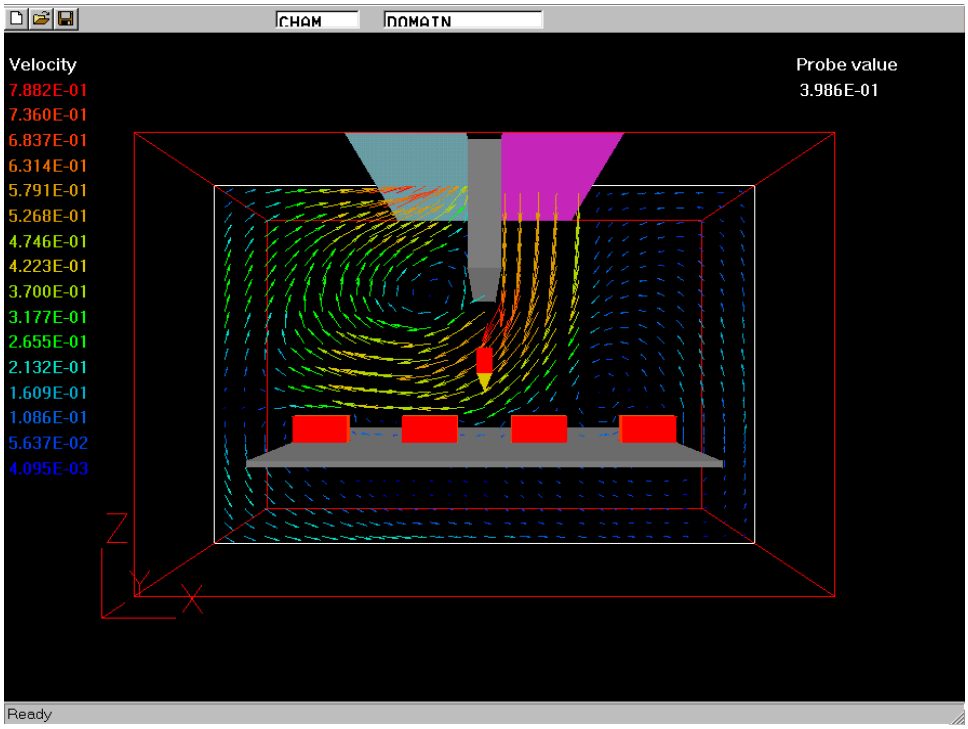
- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' 

To change the direction of the plotting plane, set the slice direction to X, Y or Z 


To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:




## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'BRICKS' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Conjugate Heat Transfer

For this workshop you can load and modify case 274 from the core library. The basic case models the isothermal flow of air past a moving van. the movement of the van is represented by giving the floor plane the same velocity as the air inlet.

## Conjugate heat Transfer

Conjugate heat transfer is active within the solids of the domain provided solid numbers 198 and 199 are not selected, and temperature is solved.
The workshop takes the basic case, activates solution of temperature, and sets appropriate physical properties for the blocks representing the van body.

## Accessing PHOENICS-VR

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Click on 'File', 'Load from Libraries', enter 274 in the Case Number data entry box, and click 'OK'.
*To enter VR Editor:*
This is the default mode of operation.

## In the VR-Editor

### Turn Solution of the Energy Equation ON

Click on 'main menu', 'Models', and 'Energy Equation'. Select 'Temperature'. Click on 'Top menu'.

Click on 'properties', and set the reference temperature to 273. From now on, all temperatures will be in degrees C. Note that you may have to click 'Page dn' to see the Reference temperature entry box.

Click 'Top Menu' then 'OK' to leave the main menu.

### Set Physical Properties for Blockages

Double-click on each blockage in turn, and select a 'real' solid for each one. Click on 'Attributes', 'Types', 'Solids', 'OK', select any solid, say Aluminium, and click 'OK'.

### Set Thermal Boundary Conditions

Double-click on the UPSTR object, click on 'Attributes', and set the inlet temperature to 20° C

Double-click on the FWHL object, then on 'Attributes'. Click on 'Adiabatic', and select Fixed Heat Flux from the list of heat sources. Set the heat source to: 30 kW. Leave the other objects and the ROAD as adiabatic.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

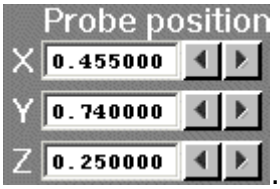
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' 

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons



.

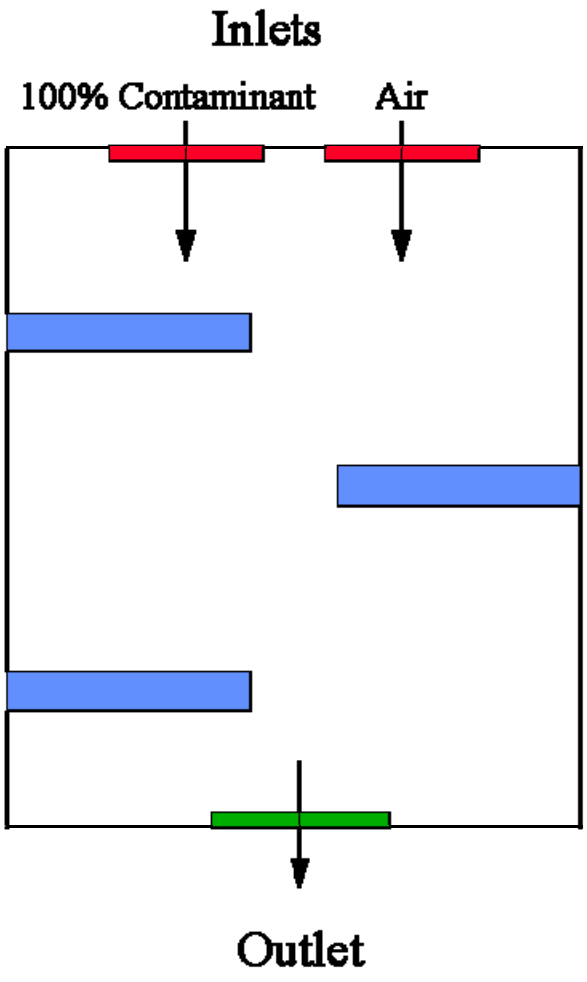
## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'CONJUGAT ' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Introducing Contaminants

In this workshop contaminants will be introduced to the flow domain.
To start with, a contaminant of the same density as air at 20° C is introduced.
The workshop is then expanded to allow the introduction of a contaminant of a different density than that of air.
The method of introduction is through the Main Menu in the VR Editor
The two-dimensional flow domain used in this workshop is shown below.



## Within PHOENICS Environment:

Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core', then click on 'OK'; to confirm the resetting.

## Within VR Editor:

### Set the domain size, and activate solution of variables

Click on 'Main menu'

Click on 'Geometry'

Change x-direction Domain size to 0.6

Set the Number of cells to 30 for both the x-direction and the z- direction.

Click on 'OK' to close the Grid mesh settings dialog.

Click on 'Models' and switch 'Solution for velocities and pressure' to ON

Change the Turbulence models to KEMODL. Click on 'OK';

Click on 'Solution controls / Extra variables'. You may have to click on 'page down' if this button is not visible near the bottom of the panel. In the SOLVE entry box, type C1. Click on the **>** button next to 'Variable', and you will see a new variable called C1. This will represent the mass fraction of contaminant. Click on 'Previous Panel'

Click on 'Top Menu'; and 'OK';

Click on 'reset view parameters' then 'Fit to window' (on the hand-set) to re-scale the displayed geometry.

**Create the CONTAMINANT INLET object.**

Click on 'New Object.'

Change name to INL1.

Change the following to:

XPos: 0.1 XSize: 0.15

YPos: 0.0 YSize: 1.0

ZPos: 1.0 ZSize: 0.0

Click on Type dialogue box and change to INLET. Click on OK.

Click on 'Attributes' and change z-direction velocity to -1.0 m/s.

Enter 1.0 in the data entry box next to 'Inlet value' for C1. Click on OK.

Click on 'OK';.

**Create the Air Inlet Object by duplication**

Click on the 'Duplicate button' and on the new object which appears at the origin.

Change the following to:

XPos: 0.35

YPos: 0.0

ZPos: 1.0

Change Name to INL2

Click on 'Attributes'.

Enter 0.0 in the data entry box next to 'Inlet value' for C1. Click on OK.

Click on OK

**Create the OUTLET object.**

Click on the 'New object' button.

Change name to OUTLET.

Change the following to:

XPos: 0.2 XSize: 0.2

YPos: 0.0 YSize: 1.0

ZPos: 0.0 ZSize: 0.0

Click on Type dialogue box and change to OUTLET. Click on OK

Click on OK

**Create the first Blockage.**

Click on 'New Object'

Change name to BLOCK1.

Change the following to:

XPos: 0.0 XSize: 0.3

YPos: 0.0 YSize: 1.0

ZPos: 0.71 ZSize: 0.02

Click on OK

**Create the second Blockage.**

Click on 'New Object'

Change name to BLOCK2.

Change the following to:

XPos: 0.3 XSize: 0.3

YPos: 0.0 YSize: 1.0

ZPos: 0.44 ZSize: 0.02

Click on OK

**Create the third Blockage**

Click on 'New Object'

Change name to BLOCK3.

Change the following to:

XPos: 0.0 XSize: 0.3

YPos: 0.0 YSize: 1.0

ZPos: 0.16 ZSize: 0.02

Click on OK

**Set the remaining solution control parameters**

Click on 'Main menu'

Click on 'Numerics'

Change 'Total number of iterations' to 200

Click on Top Menu/OK

**Set the monitoring point.**

Move Monitoring Point using the Position buttons to:

x = 0.34 y = 0.5 z = 0.5

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

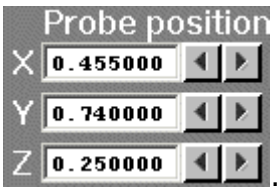
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

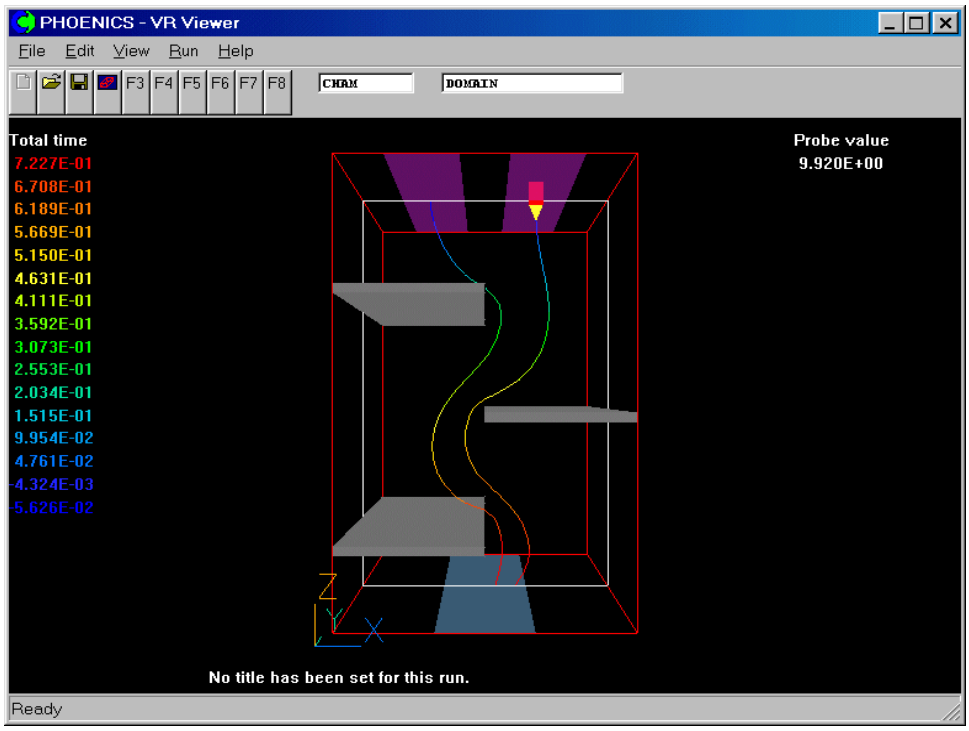
- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons




(Click on the 'C' button and select C1 to display the contaminant distribution)
A typical plot from this case is:




## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'Contaminants' (e.g.) and save as 'CASE1' (e.g.).

**In this section the density of the contaminant will be twice that of air**

## Theory

The density is to be calculated using the mass fraction in each cell, hence,

$$1/\rho = 1 / (C_1/\rho_1 + C_2/\rho_2)$$

$$\rho = \rho_1\rho_2 / (\rho_1 + (\rho_2 - \rho_1)\, C_1) \qquad (1)$$

where, $\rho_1$ is the density of the contaminant (2 x 1.189 kg/$m^3$), $\rho_2$ is the density of air (1.189 kg/$m^3$), $C_1$ is the mass fraction of contaminant and $C_2$ is the mass fraction of air (actually 1-$C_1$).

One of the available density relationships is 'Inverse linear'. This uses the following expression for density:

$$\rho = 1/(A + B\,\phi) \qquad (2)$$

where $\rho$ is the resulting density in each cell and $\phi$ is the variable.

Manipulating expression (1) to be in the same form as expression (2),

$$\rho = 1/((1/\rho_2) + (\rho_2 - \rho_1)/(\rho_1\rho_2)C_1)$$

Hence, $A = 1/\rho_2$ ; $B = (\rho_2 - \rho_1)/(\rho_1\rho_2)$ ; $\phi = C_1$

## Within VR Editor

Click on 'Run - Pre processor - GUI Pre processor (VR-Editor)' to return to the Editor.

**Set the required density relationship**

Click on 'Main Menu'

Click on 'Properties'

Click on 'Edit properties of current material'

Click on Density dialogue box and select the INVERSE-LINEAR option. Click on OK.

Set Variable to: C1

Set A to : 1/1.189 = 0.841

Set B to : (1.189 - 2*1.189)/(2*1.189*1.189) = -0.4205

Set Storage for density to : ON

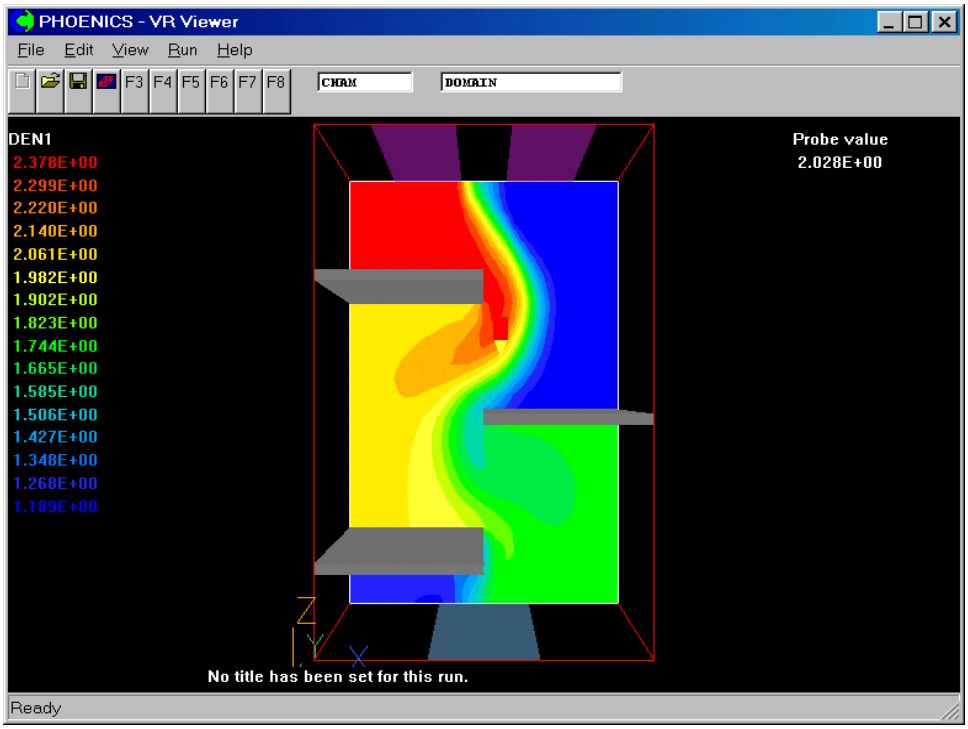
Click on Previous panel/Top Menu/OK.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) , and view the velocity vectors, contours and isosurfaces.
Click on the 'C' button and select C1 to display the contaminant distribution.
Click on the 'C' button and select DEN1 to display the density distribution.
A typical plot from this case is:



## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'CONTAMIN' (e.g.) and save as 'CASE2' (e.g.).

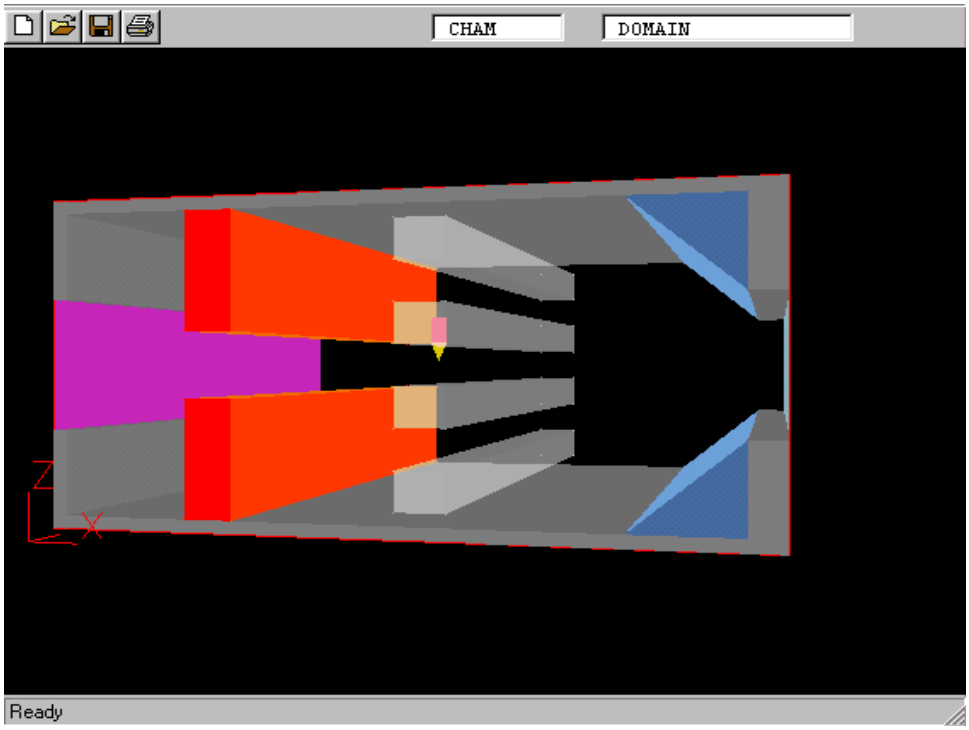
Home Page

# WORKSHOP - Editing Properties of the Domain Fluid

In this workshop the gas methane ($CH_4$) will be used as the domain fluid.
The properties of this gas are not stored in the PROPS file so the user will have to introduce them using the Main Menu. The properties to be introduced are:

- Density (Ideal Gas Law)
- Viscosity (Linear-in-T relationship)
- Specific Heat
- Conductivity
- Thermal expansion

The workshop also shows how objects 'made' from 'Domain material' can be used to make holes in other solids.
The geometry is shown below:




## Within PHOENICS VR Environment

Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.

## Within VR Editor

### Set domain Size and Activate Solution of Variables

Click on ‘Main Menu’

Click on ‘Geometry’

Click on ‘Domain size’ under ‘z-direction’ and enter 0.5

Click on ‘Number of cells’ and under x-direction enter 30. Under z-direction enter 40.

Click on 'OK' to close the Grid mesh settings dialog.

Click on ‘Models’ and switch on ‘Solution for velocities and pressure’. Switch on the Energy Equation, select Temperature and click on OK. Switch ‘TOTAL’ for the Energy Equation to ‘STATIC’.

Click on ‘Turbulence Models’ (LAMINAR) and select ‘KEMODL’. Click on OK.

### Set the properties of the Fluid

Click on ‘Properties’

Click on ‘The current domain material is’ dialogue box 'Gases' and select ’42 dummy fluid’. Click on OK

Click on ‘Edit properties of current material’

Change the following to:

**DENSITY:**

Click on Density ‘CONSTANT’ and select ‘Ideal Gas’. Enter 518 for GASCON and 1.4 for GAMMA.

Switch storage to ON.

**VISCOSITY:**

Click on Viscosity ‘CONSTANT’ and select ‘Linear in Tabs’. Click on OK.

Enter –5.96E-06 for A and 4.5838E-08 for B.

Switch storage to ON.

**SPECIFIC HEAT:**

Click on CP='1’ and enter 2230

**CONDUCTIVITY:**

Click on COND=’10' and enter 3.42E-04

**THERMAL EXPANSION COEFFICIENT:**

Click on Beta=’0' and enter 3.330E-03

Click on ‘Previous Panel’

Click on ‘Top Menu’ and 'OK';.

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

## Create the UWALL object

Click on ‘New Object’

Change Name to UWALL

Change the following to:

XPos: 0.0 XSize: 1.0

YPos: 0.0 YSize: 1.0

ZPos: 0.48 ZSize: 0.02

Click on ‘Attributes’

Click on 'Other materials', then 'Solids' and select ‘111 STEEL at 27 deg C (C=1%)’

Click on OK/OK/OK

**Create LWALL object by duplication from UWALL.**

Click on Duplicate button in the VR Editor.

Click twice on the new object which appears at the origin and change the Name to LWALL.

Click on OK

**Create WWALL1 Object**

Click on ‘New Object’

Change Name to WWALL1

Change the following to:

XPos: 0.0 XSize: 0.02

YPos: 0.0 YSize: 1.0

ZPos: 0.35 ZSize: 0.15

Click on ‘Attributes’

Click on 'Other materials', then 'Solids' and select ‘111 STEEL at 27 deg C (C=1%)

Click on OK/OK/OK

**Create WWALL2 object by duplication from WWALL1.**

Click on Duplicate button in the VR Editor and click twice on the new object

which appears at the origin. Change the Name to WWALL2. Click on OK.

**Create EWALL1 object by duplication from WWALL2.**

Click on Duplicate button in the VR Editor and click twice on the new object which appears at the origin. Change the Name to EWALL1.

Change the following to:

XPos: 0.95 XSize: 0.05

YPos: 0.0 YSize: 1.0

ZPos: 0.35 ZSize: 0.15

Click on OK.

**Create EWALL2 object by duplication from WWALL2.**

Click on Duplicate button in the VR Editor and click twice in the new object which appears at the origin.

Change the Name to EWALL2
Change the following to:
XPos: 0.95 XSize: 0.05
YPos: 0.0 YSize: 1.0
ZPos: 0.0 ZSize: 0.15
Click on OK.

**Create Upper Wedge, WED1**

Click on ‘ New Object’

Change Name to WED1
Change the following to:
XPos: 0.8 XSize: 0.15

YPos: 0.0 YSize: 1.0
ZPos: 0.35 ZSize: 0.13
Click on Geometry dialogue box and select SHAPES/WEDGE1.DAT. Click on Open.
Click on ‘Attributes’
Click on 'Other materials', then 'Solids' and select ‘111 STEEL at 27 deg C (C=1%)’
Click on OK/OK
Click on ‘Rotation and click on '>' 16/24 appears.
Click on OK/OK.

**Create Lower Wedge by duplication from WED1**

Click on Duplicate button in the VR Editor and on the new object which appears at the origin.

Change the Name to WED2
Change the following to:
XPos: 0.8 XSize: 0.15
YPos: 0.0 YSize: 1.0
ZPos: 0.02 ZSize: 0.13
Click on Rotation and on '>' until 2/24 appears.
Click on OK/OK.

**Create Upper Heat Source, HEAT1**

Click on ‘New Object’

Change Name to HEAT1
Change the following to:
XPos: 0.2 XSize: 0.07
YPos: 0.0 YSize: 1.0
ZPos: 0.3 ZSize: 0.18
Click on ‘Attributes’
Click on 'Other materials', then 'Solids' and select ’103 COPPER at 27 deg C’/OK
Set Heat Source to 500W. Click on 'Adiabatic', select 'Fixed heat flux', and enter 500 for 'Value'.
Click on OK/OK.

**Create Lower Heat Source, HEAT2, by duplication from HEAT1**

Click on Duplicate button in the VR Editor and twice on the new object which appears at the origin.

Change Name to HEAT2 and the following to:
XPos: 0.2 XSize: 0.07
YPos: 0.0 YSize: 1.0
ZPos: 0.02 ZSize: 0.18
Click on OK

**Create blockage BLOCK1**

Click on ‘New Object’

Change Name to BLOCK1
Change the following to:
XPos: 0.5 XSize: 0.07
YPos: 0.0 YSize: 1.0
ZPos: 0.02 ZSize: 0.46

Click on OK

**Create first hole in BLOCK1 - AIRHOL1** (must be created AFTER object BLOCK1 so that it overwrites the material properties of the blockage)

Click on ‘New Object’

Change Name to Airhol1
Change the following to:
XPos: 0.5 XSize: 0.07
YPos: 0.0 YSize: 1.0
ZPos: 0.04 ZSize: 0.06
Click on Attributes/Types/Gases/’Domain material’/OK/OK

Click on OK/OK

**Create remaining holes in BLOCK1 by array-copying AIRHOL1**

Click on Array button  and enter:

| Direction | Dimension | Pitch |
|---|---|---|
| X | 1 | 0.0 |
| Y | 1 | 0.0 |
| Z | 4 | 0.12 |

Click on OK

To see the hole objects, click on BLOCK1 twice to bring up the object dialog, then click 'Hide'.

**Create Inlet to domain**

Click on ‘New Object’

Change Name to INLET

Change the following to:

XPos: 0.0 XSize: 0.0

YPos: 0.0 YSize: 1.0

ZPos: 0.15 ZSize: 0.2

Click on Type dialogue box and select INLET. Click on OK.

Click on ‘Attributes’ and enter 20 degrees for temperature, and 1.0 m/s for x-direction velocity.

Click on OK/OK.

**Create Outlet from to domain**

Click on the Duplicate button in the VR Editor and twice on the new object which appears at the origin.

Change Name to OUTLET

Change the following to:

XPos: 1.0 XSize: 0.0

YPos: 0.0 YSize: 1.0

ZPos: 0.15 ZSize: 0.2

Click on the Type dialogue box and select OUTLET. Click on OK/OK.

**Set remaining solution control parameters.**

Click on Main Menu

Click on ‘Initialisation’ and enter 20 under TEM1 for the FIINIT value. If TEM1 does not appear, click '>' to scroll the list sideways until TEM1 is visible.

Click on 'Numerics' and set Total number of Iterations to 200.

Click on ‘Relaxation control’ and change relaxation factors to:

| | |
|---|---|
| U1 | 1.0 |
| W1 | 1.0 |
| TEM1 | 1000.0 |

Click on ‘Previous panel.

Click on Top menu

Click on Ok

**Set the Monitoring Point**

Adjust Monitoring position using the ‘Position’ buttons to;

X 0.4

Y 0.5

Z 0.24

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.
To view:

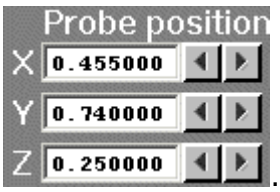
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

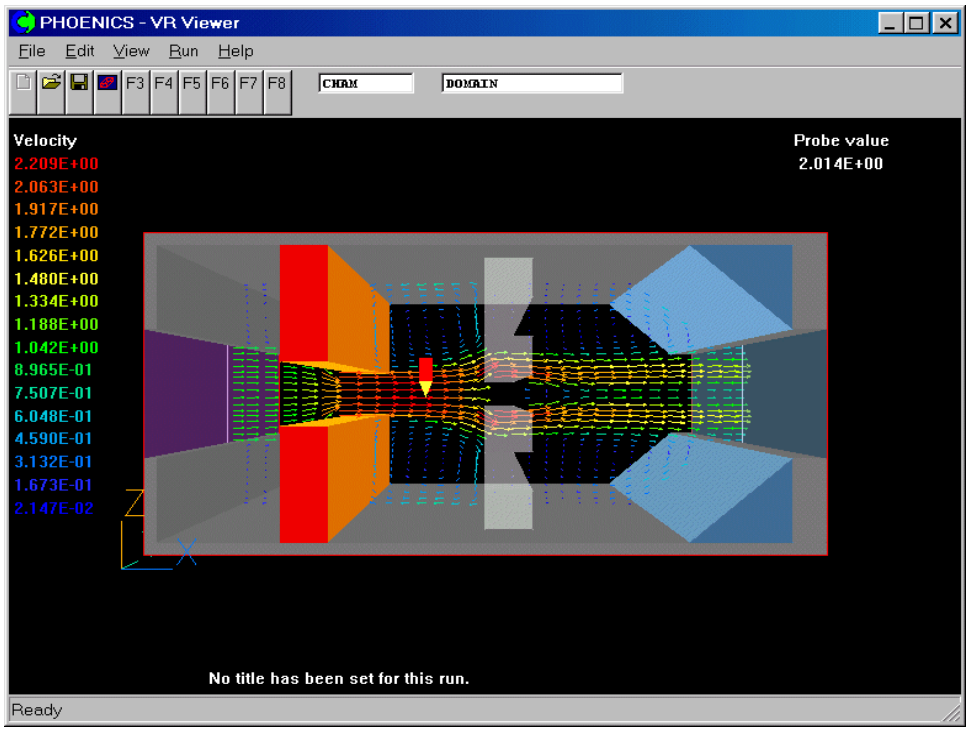
To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons




.

To make the objects see-through, click the 'Wire-frame' toggle (to the right of the Macro button on the second row of buttons). This will make the temperature contours inside the solids visible.
A typical plot from this case is:



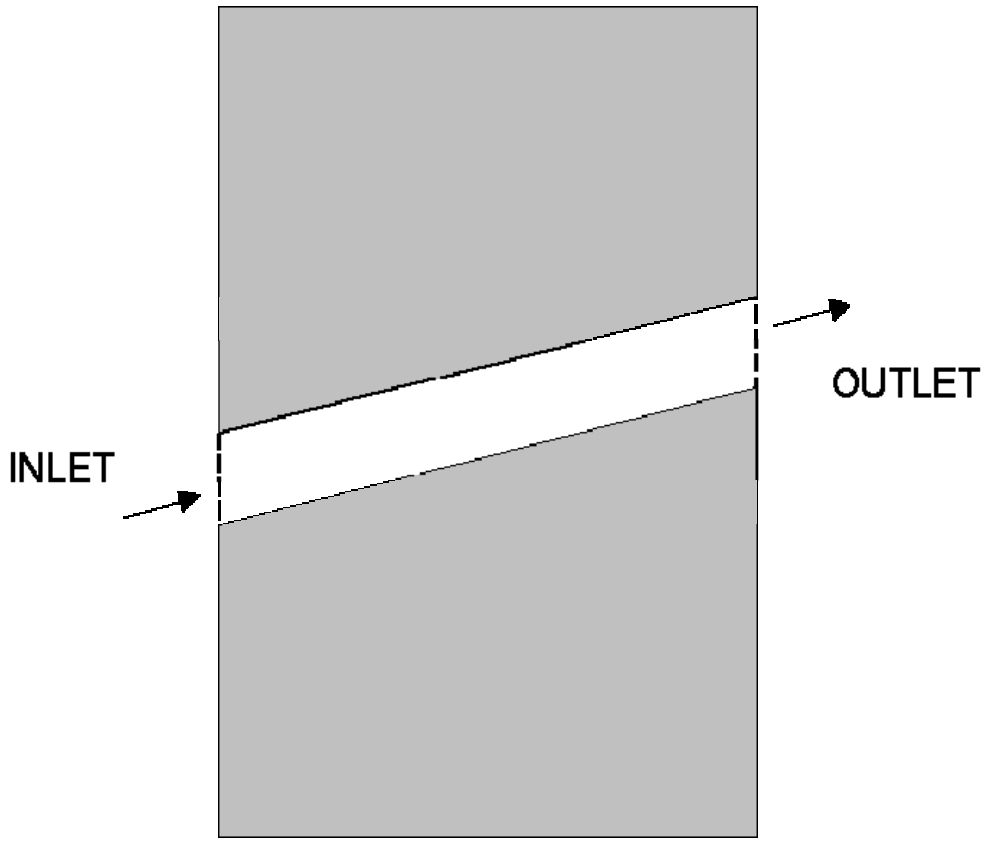

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'Properties' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP: The use of PARSOL

This workshop is designed to show the cut-cell technique called PARSOL. The geometry for the example is shown below.




The flow to be set up is a two-dimensional duct at 30° to the horizontal with **frictionless** walls.
The flow enters at 1.0 m/s, again at 30° to the horizontal.
There should hence be **no** velocity profile across the duct, and **no** pressure drop along it.
The case will be first run without PARSOL activated and then with PARSOL activated and the results compared.

## Without PARSOL Activated

### Within PHOENICS Environment:

Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core', then click on 'OK'; to confirm the resetting.

## Within VR Editor:

### Set the domain size, and activate solution of variables.

Click on 'Main Menu'**.**

Click on 'Geometry' and set the Domain size to 8.0 in the x-direction and 10.0 in the y-direction.

Set the Number of cells to 16 in the x-direction and 20 in the y-direction.

Click on 'OK' to close the Grid mesh settings dialog.

Click on 'Models' and set 'Solution for velocities and pressure' to 'ON'.

Click on 'Top menu' and 'OK';.

### Rotate the view to see the working plane (the X-Y plane).

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

Press the 'View down/move down' button to rotate the domain 90º so it can be viewed in the xy-plane rather than the default xz-plane.

Click on 'Reset' then 'Nearest head-on' to align the view exactly with the Z axis.

### Create the first (upper) angled blockage

Click on 'New Object' and enter:

XPos: 0.0 XSize: 8.0

YPos: 4.0 YSize: 6.0

ZPos: 0.0 ZSize: 1.0

Set Geometry to: SHAPES\WEDNEW.DAT

Set the name to TRAPEZ1.

Click on 'Attributes'. Change the material to '199 Solid allowing fluid slip..'.

Click on 'OK'; and 'OK';.

Click on 'Rotate object up' (second row of buttons on the handset) twice.

**Create the first (upper) angled blockage by duplication.**

Click on 'Duplicate object' button and then click on 'Rotate object down' twice. (The above will create a channel at 30° to the horizontal)

**Create the fluid inlet**

Click on 'New Object' and enter:

XPos: 0.0 XSize: 0.0

YPos: 2.0 YSize: 2.0

ZPos: 0.0 ZSize: 1.0

Set the name to IN.

Set Type to: Inlet

Click on Attributes and set:

X Direction velocity to 0.866 m/s

Y Direction velocity to 0.5 m/s

Z Direction velocity to 0.0 m/s

Click on 'OK'; and on 'OK';.

**Create the fluid outlet**

Click on 'New Object' and enter:

XPos: 8.0 XSize: 0.0

YPos: 6.0 YSize: 2.0

ZPos: 0.0 ZSize: 1.0

Set the name to OUT.

Set Type to: Outlet.

Click on 'OK';

**Set remaining Solution Control parameters**

Click on 'Main Menu'.

Under 'Numerics' set 'Total number of iterations to 100,

Click on 'Output' and set:

- IXMON = 8
- IYMON = 10
- IZMON = 1

Click on 'Top menu' and on 'OK';.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

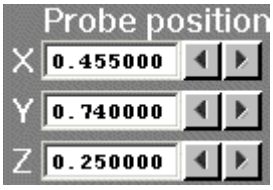
In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.
To view:

- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z 
To change the position of the plotting plane, move the probe using the probe position buttons



.

In particular, note the non-uniform velocity vectors, and the pressure drop from inlet to outlet.

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'PARSOL' (e.g.) and save as 'CASE1' (e.g.).

# With PARSOL Activated

## Within VR Editor:

Click on 'Run - Pre processor - GUI Pre processor (VR-Editor)' to return to the Editor.
**Activate PARSOL.**

Click on 'Main Menu'.

Click on 'Geometry' and set Partial Solids Treatment to ON.

Exit from the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

View the results using the VR Viewer. Note the minimal velocity variation, and the small pressure drop from inlet to outlet.
Compare the results for the velocities and pressure drops for the two cases.

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'PARSOL' (e.g.) and save as 'CASE2' (e.g.).

Home Page

# WORKSHOP - Fine Grid Embedding.

This workshop shows how to use a Fine Grid Volume to increase the local resolution. The case considered is the flow over a backward-facing step.

## Within PHOENICS Environment:

Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core’, then click on 'OK'; to confirm the resetting.

## Within VR Editor:

### Set domain size and activate solution of variables

Click on ‘Main Menu’.

Click on ‘Geometry’.
Set the Domain size to 4.0 m in X, 1.0 m in Y and 1.0 m in Z
Set the Number of cells to 25 cells in X, 1 cell in Y and 15 in Z.
Click 'OK' to close the Grid mesh settings dialog.
Click on ‘Models’.
Solution for velocities and pressure: ON
Turbulence models: KECHEN
Click on ‘Top menu’.
Click on OK.

### Create blockage for STEP

Click on ‘New Object’.

Name: STEP

Xpos: 0.0 Xsize: 0.4

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 0.5

Type: BLOCKAGE

Click on OK.

**Create Fluid Inlet**

Click on ‘New Object’.

Name: inlet

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.5 Zsize: 0.5

Type: INLET

Click on ‘Attributes’.

Set a velocity of 1.0 m/s in the x-direction.

Click on OK/OK.

**Create fluid Outlet**

Click on ‘New Object’.

Name: outlet

Xpos: 4.0 Xsize: 0.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 1.0

Type: OUTLET

Click on OK/OK.

**Create Fine-grid volume adjacent to step**

Click on ‘New Object’.

Name: finegrid

Xpos: 0.4 Xsize: 0.8

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 0.5

Type: FINE_GRID_VOL

Click on ‘Attributes’.

Fineness X: 2

Fineness Y: 1

Fineness Z: 2

This will double the grid within the fine-grid object.

Click on OK/OK.

**Set remaining solution control parameters**

Click on ‘Main menu’.

Click on ‘Numerics’.
Total number of iterations: 300
Click on ‘Relaxation control’ and change relaxation settings to:

| | |
|---|---|
| P1 | -0.7 |
| U1 | 0.5 |
| W1 | |
| KE | -0.3 |
| EP | |

Click on 'Previous panel', ‘Top menu’.
Click on OK.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

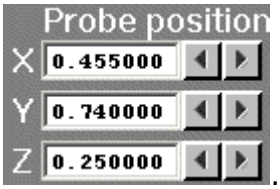
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

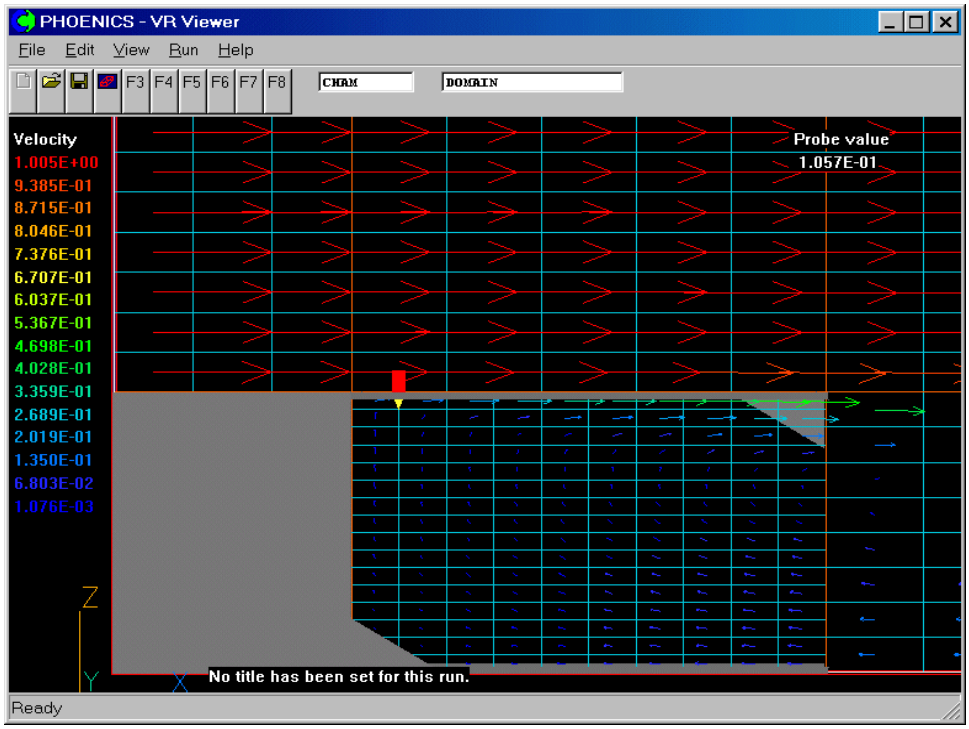
- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:



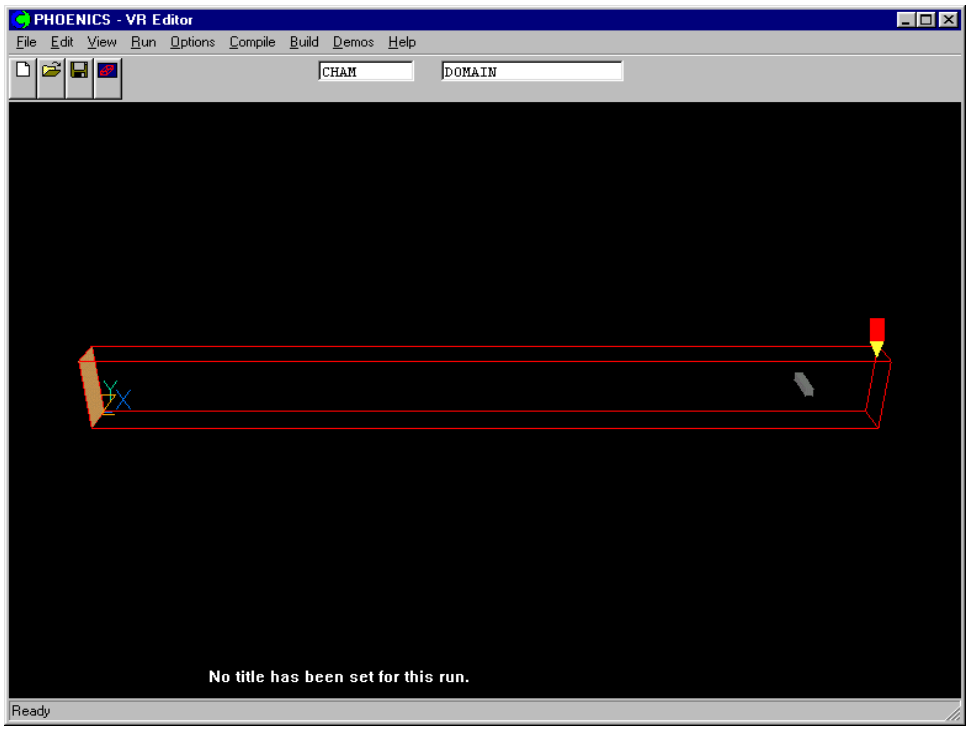

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'Finegrid' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Transient Heat Conduction

The workshop is concerned with the prediction of the transient temperature distribution in a metal bar, heated at one end.
The bar is 0.1m long, 0.01m high and 0.005m thick. It is made of steel, and one end is held at a constant temperature of 100 $^{o}$C. All the other surfaces are assumed to be adiabatic. The initial temperature is 10 $^{o}$C. We require the temperature variation of the other end during a period of 10 minutes.
The simulation is two-dimensional and its geometry is shown below.




The geometry and simulation options are set up using VR Editor, the calculations are performed using the PHOENICS solver, EARTH, and the results are viewed using VR Viewer. Only temperature need be simulated. We will use the X-Y plane for the calculation.

## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

# Base case - uniform properties

## Within VR Editor.

### Set the domain size, grid and time steps:

Click on 'Main Menu' and then on 'Geometry'.

Change the X-Domain Size to 0.1 m.

Change the Y-Domain Size to 0.01 m.

Change the Z-Domain Size to 0.005 m.

Set the number of cells in X to 60.

Set the number of cells in Y to 8.

Leave the number of cells in Z at 1.

Click on 'Steady' to toggle to 'Transient'.

Click on 'Time step settings'.

Set the 'Time at end of last step' to 600s.

Set the 'Last step number' to 120. This will produce time steps of 5s each.

Click 'OK' to close the Time step settings dialog.

Click 'OK' to close the Grid mesh settings dialog.

**Activate solution of variables:**

Click on 'Models'.

Click on 'energy equation' and select 'Temperature' as energy formulation.

**Set the material properties:**

Click on 'Properties'

Click on the button next to 'The current domain material is:'. Select 'Solids' from the list of material types, then 'Steel' from the list of solids.

**Set the Initial temperature:**

Click on 'Initialisation'.

Set the FIINIT value for TEM1 to 10.

**Set the solution control parameters:**

Click on 'Numerics'

Set the Total number of iterations to 20. This is the number of iterations (sweeps) performed on each time step.

**Set the output parameters:**

Click on 'Output', then on 'Settings' next to Field dumping.

Set the intermediate field dump step frequency to 1. Leave the other settings at their defaults. This will instruct the Earth solver to create a special output file, in which the Z axis is time, and each Z plane is the X-Y solution at that time.

Click on 'Previous panel', 'Top menu' and then on 'OK'.

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

Use the 'View down' button on the handset to rotate the image so that you are looking at the X-Y plane.

**Create the Hot end:**

Click on 'New Object'.

Change name to HOT.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.0

Ypos: 0.0 Ysize: 0.01

Zpos: 0.0 Zsize: 0.005

The Type is Blockage (default). Change the Type to PLATE.

Click on 'Attributes', then on 'Adiabatic'. Select 'Surface temperature' from the list of heat sources. Enter 100 as the Value.

Click on 'OK' to exit the Attributes menu and on 'OK' to exit the Object Dialogue Box.

**Create a temperature probe:**

Click on 'New Object'.

Change name to PROBE.

Set POSITION and SIZE of object as:

Xpos: 0.09 Xsize: 0.001

Ypos: 0.006 Ysize: 0.001

Zpos: 0.0 Zsize: 0.005

The change the Type to POINT_HISTORY.

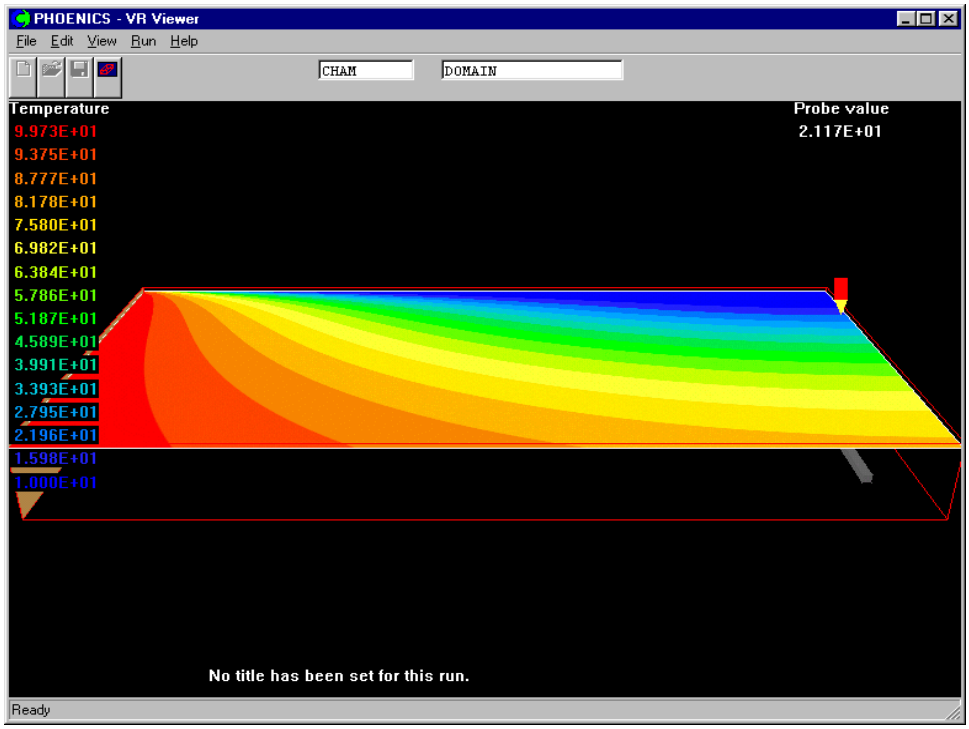
Click on 'OK' to exit from the Object Dialogue Box.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

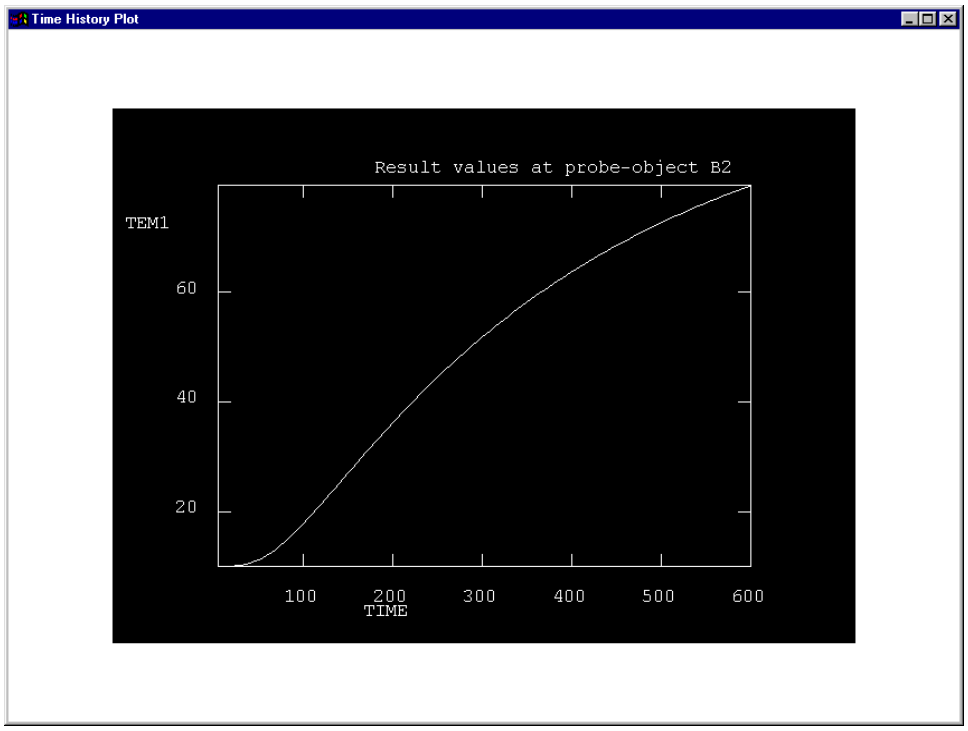
## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'.
Click 'No' next to 'Use intermediate files' to toggle to 'Yes', then click 'OK' to read the file.
Click on 'Edit', then 'Editor parameters'. Set the Z scale to 0.00008 (remove one zero from the default setting). Set the Snap Size to 0.000001, otherwise the Probe will move too far for each click on the probe position buttons.
Select temperature as the plotting variable, and turn the Contours on.
Note that the Z axis now represents time, not distance.
Contours in the Z plane show the temperature distribution at a point in time.
Contours in the Y plane show time variations at particular Y locations.




Contours in the X plane show time variations at particular X locations.

Double-click on the probe object, then click the 'Results' button on the object dialog. The time-history of temperature will be displayed.

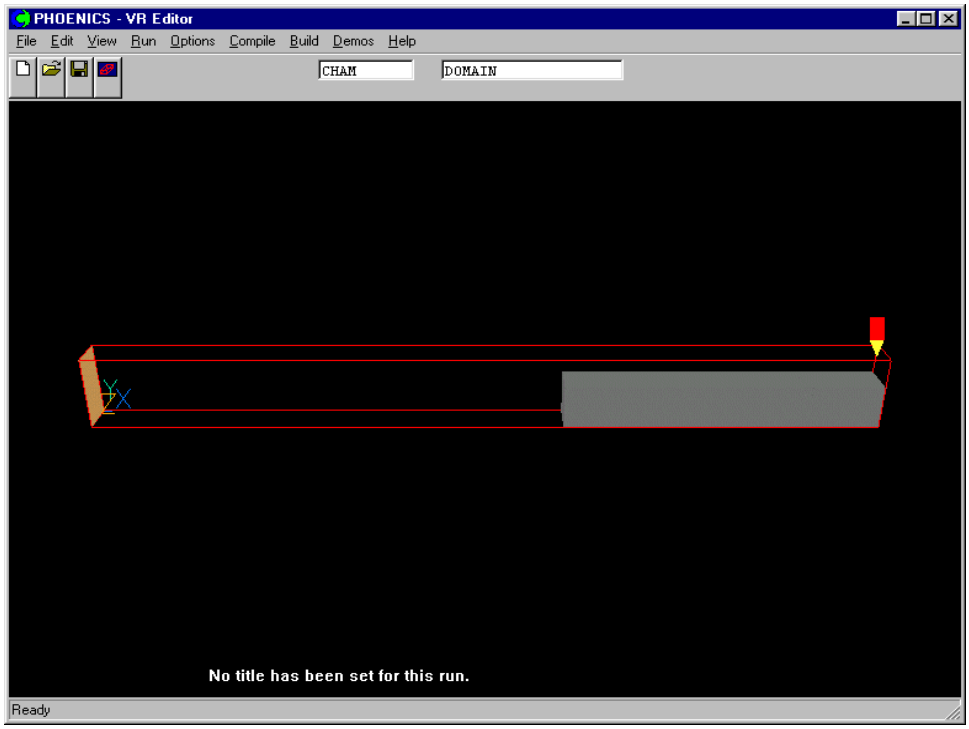



## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'HOTBAR' (e.g.) and save as 'CASE1' (e.g.).

## Second case - with insulation

We will now create an insulation object at the far end of the bar.
Click on 'Run - Pre processor - GUI Pre processor (VR-Editor)' to return to the Editor.




**Create the insulator:**

Click on 'New Object'.

Change name to HANDLE.

Set POSITION and SIZE of object as:

Xpos: 0.06 Xsize: 0.04

Ypos: 0.0 Ysize: 0.006

Zpos: 0.0 Zsize: 0.005

The Type is Blockage (default).

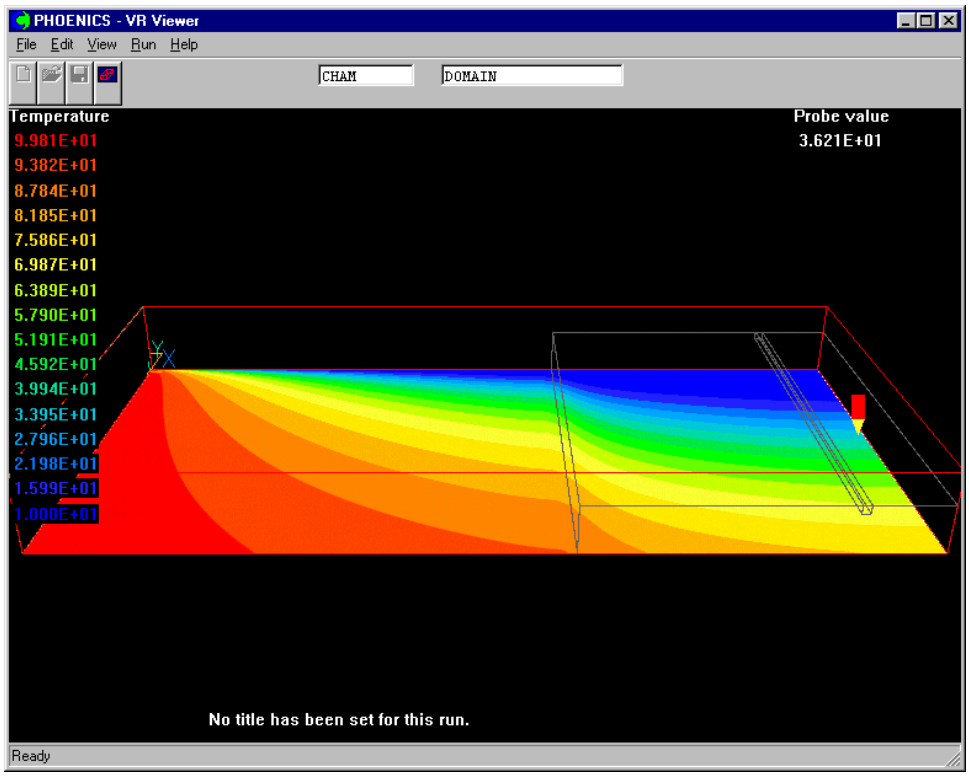
Click on 'Attributes', then on 'Other materials'. Select 'Solids', then Epoxy. Click 'OK' to close the Attributes dialog, then 'OK' to close the Object dialog.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'No' next to 'Use intermediate files' to toggle to 'Yes', then click 'OK' to read the file.
Click on 'Edit', then 'Editor parameters'. Set the Z scale to 0.00008 (remove one zero from the default setting). Set the Snap Size to 0.000001, otherwise the Probe will move too far for each click on the probe position buttons. Click on the 'Bounding box' button on the hand-set to switch to wire-frame mode. Contours inside the solid will now be visible.
Contours in the Y plane show time variations at particular Y locations - in this case on the lower surface of the bar.




Note that although the temperature stays lower on the lower surface of the insulated region, it is higher on the upper surface of the bar.

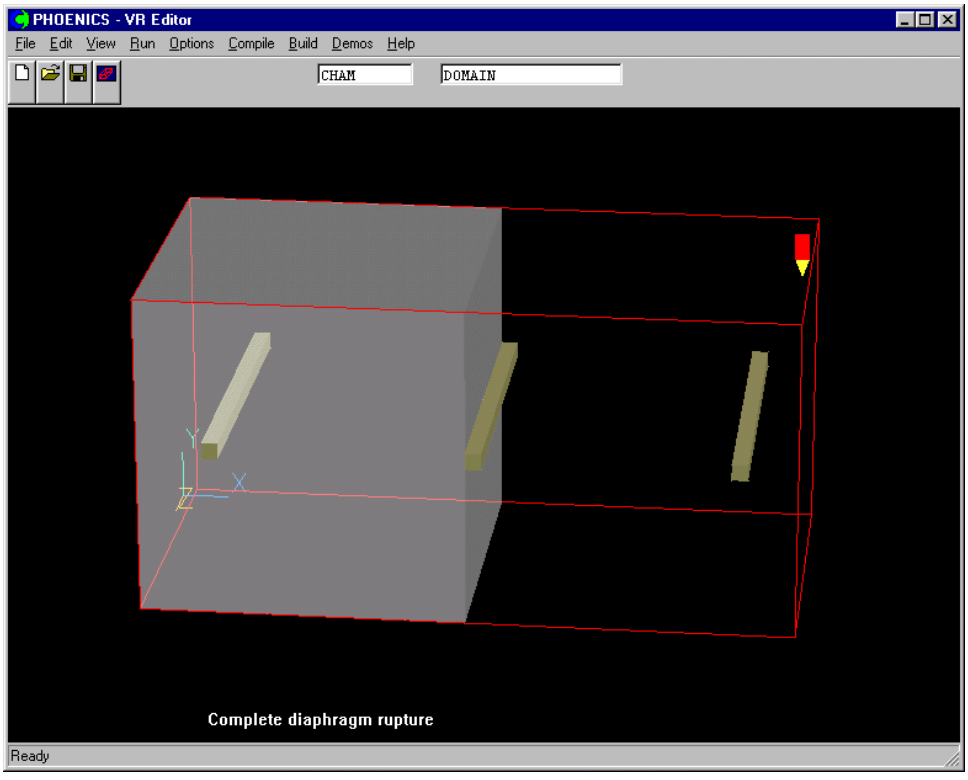
## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', and save as 'CASE2' (e.g.). in the folder 'HOTBAR' (e.g.)

Home Page

# WORKSHOP - Abrupt Rupture of a Diaphragm

The workshop is concerned with the prediction of the transient flow and temperature distribution consequent on the abrupt rupture of a diaphragm.
A chamber is divided into a high-pressure and a low-pressure zone by a diaphragm. At time t=0, the diaphragm ruptures, initiating flow.
The chamber is 0.2m long, and .1m*.1m in cross-section.
The high-pressure end is initially 1 bar above atmospheric pressure. Both ends have an initial uniform temperature of 23 $^{o}$C.
In the first case, the entire diaphragm ruptures, in the second case only the central part disappears.
The simulation is two-dimensional and its geometry is shown below.




The geometry and simulation options are set up using VR Editor, the calculations are performed using the PHOENICS solver, EARTH, and the results are viewed using VR Viewer. Only temperature need be simulated.

We will use the X-Y plane for the calculation. This allows the Z axis to be used to display **time** in the VR-Viewer.

### Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Base Case - entire diaphragm ruptures

### Within VR Editor.

Click on 'Main Menu'. Enter 'Complete diaphragm rupture' as the TITLE.
**Set the domain size, grid and time steps:**

Click on 'Geometry'.

Change the X-Domain Size to 0.2 m.

Change the Y-Domain Size to 0.1 m.

Change the Z-Domain Size to 0.1 m.

Set the number of cells in X to 40.

Set the number of cells in Y to 20.

Leave the number of cells in Z at 1.

Click on 'Steady' to toggle to 'Transient'.

Click on 'Time step distribution'.

Set the 'Time at end of last step' to 0.001s.

Set the 'Last step number' to 100. This will produce time steps of 0.00001s each.

Click 'OK' to close the Time step settings dialog.

Click 'OK' to close the Grid mesh settings dialog.

**Activate solution of variables:**

Click on 'Models'.

Turn Solution of Velocity and pressure on.

Click on 'energy equation' and select 'Temperature' as energy formulation. Click on 'Total' to toggle to 'Static'.

**Set the material properties:**

Click on 'Properties'

Click on the button next to 'The current domain material is:'. Select 'Gases' from the list of material types, then 'Air using Ideal gas Law' from the list of gases.

Note that by default the reference pressure and temperature are set to 1E5 bar and 273 K respectively.

**Set the Initial temperature:**

Click on 'Initialisation'.

Set the FIINIT value for TEM1 to 23.

**Set the solution control parameters:**

Click on 'Numerics'

Set the Total number of iterations to 30. This is the number of iterations (sweeps) performed on each time step.

Click on 'Relaxation'. Set the self-adjustment parameter, SARAH, to 0.005. Click on 'Previous panel'.

**Set the output parameters:**

Click on 'Output', then on 'Settings' next to Field dumping.

Set the intermediate field dump step frequency to 1. Leave the other settings at their defaults. This will instruct the Earth solver to create a special output file, in which the Z axis is time, and each Z plane is the X-Y solution at that time.

Click on 'Previous panel', 'Top menu' and then on 'OK'.

If the graphics image still shows the default geometry, click on 'reset view parameters' then 'Fit to window' to update the view.

Use the 'View down' button on the handset to rotate the image so that you are looking at the X-Y plane.

**Create the High Pressure end:**

Click on 'New Object'.

Change name to HIGH-P.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.1

Ypos: 0.0 Ysize: 0.1

Zpos: 0.0 Zsize: 0.1

The Type is Blockage (default). Click on 'Attributes', then on 'Other materials'. Select domain material.

Click on 'Initial values'. Set the initial pressure to 1E5 (relative to 1E5).

Click on 'OK' to exit the Attributes menu and on 'OK' to exit the Object Dialogue Box.

**Create probes:**

Click on 'New Object'.

Change name to PROBE.

Set POSITION and SIZE of object as:

Xpos: 0.18 Xsize: 0.005

Ypos: 0.05 Ysize: 0.005

Zpos: 0.0 Zsize: 0.1

The change the Type to POINT_HISTORY.

Click on 'OK' to exit from the Object Dialogue Box.

Click on the 'Duplicate object' button on the hand set. Move the duplicate object to XPOS 0.1, YPOS 0.05.

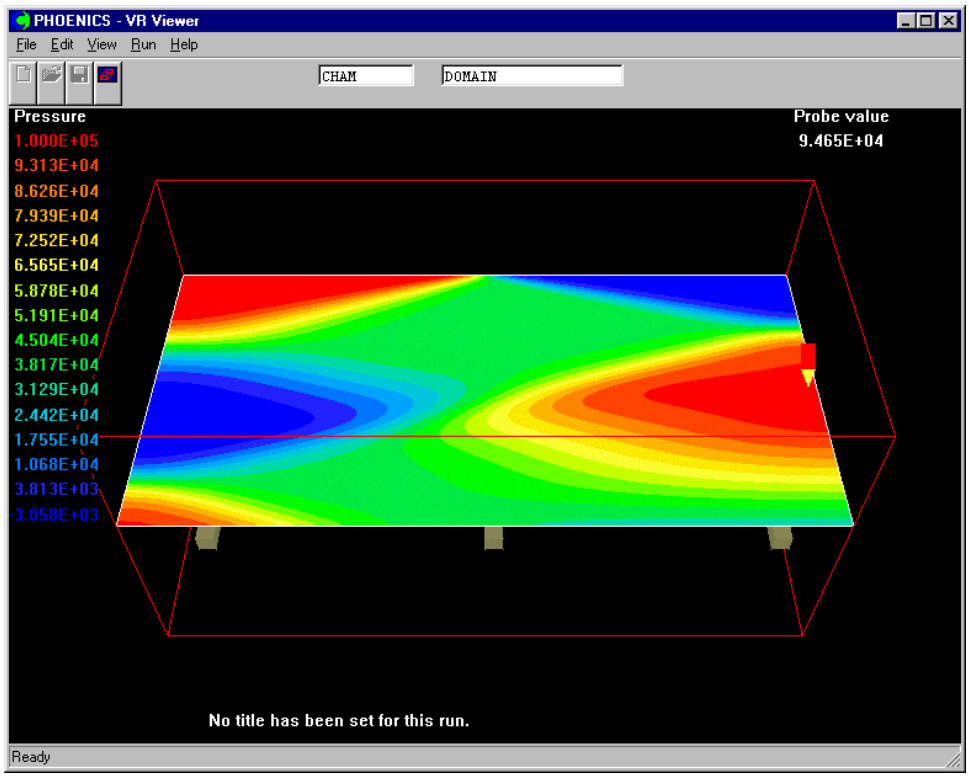
Click on the 'Duplicate object' button on the hand set. Move the duplicate object to XPOS 0.02, YPOS 0.05.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'No' next to 'Use intermediate files' to toggle to 'Yes', then click 'OK' to read the file.
Note that the Z axis now represents time. Click on 'reset view parameters' then 'Fit to window' to update the view if necessary.
Select Pressure as the plotting variable, and turn the Contours on.
Contours in the Z plane show the temperature distribution at a point in time.
Contours in the Y plane show time variations at particular Y locations.



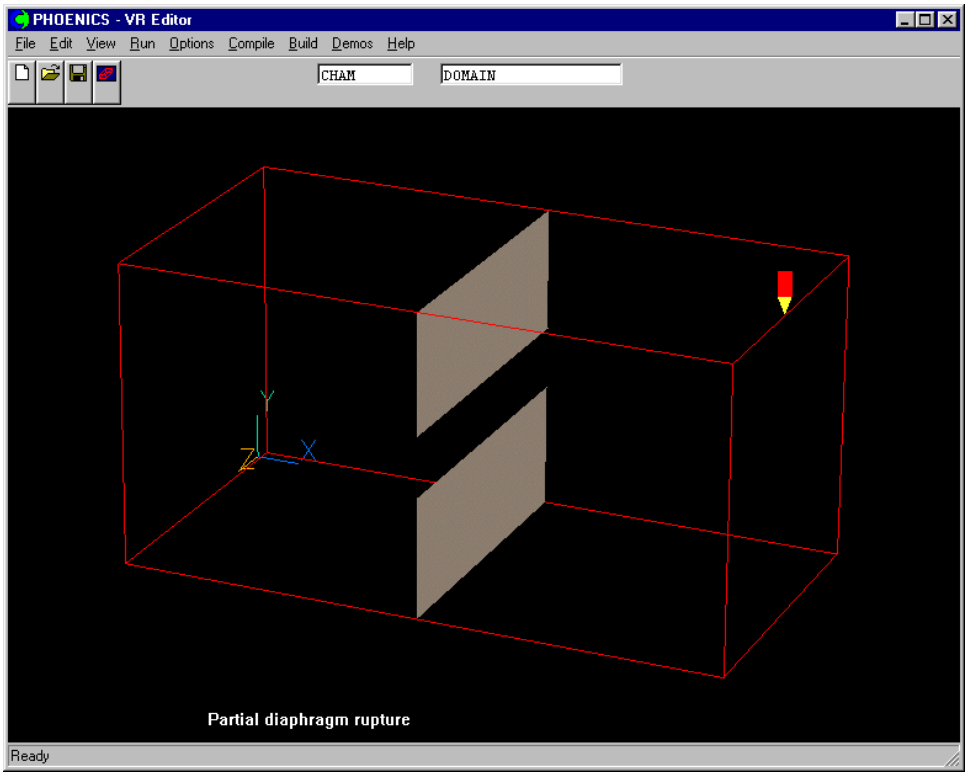

Contours in the X plane show time variations at particular X locations.
Double-click on a probe object, and click the 'Results' button on the object dialog. Select a variable, and the time-history of that variable will be displayed.

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'BURST' (e.g.) and save as 'CASE1' (e.g.).

## Second case - with partial rupture

We will now assume that the diaphragm did not rupture completely. We can do this by introducing plates at the edge of the high-pressure region.




Click on 'Run - Pre processor - GUI Pre processor (VR-Editor)' to return to the Editor.
Click on 'Main Menu'. Enter 'Partial diaphragm rupture' as the TITLE. Click 'OK'.

**Create the lower plate:**

Click on 'New Object'.

Change name to PLATE1.

Set POSITION and SIZE of object as:

Xpos: 0.1 Xsize: 0.0

Ypos: 0.0 Ysize: 0.04

Zpos: 0.0 Zsize: 0.1

The Type is Blockage (default). Change to Plate.

Click on 'OK' to close the Object dialog.

**Create the upper plate:**

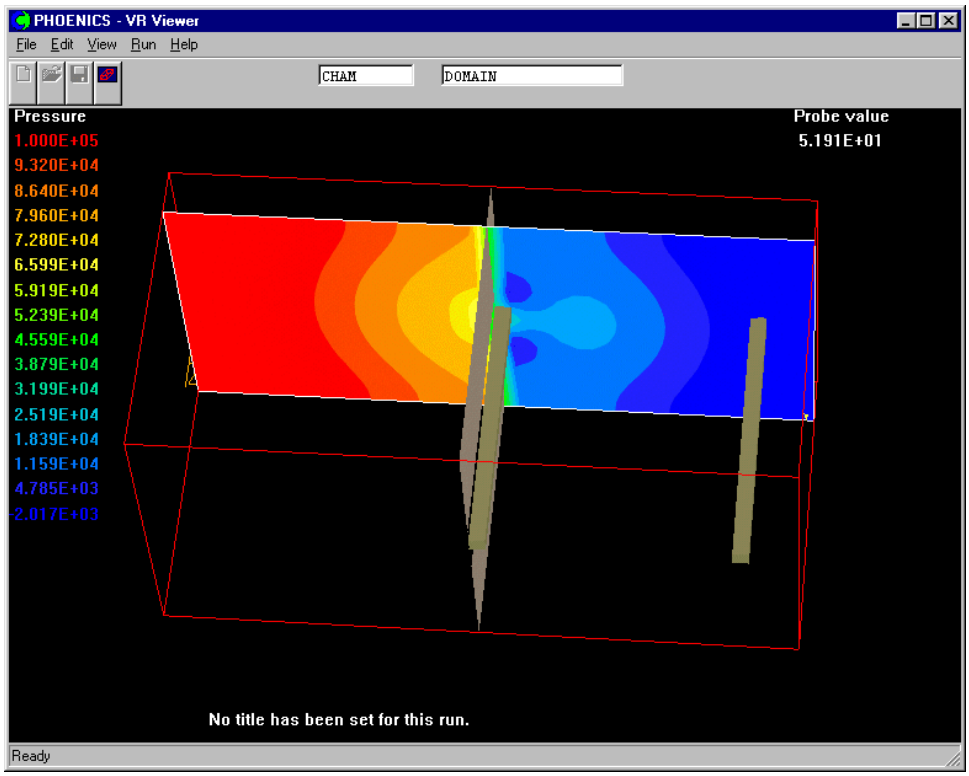
Click on 'Duplicate object'. Change the position of the duplicated object to XPOS=0.1, YPOS=0.06.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'No' next to 'Use intermediate files' to toggle to 'Yes', then click 'OK' to read the file.
Select Pressure as the plotting variable, Z as the plotting plane and turn the Contours on.
Contours in the Z plane show time variations at particular times - in this case near the start of the transient.




## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', and save as 'CASE2' (e.g.). in the folder 'BURST' (e.g.)

Home Page

# WORKSHOP - BFC Grid Generation (Points and Lines)

## Introduction

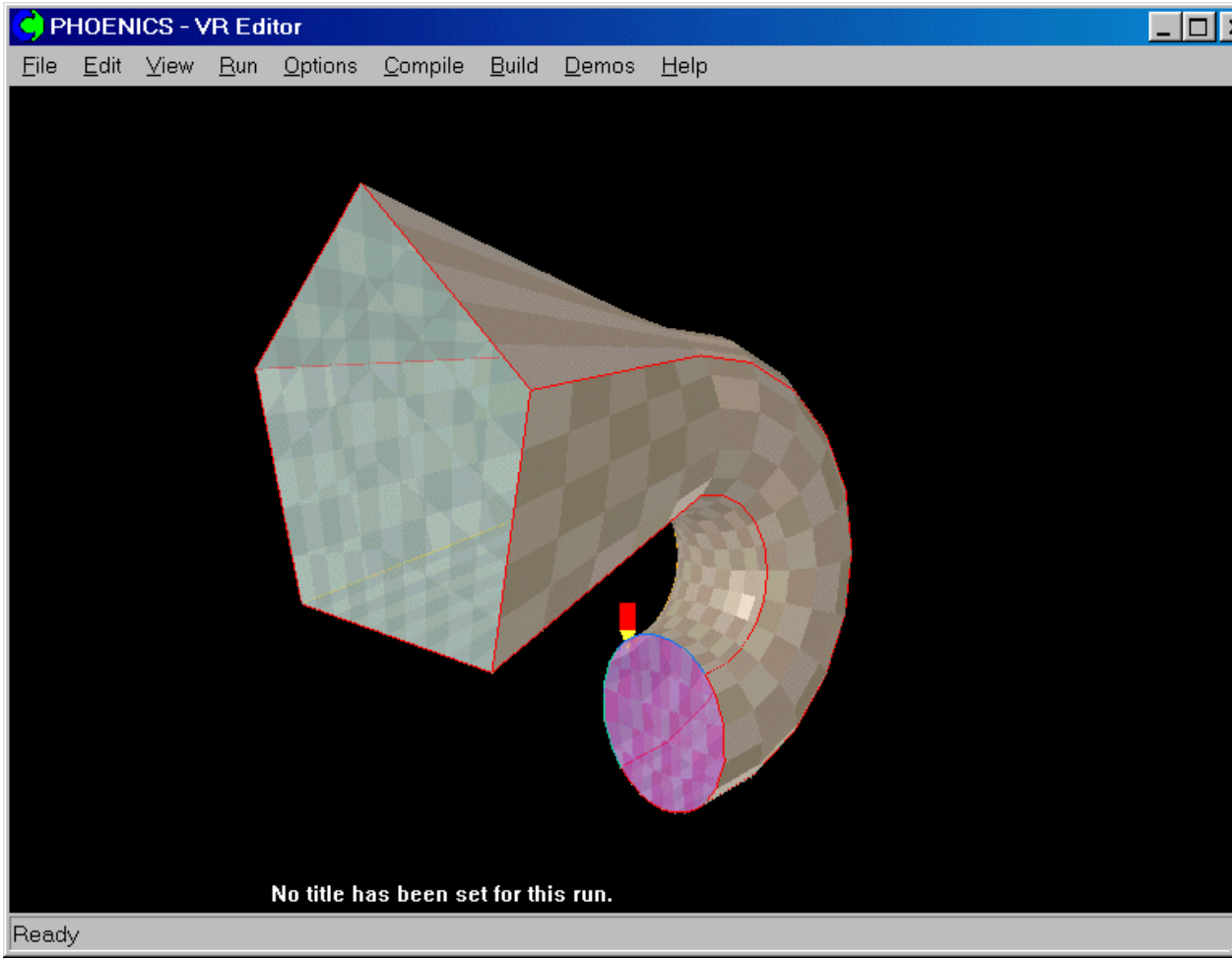
In this tutorial you will learn about:

- creating points for Body-Fitted Coordinates;
- joining points with lines; and
- some of the basic view control functions.

At the end of the tutorial you may like to confirm your understanding by the following exercises:

- creating some additional points using the mouse;
- moving existing points;
- joining points with new lines; and
- repeat the above using the mouse to position the points

In BFC, as for Cartesian/polar co-ordinates, the geometry is created first, and the mesh is fitted later. This tutorial deals with the creation of the geometry for a duct, where the cross-section changes from a pentangle to a circle along the length.




The geometry is defined in terms of Points and Lines.
Points represent either physical locations, or are used as control points for Lines.
Lines can be straight, arc segments, or splines through sequences of points.
The method employed to generate a BFC mesh requires the generation of data entities in a specific order.
The order of the data entity generation and modification is:

1. define suitable geometric POINTS;
2. connect points with LINES;
3. state the CELL distribution on each line;
4. group lines to make SURFACES, which are composed of FRAMES;
5. indicate which mesh points MATCH the frames; and
6. create VOLUMES by combining surfaces in various ways.

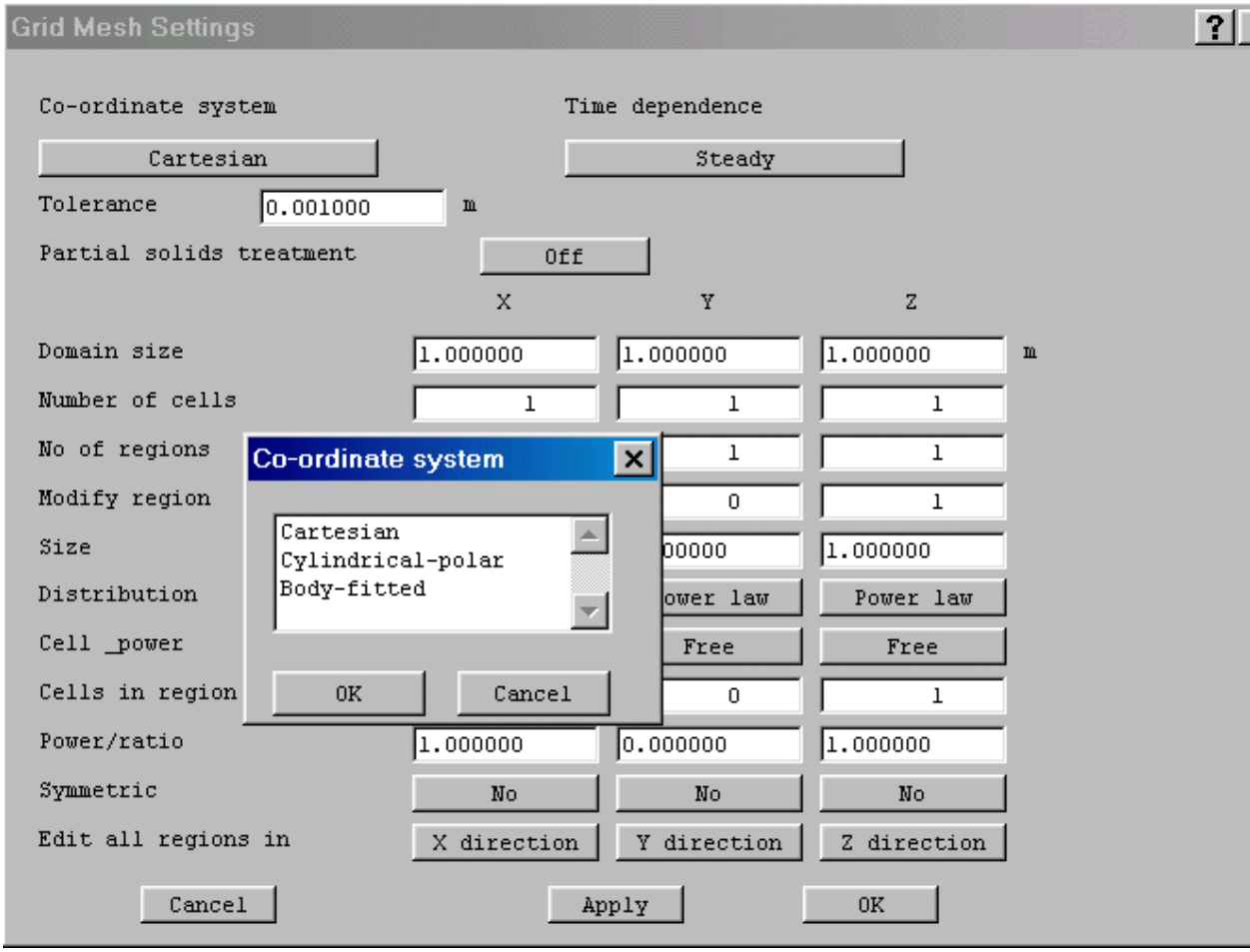
This tutorial covers stages 1 & 2.

## Accessing PHOENICS-VR.

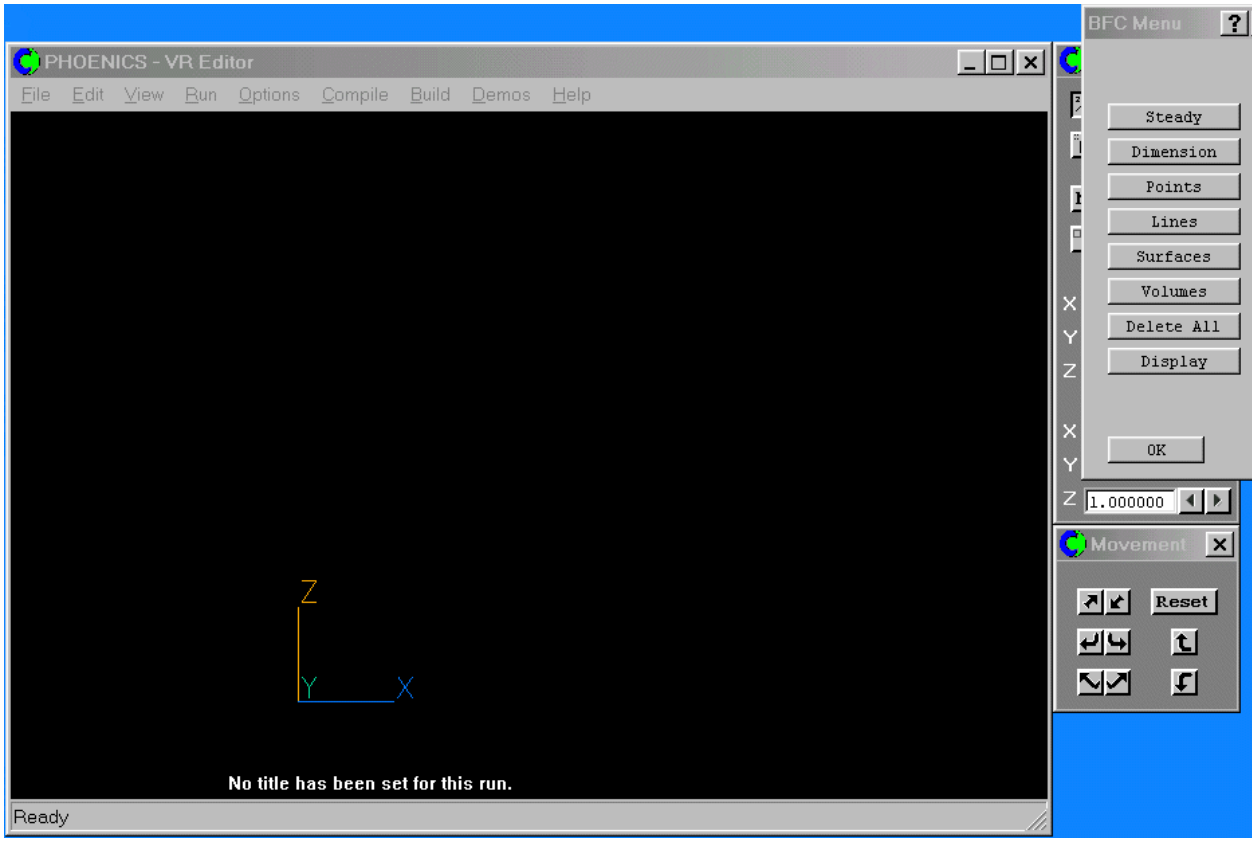
*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

Enter BFC mode by clicking on the Grid Mesh toggle  on the hand-set, then clicking anywhere within the domain to bring up the 'Grid Mesh Settings' dialog. Click on **Coordinate System - Cartesian**, and select **Body-fitted**.

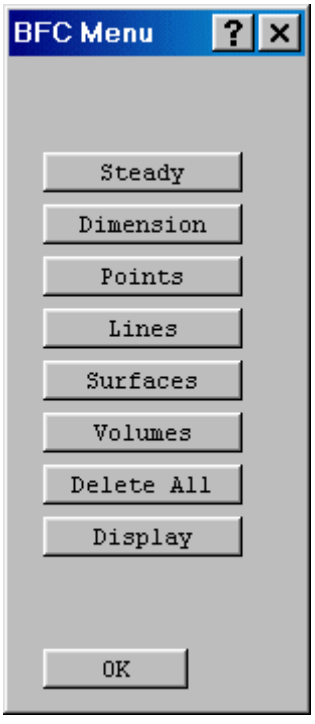



Confirm the change to BFC. Activate the Grid Generator by clicking anywhere in the domain again. A new dialog will appear, overlaid over the hand-set.




This is the main BFC Menu dialog. It contains buttons for:

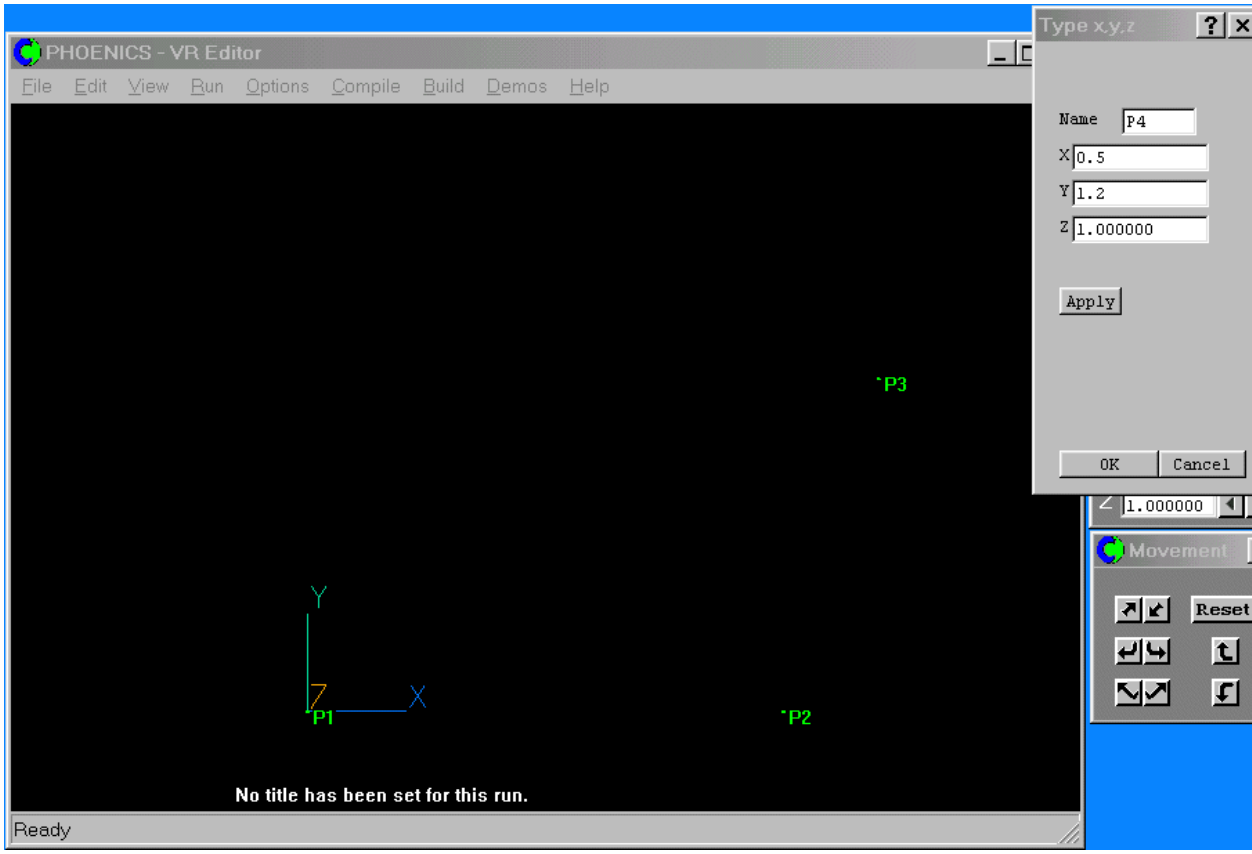
| |
|---|
| switching between steady and transient mode |
| setting the total number of cells, and the default domain size |
| creating and managing Points |
| creating and managing Lines |
| creating and managing Surfaces |
| creating and managing Volumes |
| deleting unwanted entities |
| managing the display |



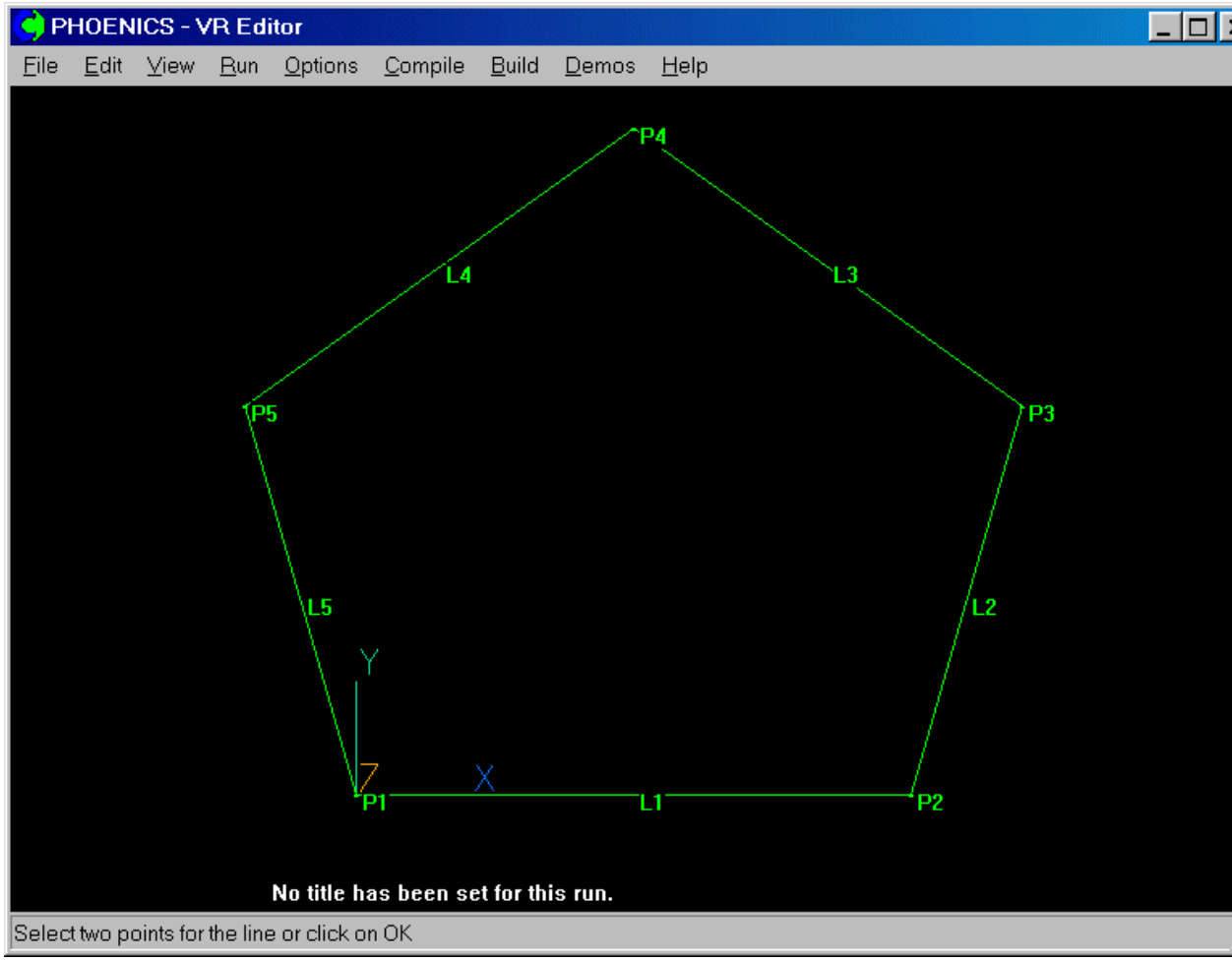
The input and modification of POINTS is often easier if the view direction is normal to the plane the points are to be created on.
We want to create points on planes of constant Z so change the view to be along the z-axis by clicking the 'View down' button on the Movement control panel, and keeping it pressed until the view is more or less along Z, with Y pointing up and X to the right.
To ensure that the axis orientation is absolutely as desired, click the 'Reset' button to jump the view to be along the nearest axis.
For this Tutorial, we are going to specify two sets of points, the pentangle at z=1, and the circle at z=0.
We now create the required shape at z=1, first by generating points, and then connecting them together with lines.
To generate points, click the 'Points' button, then 'New points', then 'Type x,y,z'. A dialog will appear, which allows the exact co-ordinates of the point to be typed in with the keyboard. The units of the co-ordinates are metres.
Create five points with the following co-ordinates:

| Point name | X co-ordinate | Y co-ordinate | Z co-ordinate |
|---|---|---|---|
| P1 | 0.0 | 0.0 | 1.0 |
| P2 | 1.0 | 0.0 | 1.0 |
| P3 | 1.2 | 0.7 | 1.0 |
| P4 | 0.5 | 1.2 | 1.0 |
| P5 | -0.2 | 0.7 | 1.0 |

After entering the data for each point, click 'Apply' to create the point and keep the dialog open for the next point. After entering the data for point P5, click 'OK' to create the point and close the dialog. The screen should look like this when creating point P4:




Click 'OK' twice to get back to the BFC Menu dialog.
Having specified the points, create the lines to join the points together.
For this particular geometrical plane, only straight lines are required; these are generated by clicking the 'Lines' button, then 'New lines', then 'New Straights'.
Click on point P1. It will be highlighted in red to confirm selection. Select point P2. Line L1 is generated. The names of the lines are automatically generated by the grid generator and are L1, L2, .... Ln.
Now generate the other lines at z=1, by clicking on points P2 and P3, P3 and P4, P4 and P5, and finally P5 and P1. At this point, the screen should look like this:




Click 'OK' twice to get back to the BFC Menu dialog. Now create the points and lines at z =0.

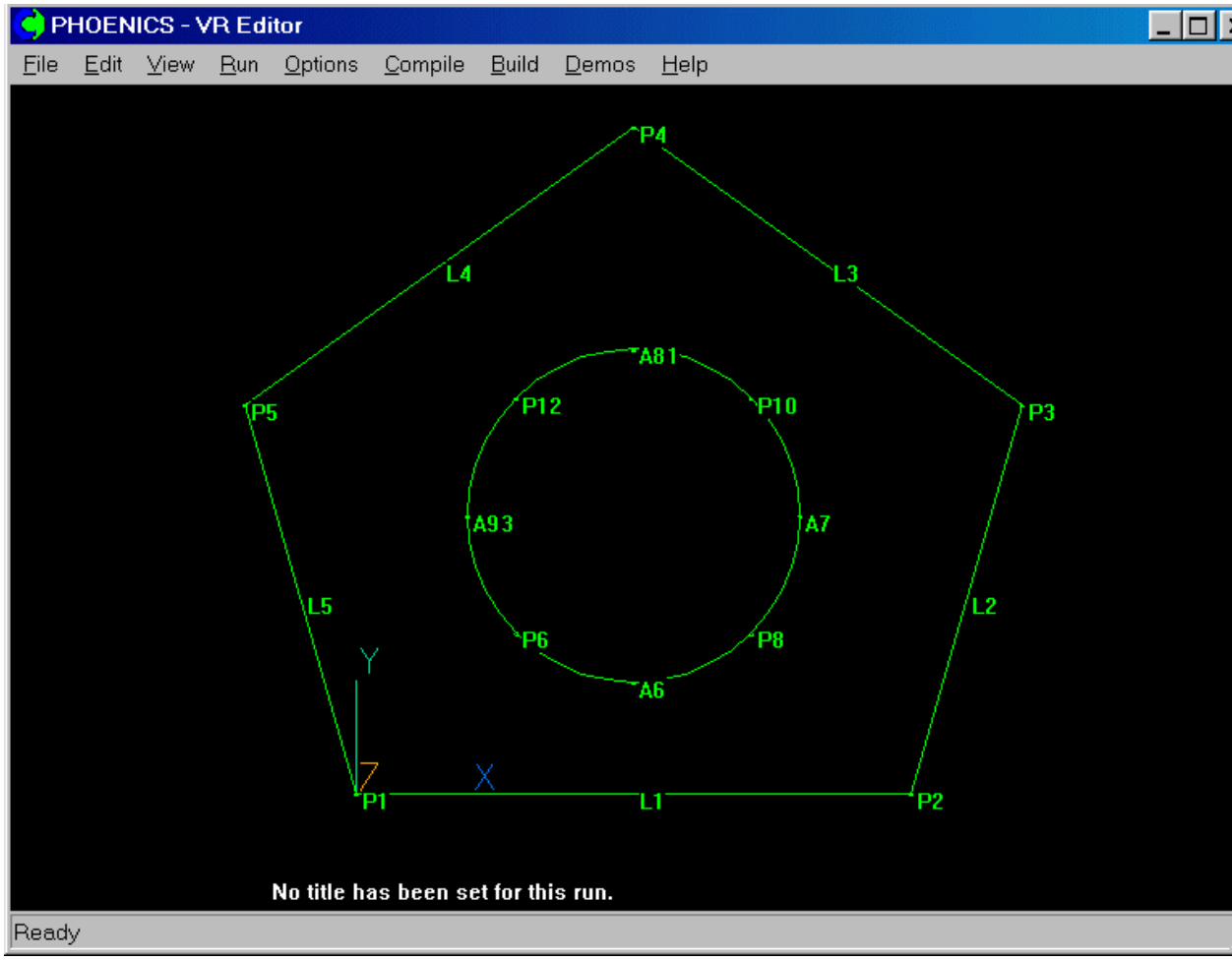
The circle will be made up of four arc segments. Each arc segment is defined by three points. We thus need eight points for the circle. The circle has radius 0.3, and its centre is at (0.5,0.5,0.0)

Click on 'Points', 'New points' then 'Type x,y,z', and create eight points with the following co-ordinates:

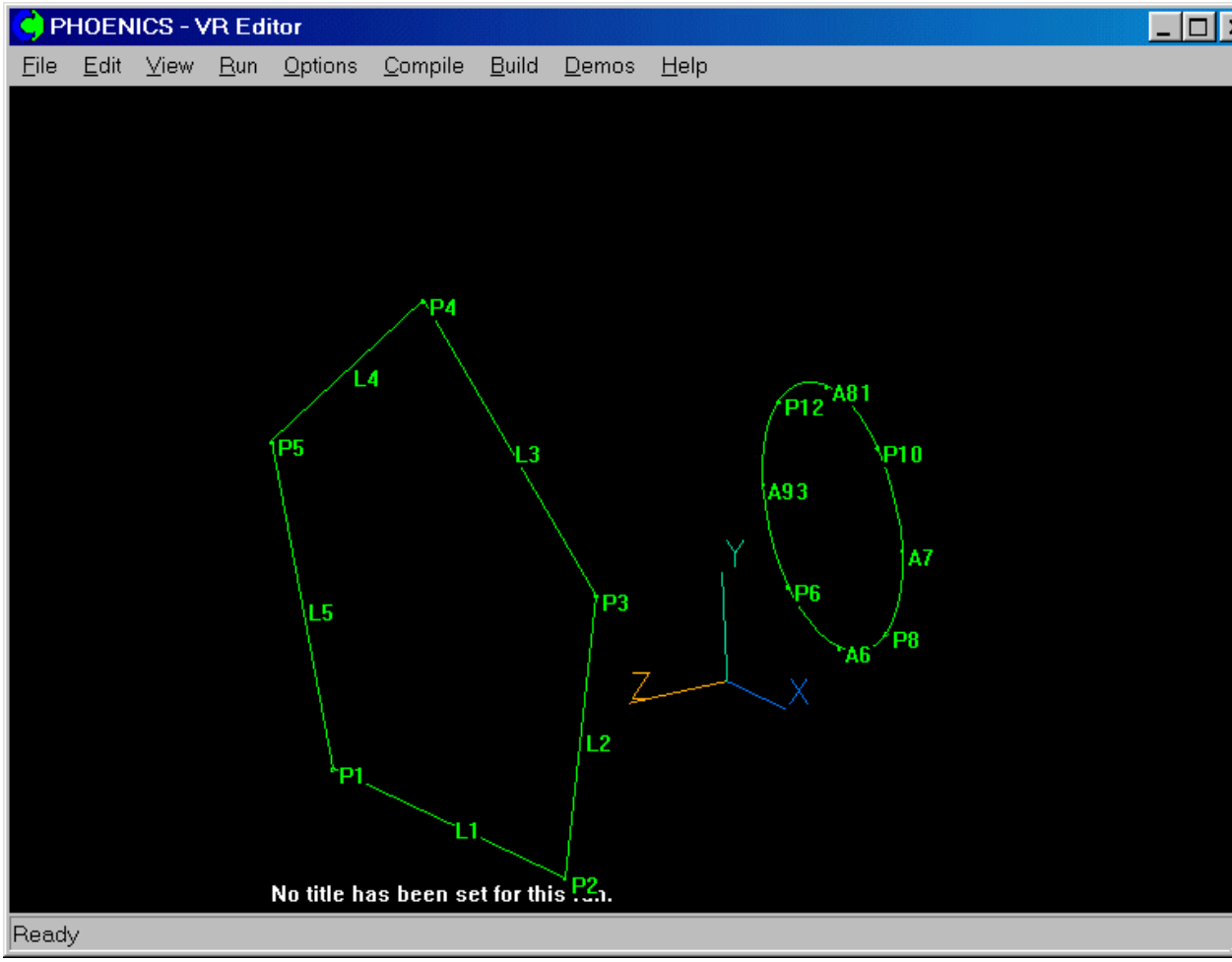
| Point name | X co-ordinate | Y co-ordinate | Z co-ordinate |
|---|---|---|---|
| P6 | 0.288 | 0.288 | 0.0 |
| P7 | 0.500 | 0.200 | 0.0 |
| P8 | 0.712 | 0.288 | 0.0 |
| P9 | 0.800 | 0.500 | 0.0 |
| P10 | 0.712 | 0.712 | 0.0 |
| P11 | 0.500 | 0.800 | 0.0 |
| P12 | 0.288 | 0.712 | 0.0 |

| | | | |
|---|---|---|---|
| P13 | 0.200 | 0.500 | 0.0 |

After entering the data for point P13, click 'OK' to create the point and close the dialog.
Define the arc segments by clicking the 'Lines' button, then 'New lines' and 'new Arcs'. Arcs are defined by 3 points, selected in order from one end to the other. Click on point P6. It will be coloured red to confirm selection. Click on point P7, the control point for the first arc - it will be coloured red on selection. Finally click point P8, the last point on the first arc. The arc itself is drawn. Note the arc number is the next line number in sequence, but has an 'A' prefix.
Now complete the remaining arcs, by clicking on P8,P9 and P10, P10,P11 and P12, then P12, P13 and P6.
Once the arcs have been created, the screen should look like this:




Change the view point by using the View control buttons on the Movement control panel.
If necessary, click on 'Reset' then 'Fit to window' to re-size the window to show all the lines etc.




The geometry required for the duct has now been defined. The next tutorial shows how a mesh is fitted to the geometry.

If you want to proceed directly to the next tutorial, click here.

To summarise, in this tutorial you have learnt about:

- creating points for Body-Fitted Co-ordinates;
- joining points with lines; and
- some of the basic view control functions.

We now suggest you practice each of these functions 'live' for yourself with the following exercises:

- creating some additional points using the mouse;
- moving existing points;
- joining points with new lines; and
- repeat the above using the mouse to position the points.

To leave the BFC Menu, click 'OK'. To save the work so far, click on 'File', 'Save as a case', make a new folder called 'BFC ' (e.g.), and save as 'BFC1'.
Note that at this point no mesh has been created, so the VR-Editor will still show the default cube!

Home Page

# WORKSHOP - BFC Grid Generation (Making Surfaces)

## Introduction

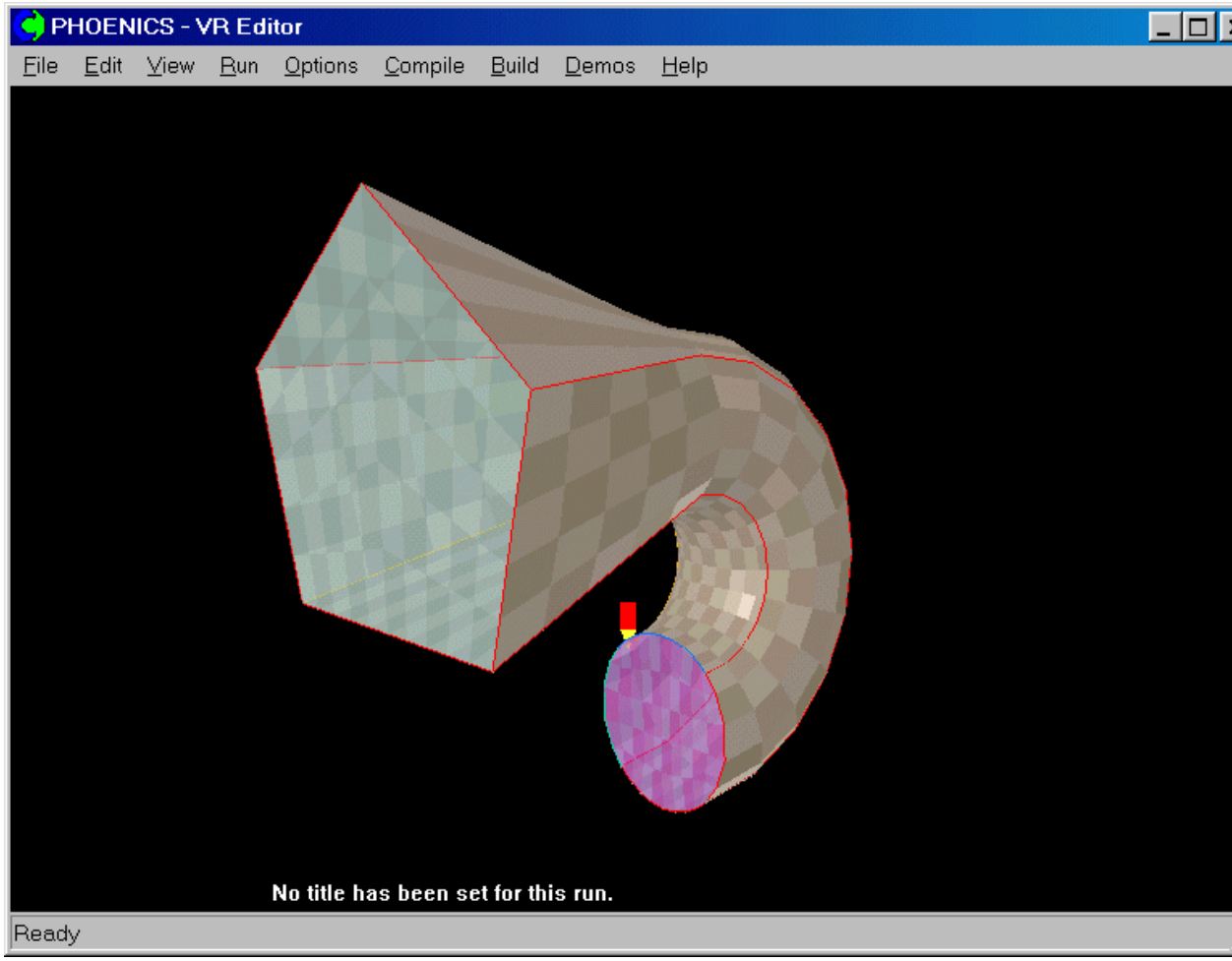
In this tutorial you will learn about:

- setting the number of cells along a line; and
- making Surfaces from Frames.

At the end of the tutorial you may like to confirm your understanding by the following exercises:

- Create points, join them with lines, and then make frames.
- Set the number of cells on opposing edges to make valid frames.

In BFC, as for Cartesian/polar co-ordinates, the geometry is created first, and the mesh is fitted later. This tutorial deals with the creation of the geometry for a duct, where the cross-section changes from a pentangle to a circle along the length.




The order of the data entity generation and modification is:

1. define suitable geometric POINTS;
2. connect points with LINES;
3. state the CELL distribution on each line;
4. group lines to make SURFACES, which are composed of FRAMES;
5. indicate which mesh points MATCH the frames; and
6. create VOLUMES by combining surfaces in various ways.

This tutorial covers stages 3 & 4.
In BFC Tutorial 1, we created the geometry for the pentangle and circle.
We now start the process of meshing by defining the cell distribution along the lines, and then creating surfaces.

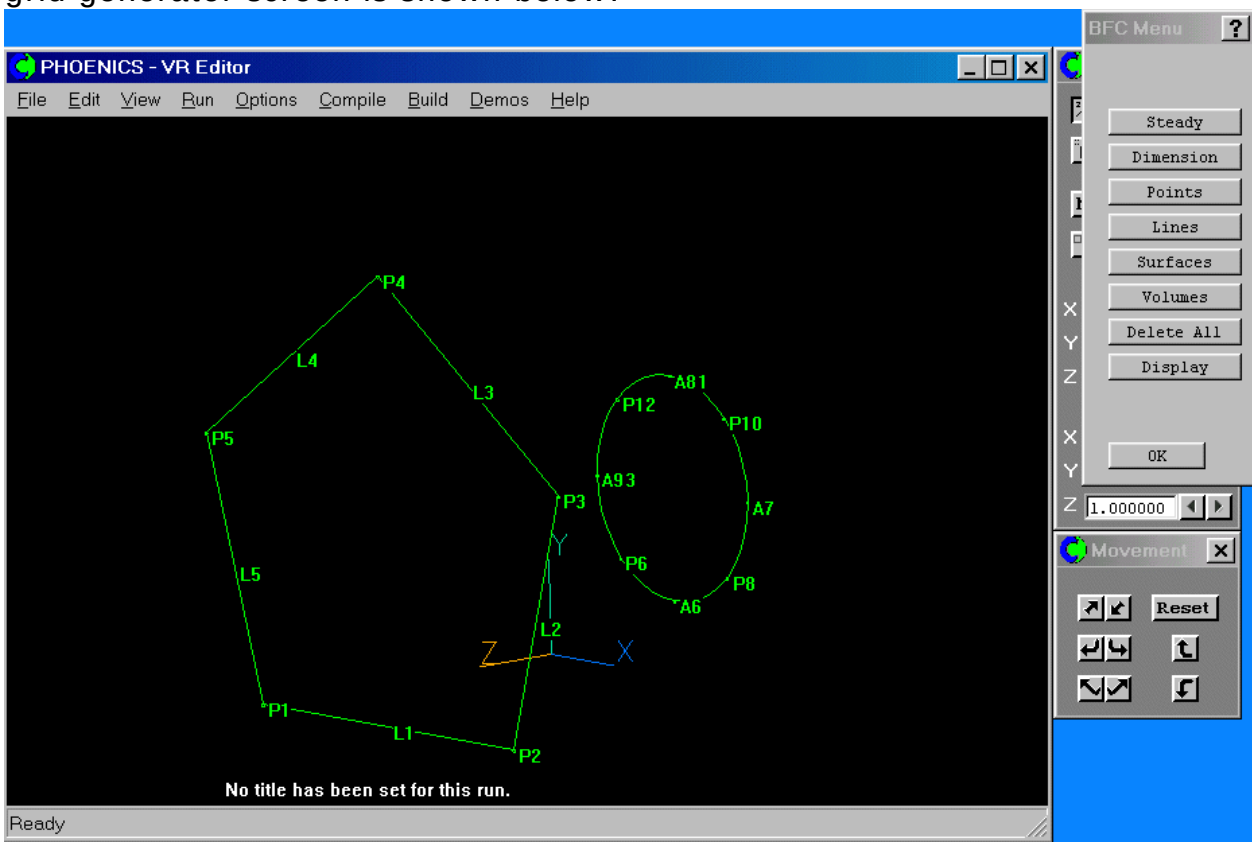
## Accessing PHOENICS-VR.

*From the system level:*

To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with Library case B111 - click on 'File' then on 'Load from libraries'. Type B111 into the Case Number box, then click OK.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

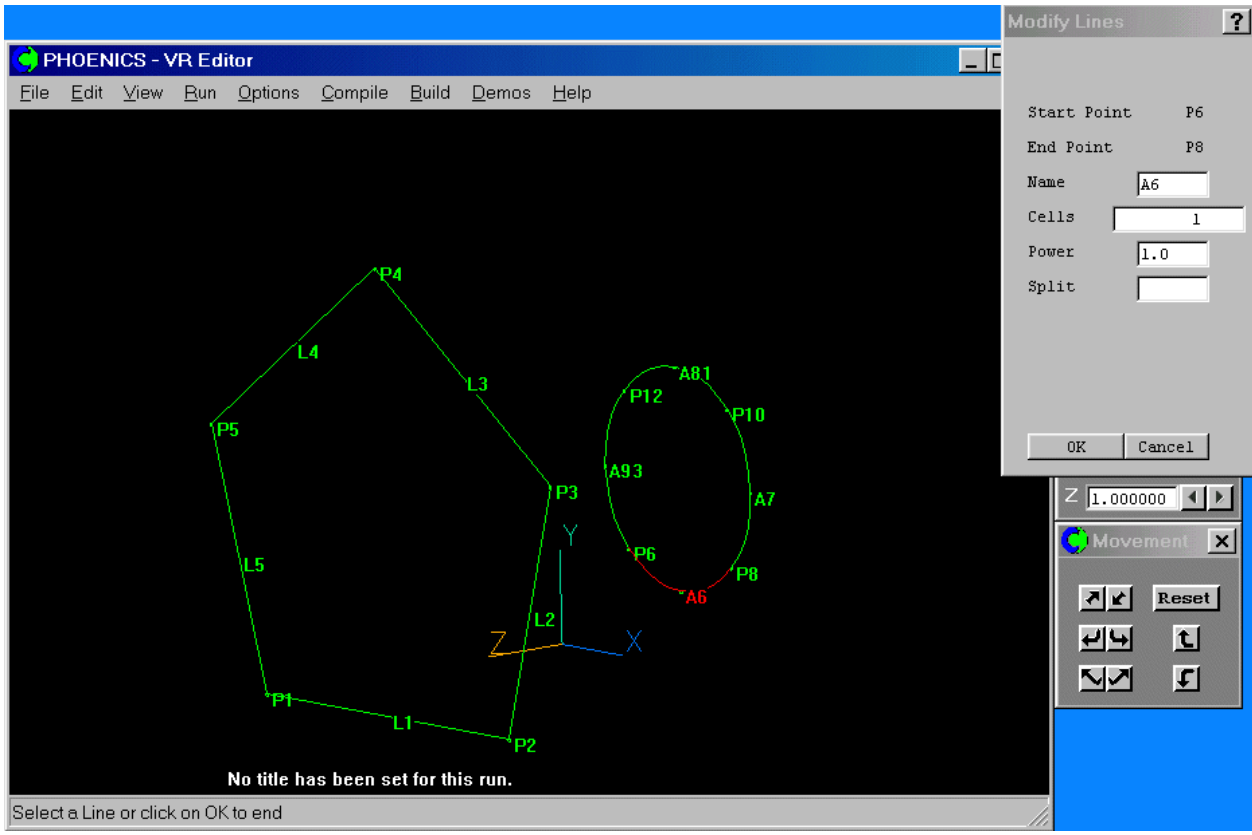
Activate the Grid Generator by clicking on the Grid mesh toggle on the hand-set , then clicking anywhere within the domain to bring up the BFC menu dialog. The grid generator screen is shown below:




**If you are continuing BFC Tutorial 1, you will join Tutorial 2 here.**
PHOENICS requires a structured mesh, and this has to be considered when setting the cell numbers.
This means that the number of cells on opposite sides of a frame must match.

The grid directions are known as I, J and K.
There is no link between these, and the Cartesian axes X Y & Z used for geometry definition.
Surfaces are made up of frames. A frame is a four-sided element, which may however be made up from more than four lines. The pentangle at z=1 will be one frame. Its corners will be points P1, P2, P3 and P5. The circle at z=0 will be another frame, with corners P6, P8, P10 and P12.
It is up to you to decide how to align the frame edges with the I/J/K grid directions.
The number of cells on each line can now be specified, bearing in mind the requirement for a structured mesh.
Click on the 'Lines' button, then 'Modify lines' . Point to line A6 (the bottom arc), and click.

When the line is selected, a sub-menu is displayed that allows the number and distribution of cells along the line to be modified. The selected line is shown in red and named in the top line of the table.
Enter the new number of cells - 10 for line A6.




When you click 'Ok' the line is re-defined and the dialog is erased.

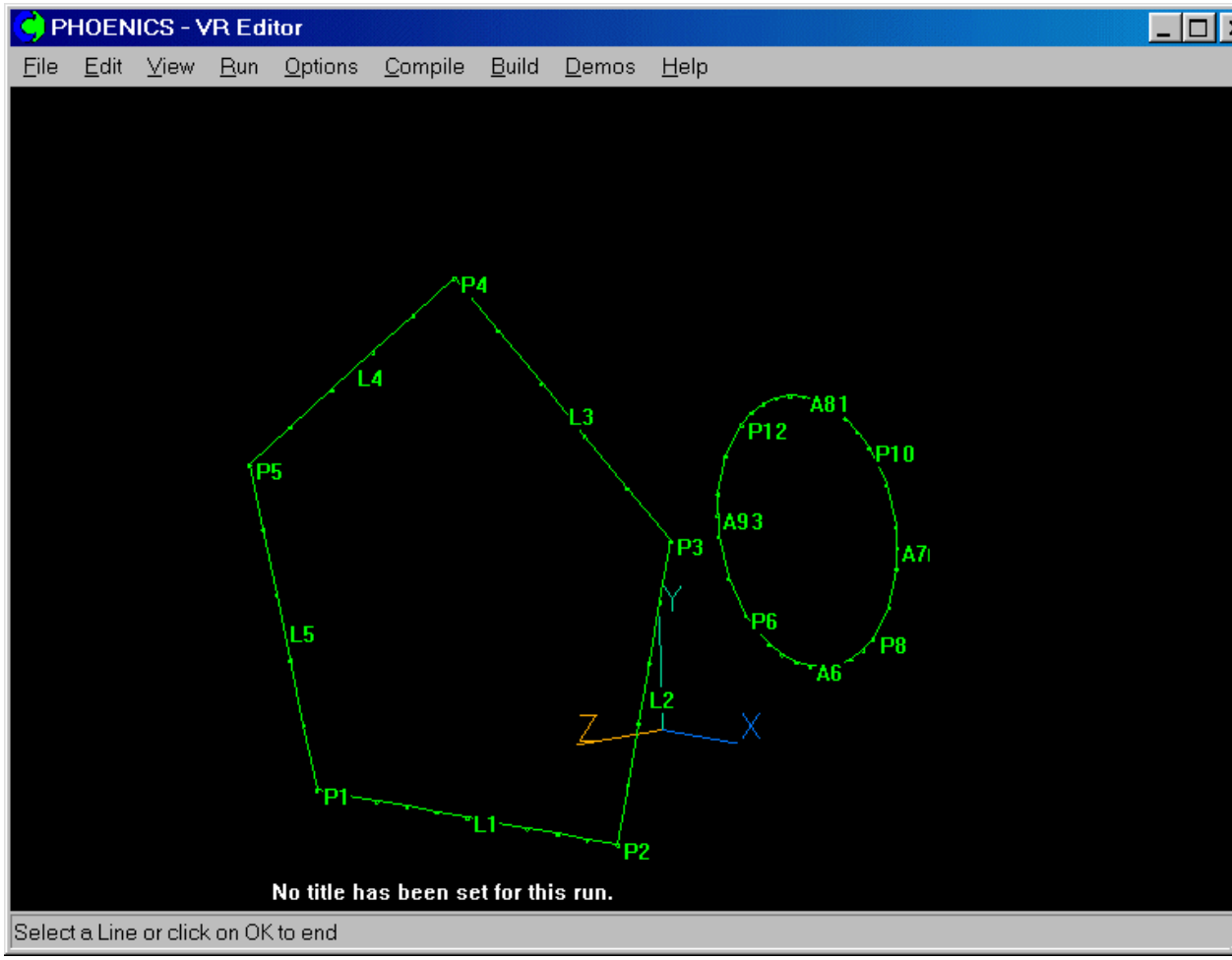
The line alteration menu is active until another function is selected, allowing the modification of many lines one after another.
At this stage in the mesh generation, we have to choose cell distributions on each line to maintain a structured mesh when the frames are defined. Therefore if we specified 10 cells on arc A6, we MUST specify 10 cells on arc A8,because these will become opposite sides of a frame.
These will become the 'I' cells.
Similarly, the specification of 5 cells on arc A7 will require the specification of 5 cells on arc A9.
These will become the J cells.
By clicking on lines A7, A8 and A9, set the number of cells as follows:

| Line name | Number of cells |
|---|---|
| A6 | 10 |
| A7 | 5 |
| A8 | 10 |
| A9 | 5 |

Therefore, on the z=0 plane, we have specified a 10 by 5 mesh. To maintain a structured mesh, we have to also specify a 10 by 5 mesh on the z=1 plane.
This requires the selection of the lines to modify, and furthermore, the mesh must be specified in a suitable orientation.
For this case, we will keep 10 cells in 'I' in line L1, and take 5 cells in lines L3 and L4 across the top of the duct.
By clicking on lines L1, L2, L3, L4 and L5, set the number of cells as follows:

| Line name | Number of cells |
|---|---|
| L1 | 10 |
| L2 | 5 |
| L3 | 5 |
| L4 | 5 |
| L5 | 5 |

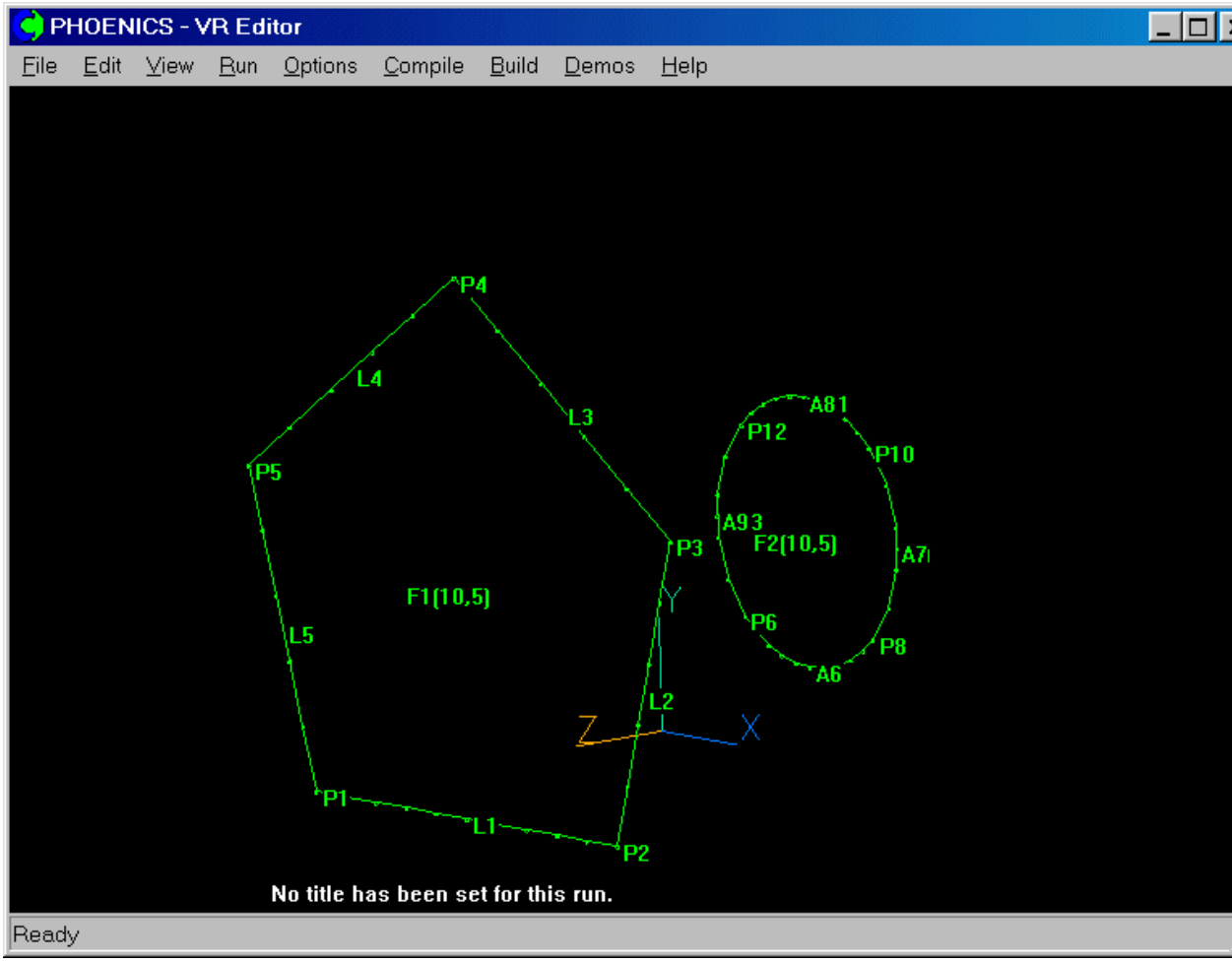
The screen should now look like this:




We can now generate the surfaces. Surfaces are composed of Frames. Frames are generated by using the 'Surfaces', 'New Frames' option. To generate a frame, each point around the frame edge must be selected in turn. The points must be joined by lines.
The direction of selection is immaterial, as long as a CORNER is selected first. Each valid selection will be indicated by the point and connecting lines changing colour to red.
We start by generating the frame at z=1 (the pentangle). Click on points P1, P2, P3, P4 and P5 in order, then on P1 again to close the frame.
When the first point is re-specified to close the frame, the point changes colour to light blue. This is to allow the specification of the remaining corner points ensuring correct mesh orientation.
The corner points are P1, P2, P3 and P5. Click on P2, P3 and P5. P4 is merely an intermediate point.
When the fourth corner point is selected, the frame is created.

A valid frame is shown by the name F1(n,m). (n,m) show the number of cells on opposing edges. The numbers in the brackets are positive when the frame is correct. If either number is negative, an error is present in the frame - usually the number of cells on opposing faces does not match. The negative number indicates the difference in cell numbers.

Now make the frame on z=0 (the circle). Click on P6, P8, P10, P12 and P6 again to close the frame.
Note, when only four lines make up a frame, the corner points are automatically known and the frame is generated without further input.




The frames for the duct have now been defined. The next tutorial shows how a mesh is fitted to the frames.
If you want to proceed directly to the next tutorial, click here.
To summarise, in this Tutorial you have learned how to:

- set the number of cells along a line; and

- make Frames.

We now suggest you practice each of these functions 'live' for yourself with the following exercises:

- Create points, join them with lines, and then make frames.
- Set the number of cells on opposing edges to make valid frames.

To leave the BFC Menu, click 'OK'. To save the work so far, click on 'File', 'Save as a case', make a new folder called 'BFC ' (e.g.), and save as 'BFC2'.
Note that at this point no mesh has been created, so the VR-Editor will still show the default cube!

Home Page

# WORKSHOP - BFC Grid Generation (Frame Matching)

## Introduction

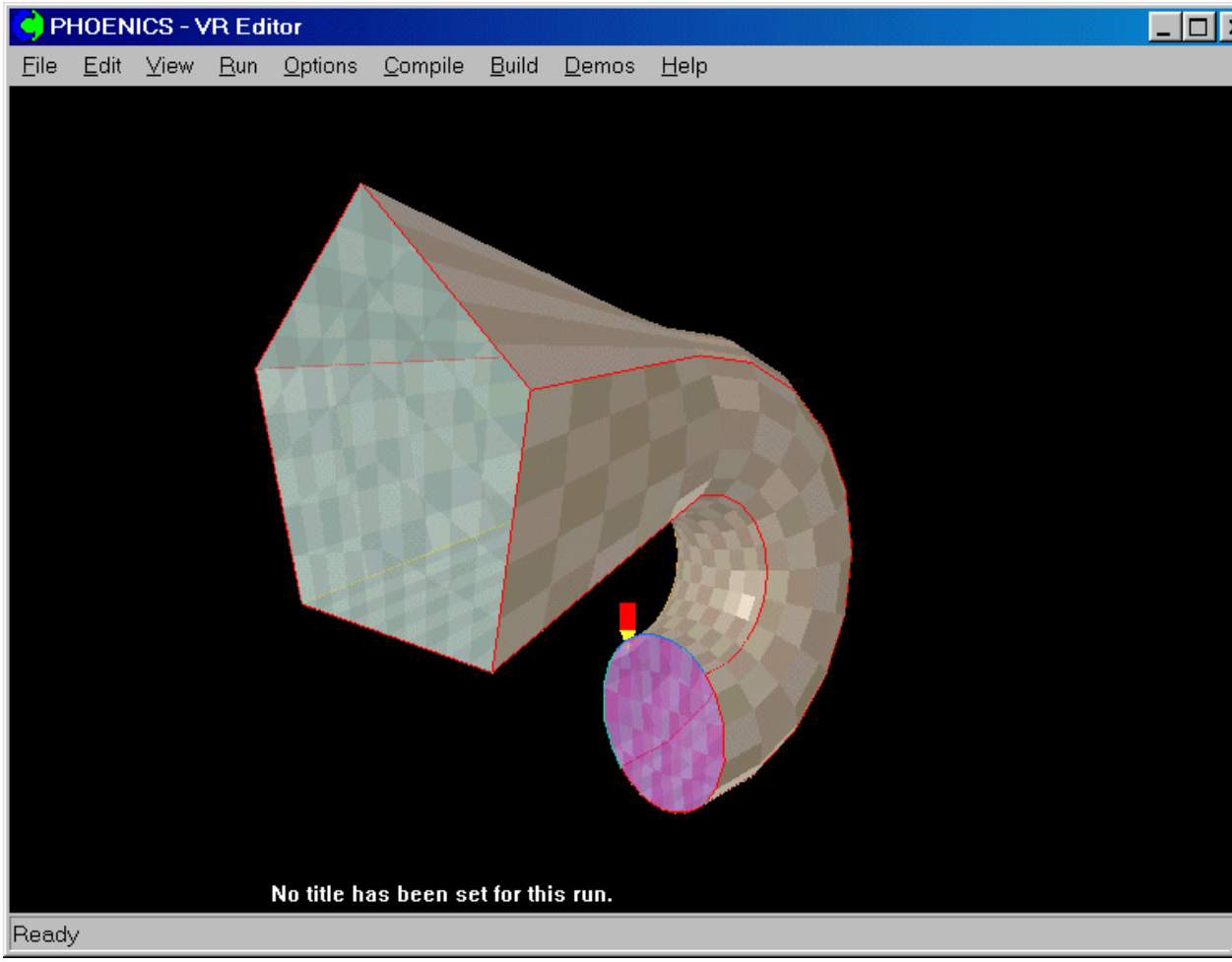
In this tutorial you will learn about:

- Setting the overall number of cells; and
- Matching Frames;

At the end of the tutorial you may like to confirm your understanding by the following exercises:

- Delete the matched frames, then change the number of cells to be used by modifying the lines.
- Reset the total number of cells. Recreate and rematch the frames.
- Add lines to join points P1-P6, P2-P8, P3-P10, P5-P12. Create and match new frames.

In BFC, as for Cartesian/polar co-ordinates, the geometry is created first, and the mesh is fitted later. This tutorial deals with the creation of the geometry for a duct, where the cross-section changes from a pentangle to a circle along the length.




The order of the data entity generation and modification is:

1. define suitable geometric POINTS;
2. connect points with LINES;
3. state the CELL distribution on each line;
4. group lines to make SURFACES, which are composed of FRAMES;
5. indicate which mesh points MATCH the frames; and
6. create VOLUMES by combining surfaces in various ways.

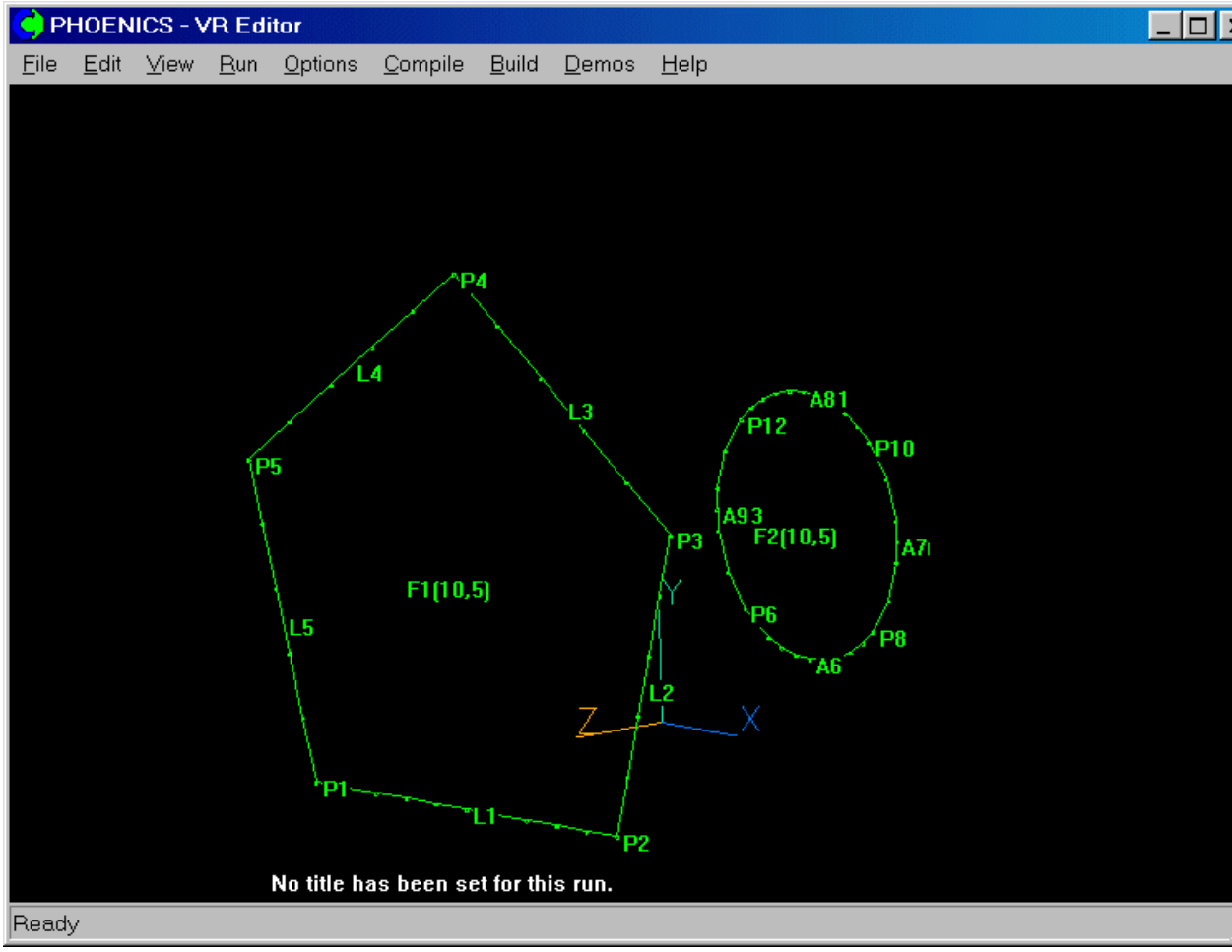
This tutorial covers stage 5.
In BFC Tutorial 1, we created the geometry for the pentangle and circle, and in BFC Tutorial 2 we created the required frames.
We now continue the process of meshing by fitting a grid into the frames.

## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with Library case B112 - click on 'File' then on 'Load from libraries'. Type B112 into the Case Number box, then click OK.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

Activate the Grid Generator by clicking on the Grid mesh toggle on the hand-set , then clicking anywhere within the domain to bring up the BFC menu dialog. The grid generator screen is shown below:



**If you are continuing BFC Tutorial 2, you will join Tutorial 3 here**.

The grid directions are known as I, J and K. There is no link between these, and the Cartesian axes X Y & Z used for geometry definition.

It is up to you to decide how to align the frame edges with the I/J/K grid directions, bearing in mind that both Cartesian AND grid co-ordinates must be right-handed.

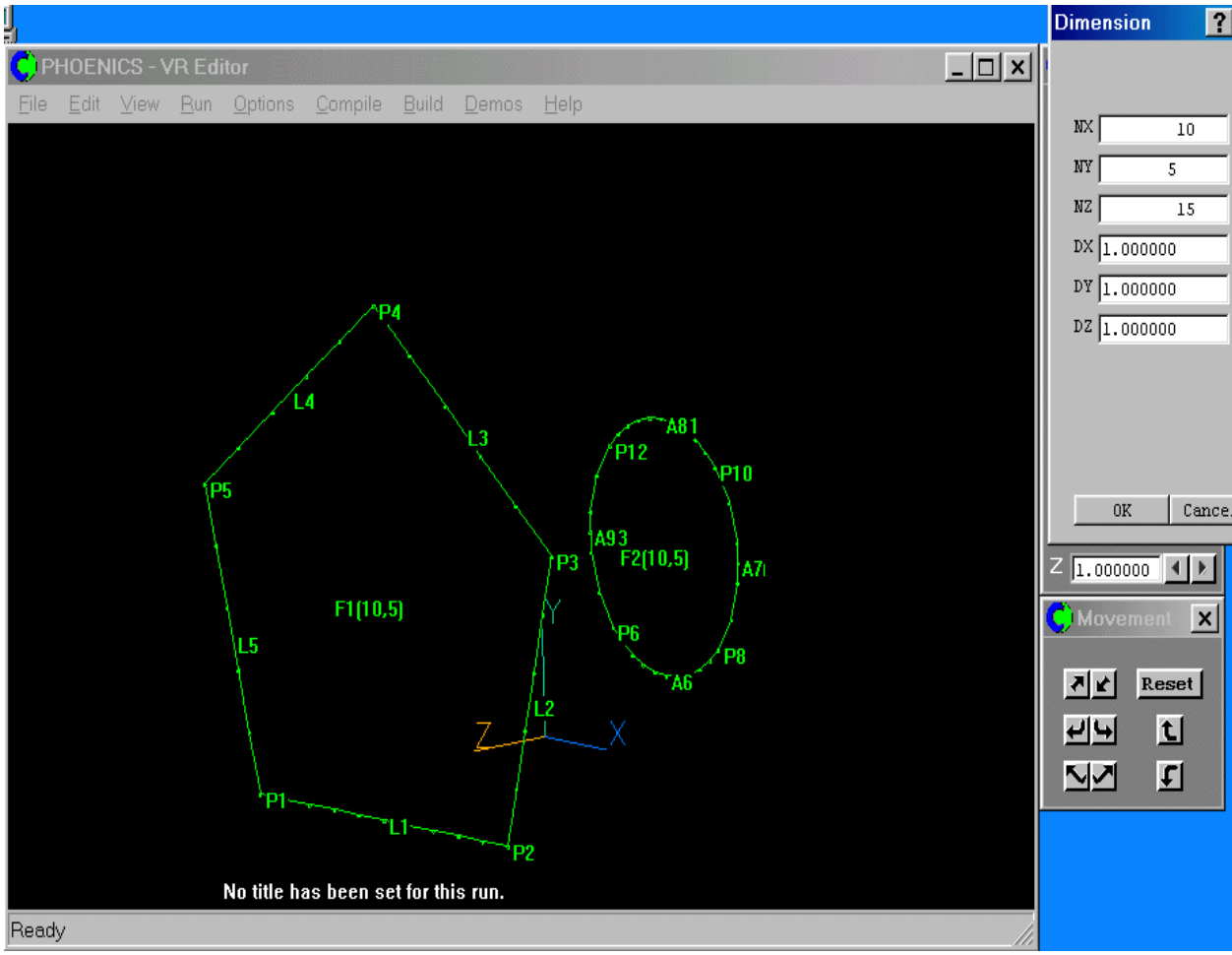
We have created two frames, each having 10*5 cells. The 10 cells are to be in the I direction, and the 5 in J. There will be 5 cells in the K direction, smoothly joining the two frames. The pentangle will be at the outlet, on plane K=16.

Once the duct becomes circular at frame F2 on plane K=11, it is to rotate through 180 degrees, centred on the X axis. This section will contain a further 10 cells in K, making 15 in all.

You must specify how many computational cells are present in each direction. For this case, it is 10*5*15.

In general, there will be many frames on each plane. The number of cells is the sum of the cells on the frame edges in each direction.

Click on the 'Dimension' button. A dialog appears which allows the selection and setting for each direction in turn. Set the number of cells in I (NX) to 10, in J (NY) to 5 and in K (NZ) to 15.

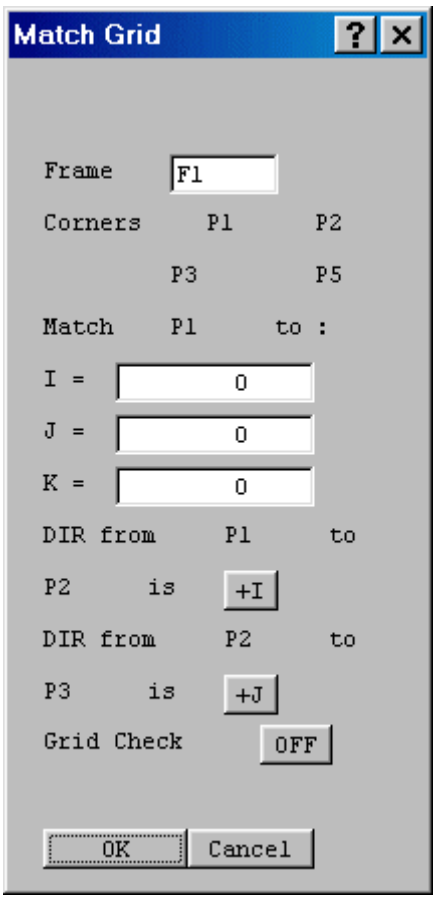



Initially, the cells are distributed evenly over a default cube. The cube size is set via the DX, DY and DZ buttons, and by default all three are '1.0'.

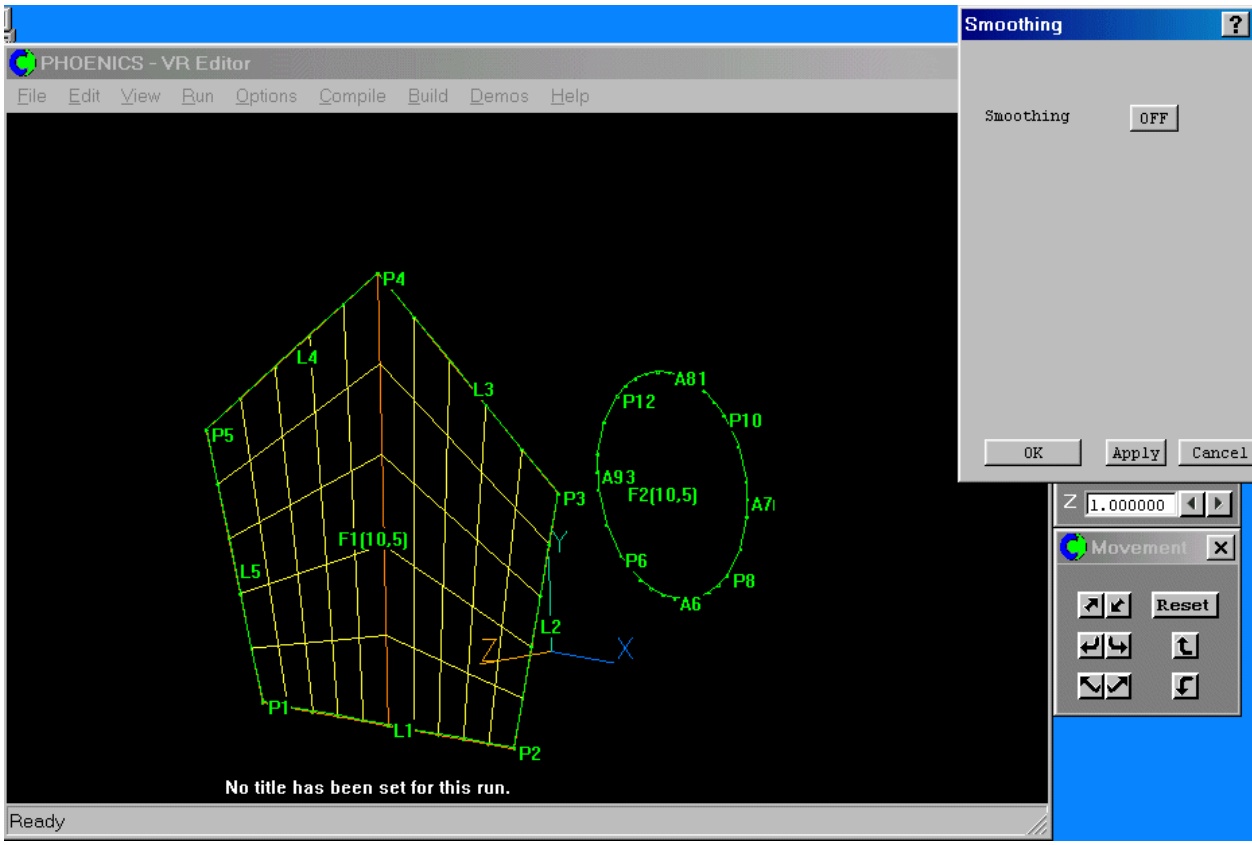
When all actions in this sub-menu are completed, click the 'Ok' button to return to the calling menu. The default mesh is set. It can be viewed by clicking 'Display', 'Mesh display', 'Outer mesh' , and turning the edge displays for East, North and High to 'On'. Having seen the default mesh, turn the edged display back to 'Off'. Click 'OK' to close dialogs until the BFC Menu.

We now have to match the FIRST corner of each frame to I, J and K locations. We must also specify the grid directions along the first two edges.

Click on 'Surfaces', then the 'match Grid' button, then click on the label of frame F1. The frame-matching dialog will appear:




Match point P1 to I = 1, J = 1 and K = 16, the last K plane.
The direction from P1 to P2 is along +I, and from P2 to P3 is along +J, so no change is needed.
The 'grid check' button switches on a visual orthogonality checker, which shades the cells from red (ok) to blue (bad).
We are not going to use the grid checking option, so click 'Ok' to end and apply the values specified in this panel.
Once matched, the mesh is shown on the frame, and the Smoothing dialog is displayed.



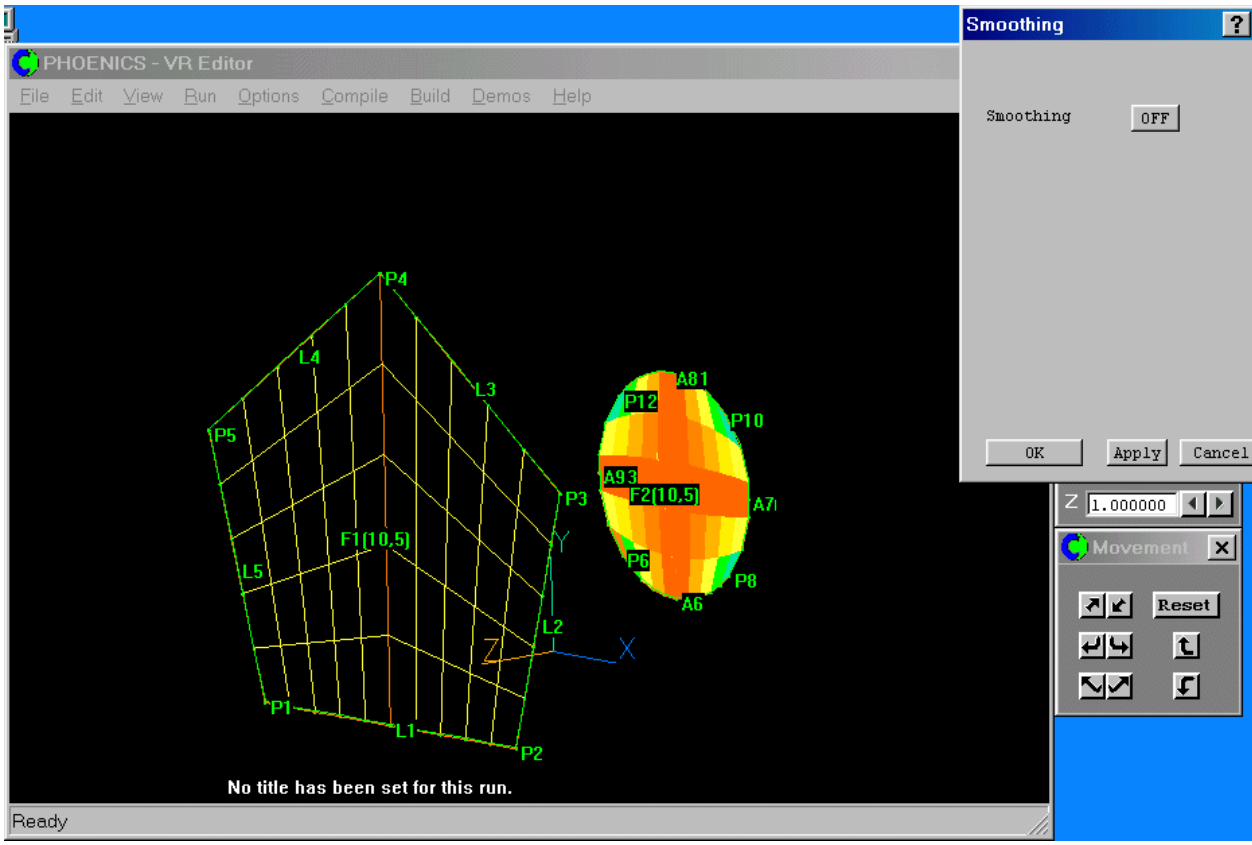

Note the red line drawn from point P4. This is a Region boundary, similar to the regions formed at object edges in the Cartesian system. Region boundaries are formed on all frame edges, and at intermediate edge points, such as P4.

We are not going to use the smoothing option for this frame, so click 'Ok' to end.

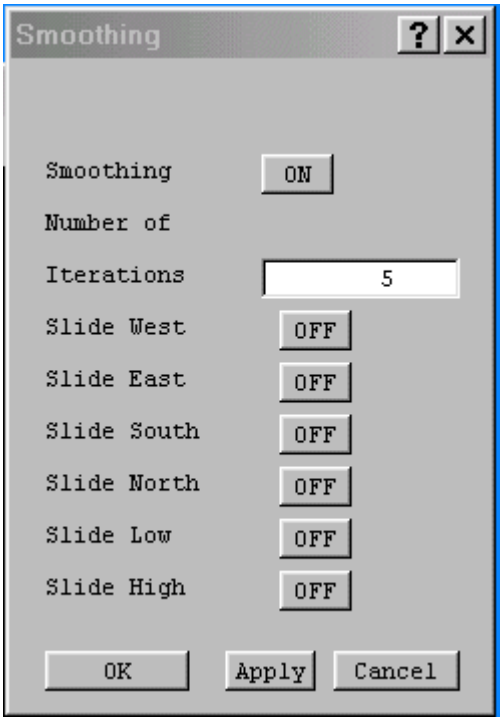
We now have to match frame 'F2'. Select it with the mouse. Match frame F2 to I=1, J=1 and K=11 (i.e. the end of the tenth cell in K).

The frame edges are curved, so turn the grid check on to see the mesh orthogonality.

Set the frame by pressing OK.

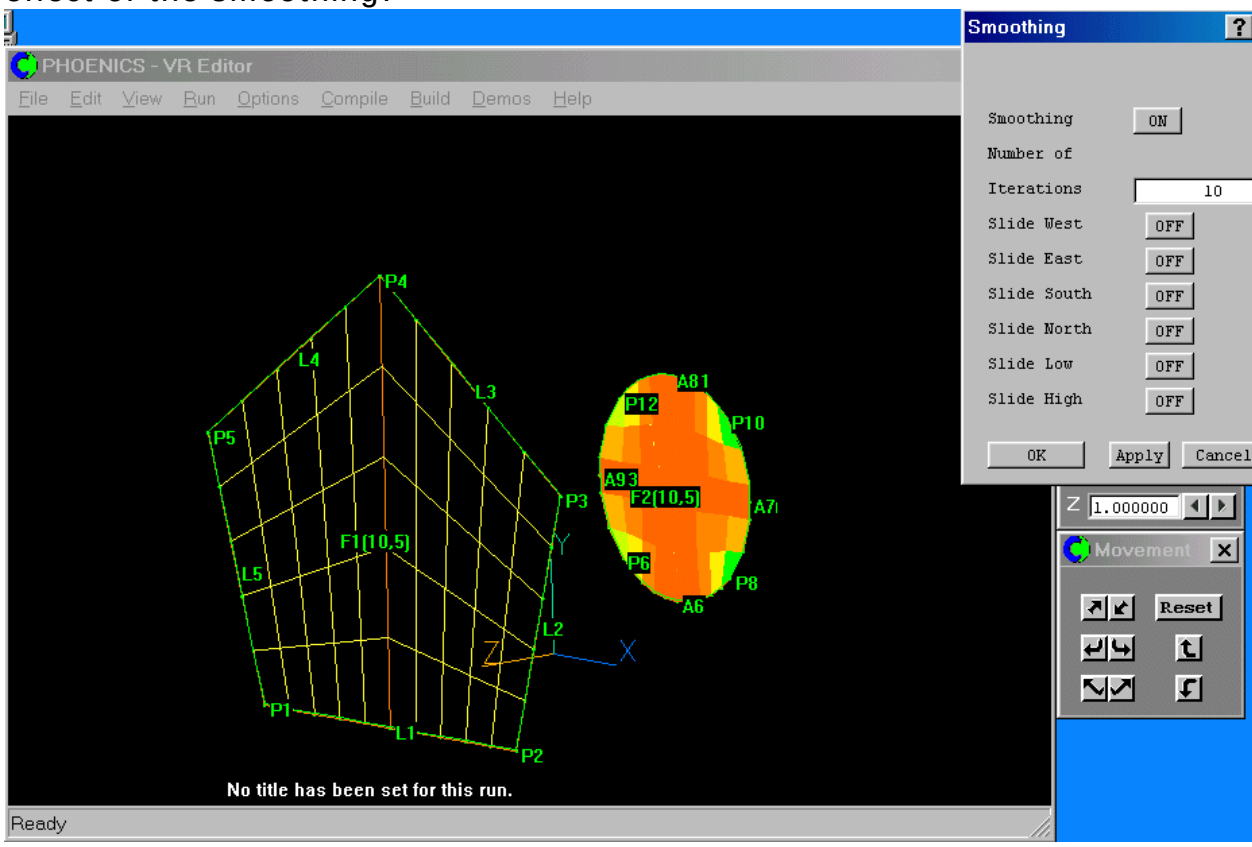



The mesh is quite poor - the 'corners' are blue/green. This could give problems with convergence. Turn the Smoothing 'On'. The dialog will change:

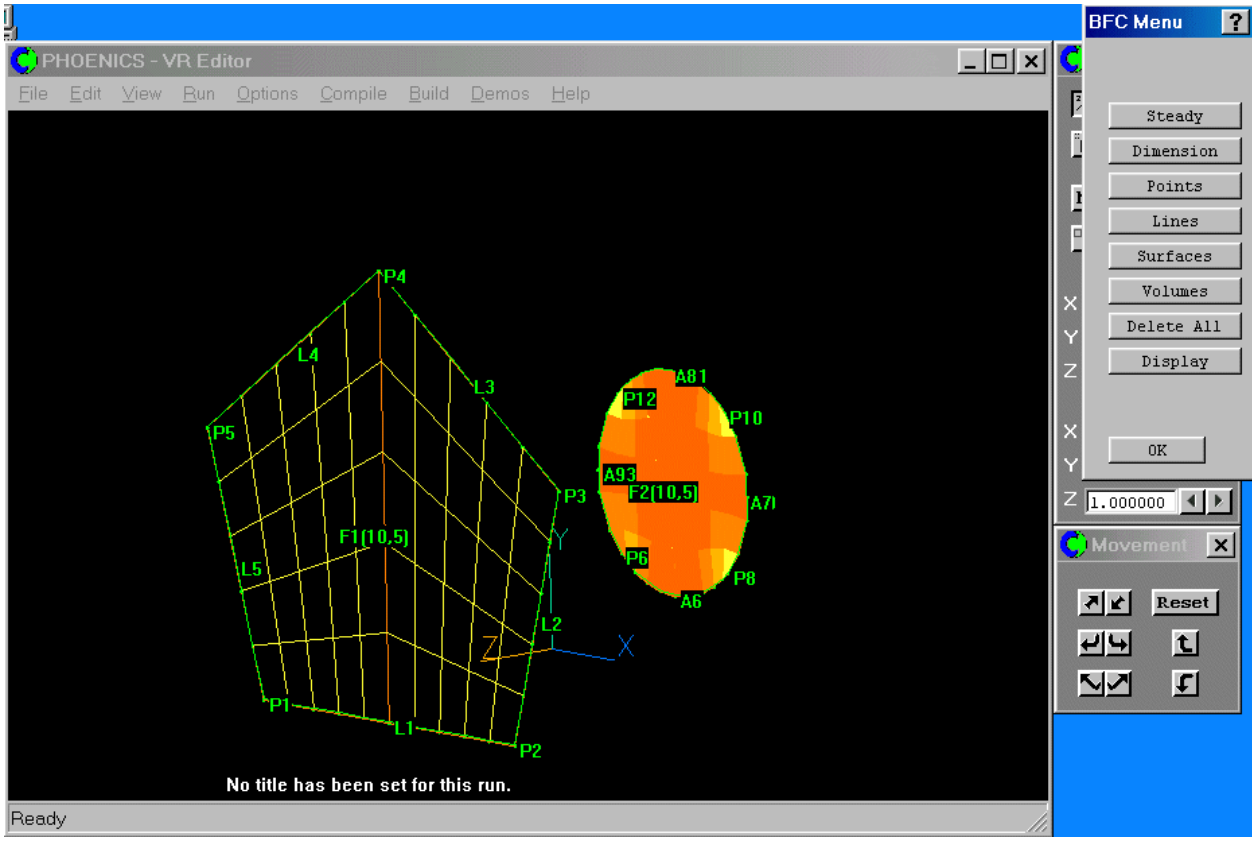



The 'Iterations' set to 5 activates a Laplace-like solver, which acts to smooth the mesh on this frame. Values

between 5 & 10 usually suffice. A value of 0 means that a transfinite (linear) mapping will be used (equivalent to Smoothing Off).
The Slide West/East/South/North/Low/High buttons allow the grid to 'slide' along the frame boundaries. If the frame boundary does not align with one of the Cartesian axes, the results may be unpredictable!
Set the number of iterations to 10. Click 'Apply' to see the effect of the smoothing:




The grid is now much better - the blue has gone. We can still improve it some more by pushing the cells towards the corners of the frame. Click on 'OK' to close the Smoothing dialog, then again to close the Grid matching dialog.
Click on the 'Lines' button, then on 'Modify line'. Select one of the arcs - A6. Set the power to S1.3. This will distribute the cells along the line symmetrically, using a power law of 1.3. Press OK to set the line.
Now do the rest of the arcs. The mesh is now much more orthogonal.




The frames for the duct have now been matched. The next tutorial shows how the geometry is completed.
If you want to proceed directly to the next tutorial, click here.
To summarise, in this Tutorial you have learned how to:

- Set the overall number of cells; and
- Match Frames

We now suggest you practice each of these functions 'live' for yourself with the following exercises:

- Delete the matched frames, then change the number of cells to be used by modifying the lines.
- Reset the total number of cells.
- Recreate and rematch the frames.
- Add lines to join points P1-P6, P2-P8, P3-P10, P5-P12. Create and match new frames.

To leave the BFC Menu, click 'OK'. To save the work so far, click on 'File', 'Save as a case', make a new folder called 'BFC ' (e.g.), and save as 'BFC3'.
Note that at this point the mesh has only been created on planes K=11 and K=16, so the VR-Editor will display a distorted default cube!

Home Page

# WORKSHOP - BFC Grid Generation (Copy / Transfer)

## Introduction

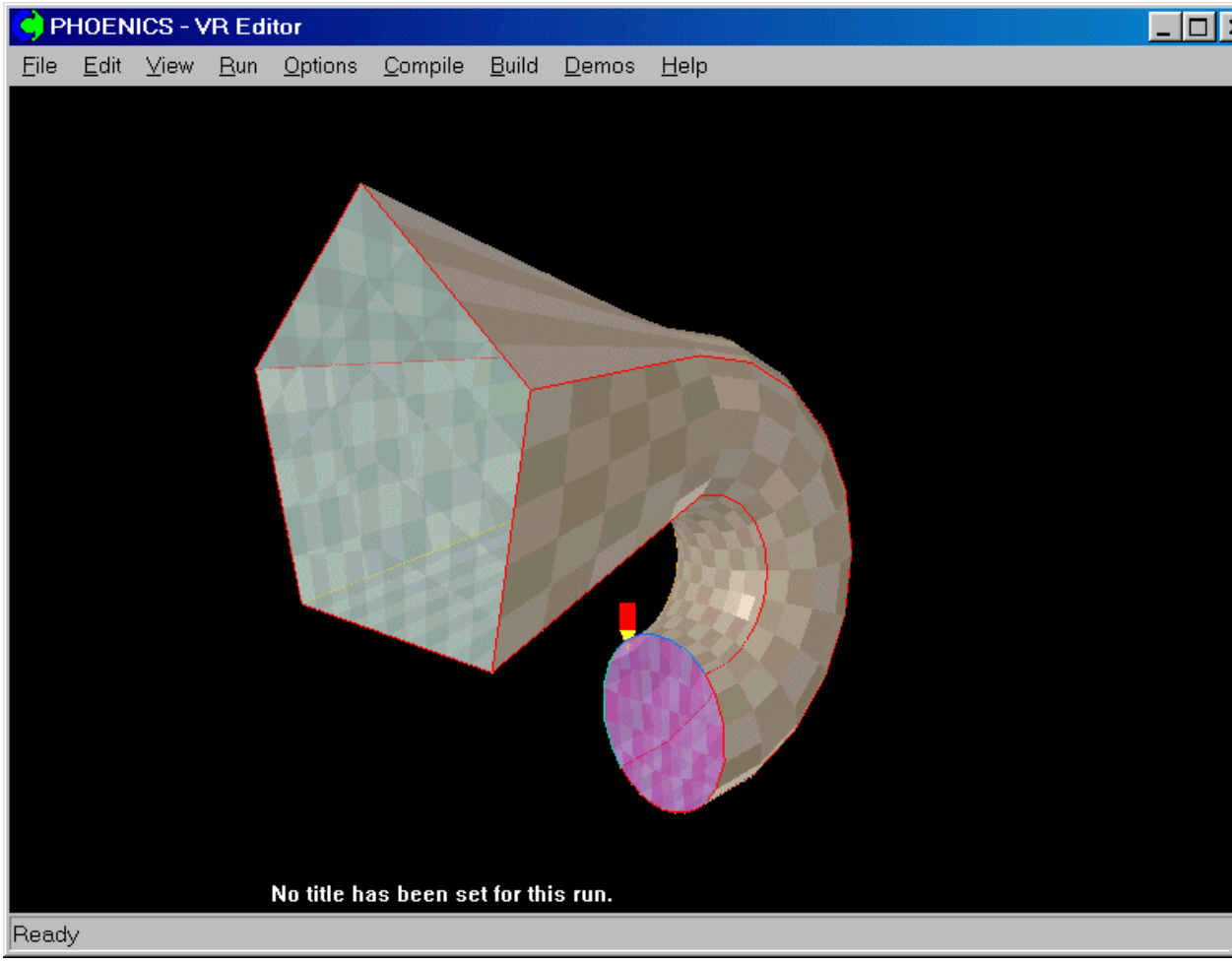
In this tutorial you will learn about:

- Creating volumes by transferring mesh from one plane to another; and
- Creating volumes by revolving mesh planes.

At the end of the tutorial you may like to confirm your understanding by the following exercises:

- Modify the 'Revolve' to rotate about a different axis, or to be an 'Extrude'.
- Modify the geometry by moving points and see how the effect transmits through the mesh.

In BFC, as for Cartesian/polar co-ordinates, the geometry is created first, and the mesh is fitted later. This tutorial deals with the creation of the geometry for a duct, where the cross-section changes from a pentangle to a circle along the length.




The order of the data entity generation and modification is:

1. define suitable geometric POINTS;
2. connect points with LINES;
3. state the CELL distribution on each line;
4. group lines to make SURFACES, which are composed of FRAMES;
5. indicate which mesh points MATCH the frames; and
6. create VOLUMES by combining surfaces in various ways.

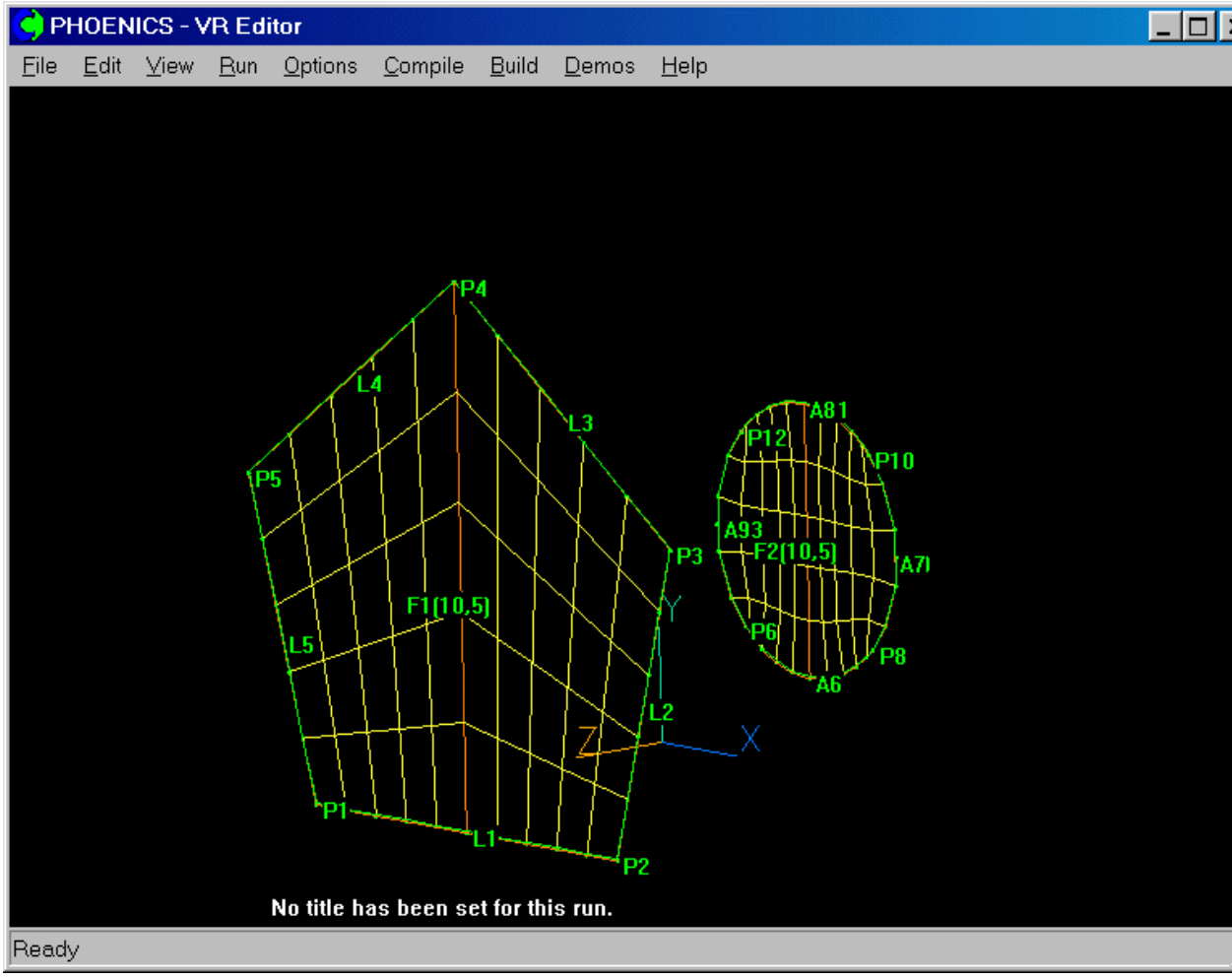
This tutorial covers stage 6.
In BFC Tutorial 1, we created the geometry for the pentangle and circle, and in BFC Tutorial 2 we created the required frames. In BFC Tutorial 3 we set the default grid, and matched the grid at planes K 11 and K 16 to the circle and pentangle respectively.
We now complete the process of meshing.

## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with Library case B113 - click on 'File' then on 'Load from libraries'. Type B113 into the Case Number box, then click OK.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

Activate the Grid Generator by clicking on the Grid mesh toggle on the hand-set , then clicking anywhere within the domain to bring up the BFC menu dialog. The grid generator screen is shown below:



**If you are continuing BFC Tutorial 3, you will join Tutorial 4 here**.

We have created two frames, each having 10*5 cells, and have matched them to the K=16 and K=11 planes.

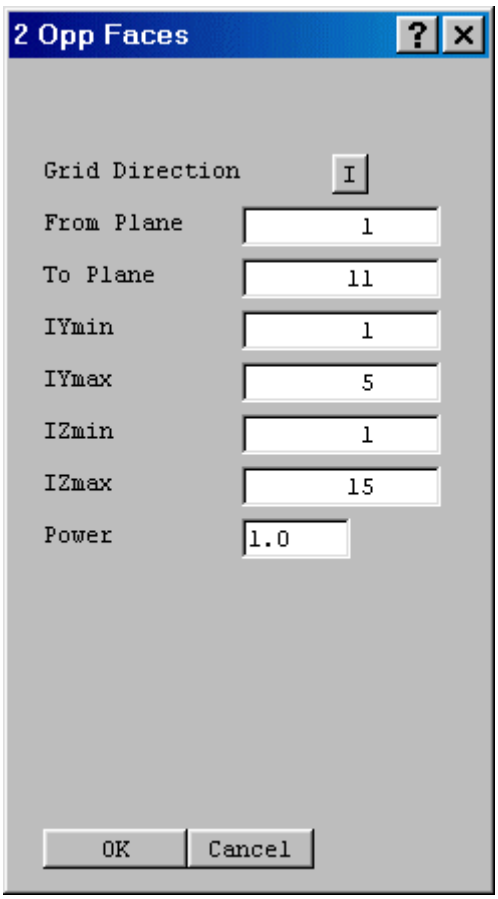
We now want to smoothly interpolate the mesh between K=11 and K=16, to create the intermediate planes.

Click the 'Volumes' button. The dialog for creating volumes has options for:

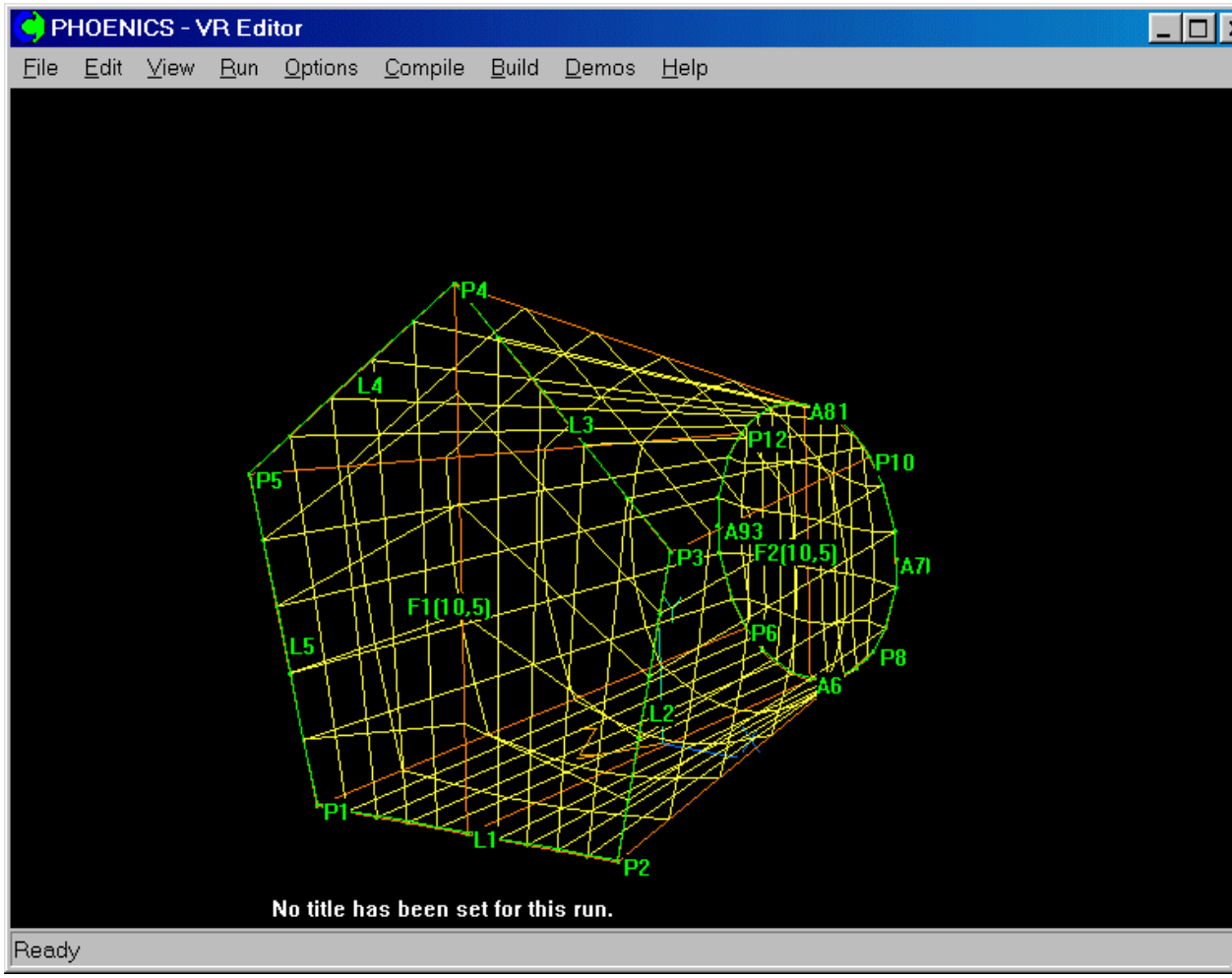
| |
|---|
| Extruding a surface |
| Revolving a surface about an axis |
| Joining two opposing surfaces |
| Filling in a volume enclosed by six surfaces |
| Modifying the list of commands |




To vary the mesh smoothly between frames 1 & 2, click on '2 Opp faces'.

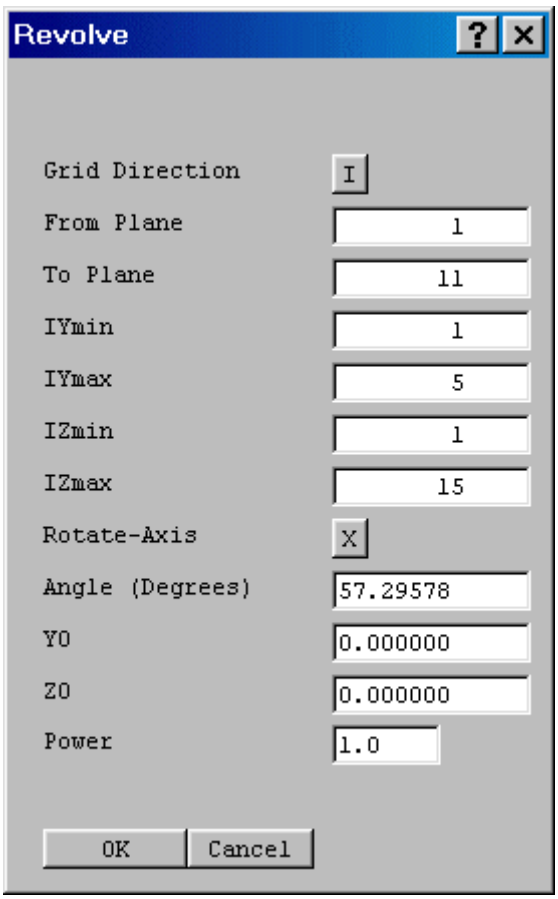



By default the transfer works on the I plane. The other planes are available by toggling. Toggle from I through J to K. Enter 11 in the 'from plane' box ( Frame F2 is matched to K 11, and F1 is matched to K 16). Click 'OK' to apply this transfer.

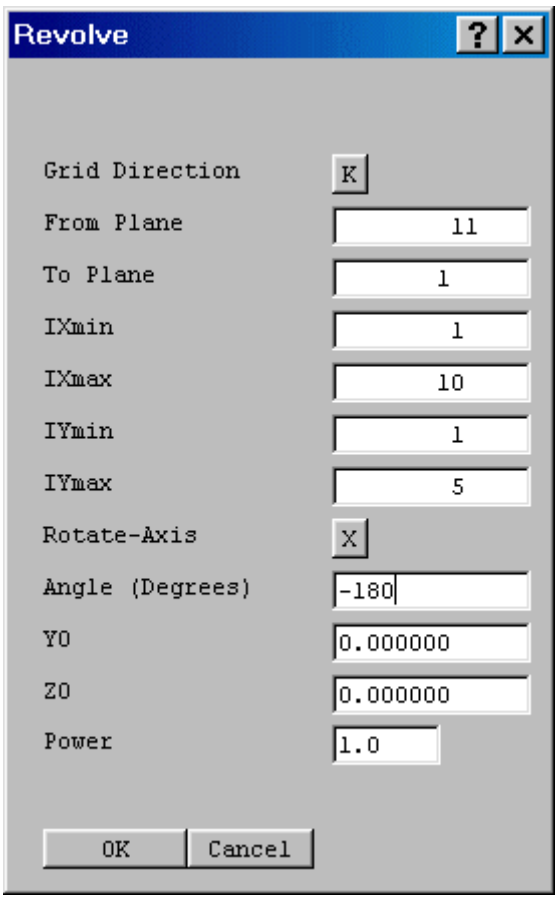



Once the duct becomes circular at frame F2, it is to rotate through 180 degrees, centred on the X axis. This section will contain a further 10 cells in K.

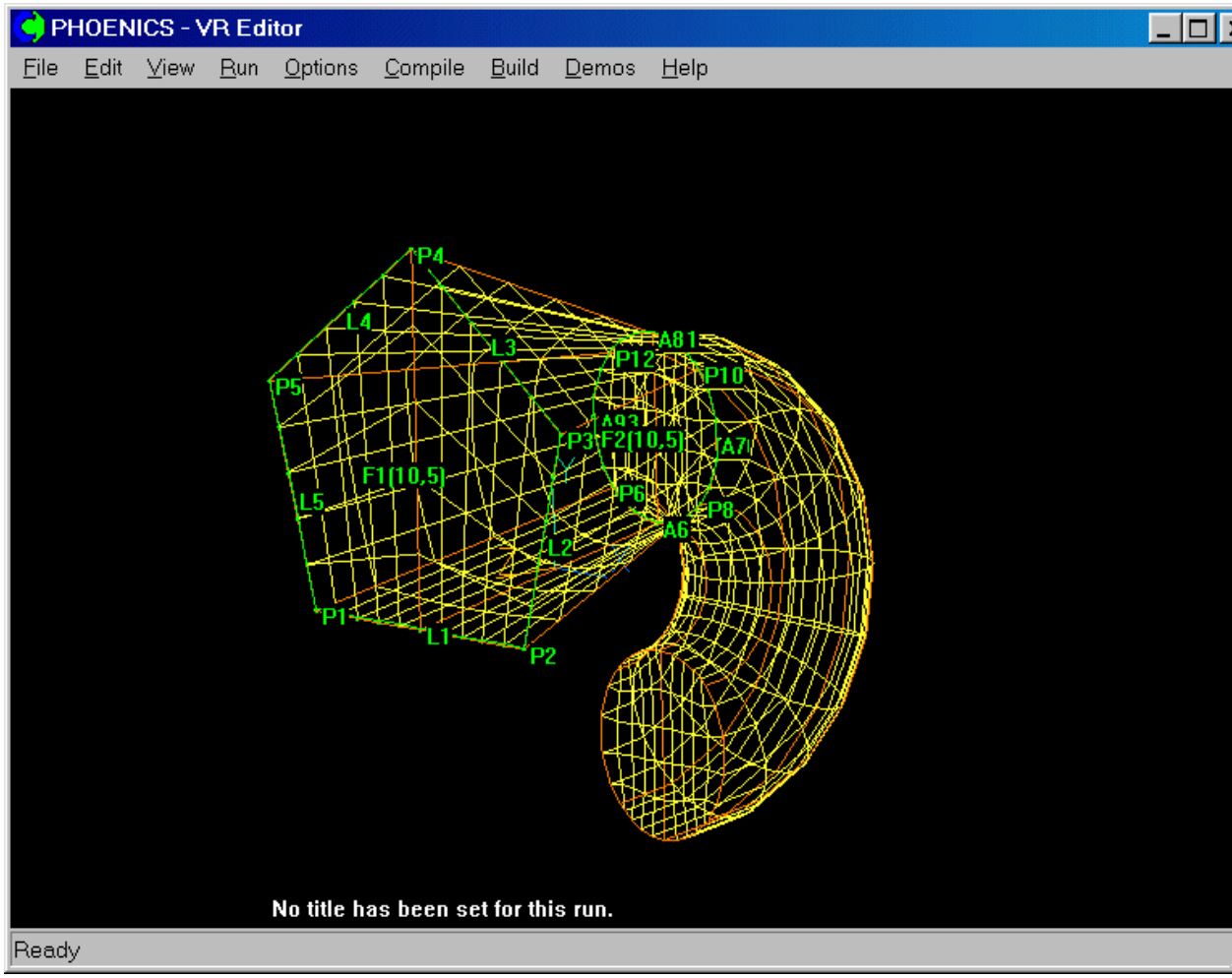
Click on 'Revolve'. The Revolve dialog will appear:



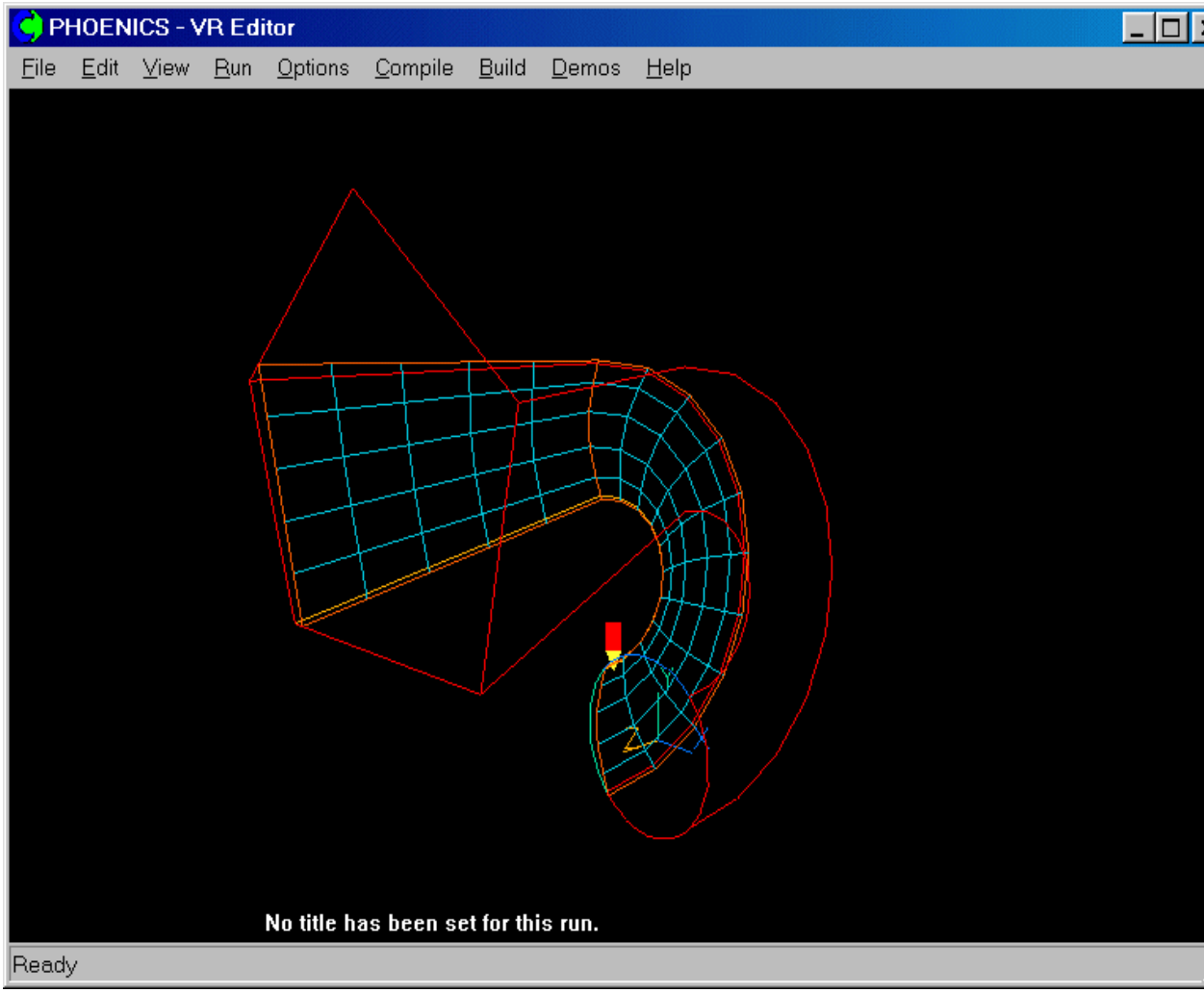

By default the I plane is revolved about the X axis by 1 radian (57 degrees). The other planes are available by toggling. Toggle through J to the K planes. Note: the default limits are the entire domain.
Enter 11 in the 'from plane' box (frame F2), and 1 in the 'to plane' box.
Select the axis to rotate about as X and set the angle of rotation to -180 degrees.
Specify the Y & Z co-ordinates of the rotation axis as (0.0, 0.0).




Click 'OK' twice to apply the modifications and return to the BFC Menu. Click on 'Reset' on the Movement control panel then on 'Fit to window' to scale the image to fit the screen:




Now that the grid is complete, click 'OK' to get back to the VR-Editor.




If you want to proceed directly to the last tutorial in the sequence tutorial, which deals with setting the flow boundary conditions, click here.
To summarise, in this Tutorial you have learned how to:

- Create a volume from two opposing faces; and
- Crete a volume by revolving a surface.

We now suggest you practice each of these functions 'live' for yourself with the following exercises:

- Modify the Revolve to rotate about a different axis, or to be an 'Extrude'.
- Modify the geometry by moving points and see how the effect transmits through the mesh.

To save the work so far, click on 'File', 'Save as a case', make a new folder called 'BFC ' (e.g.) and save as 'BFC4'.

Home Page

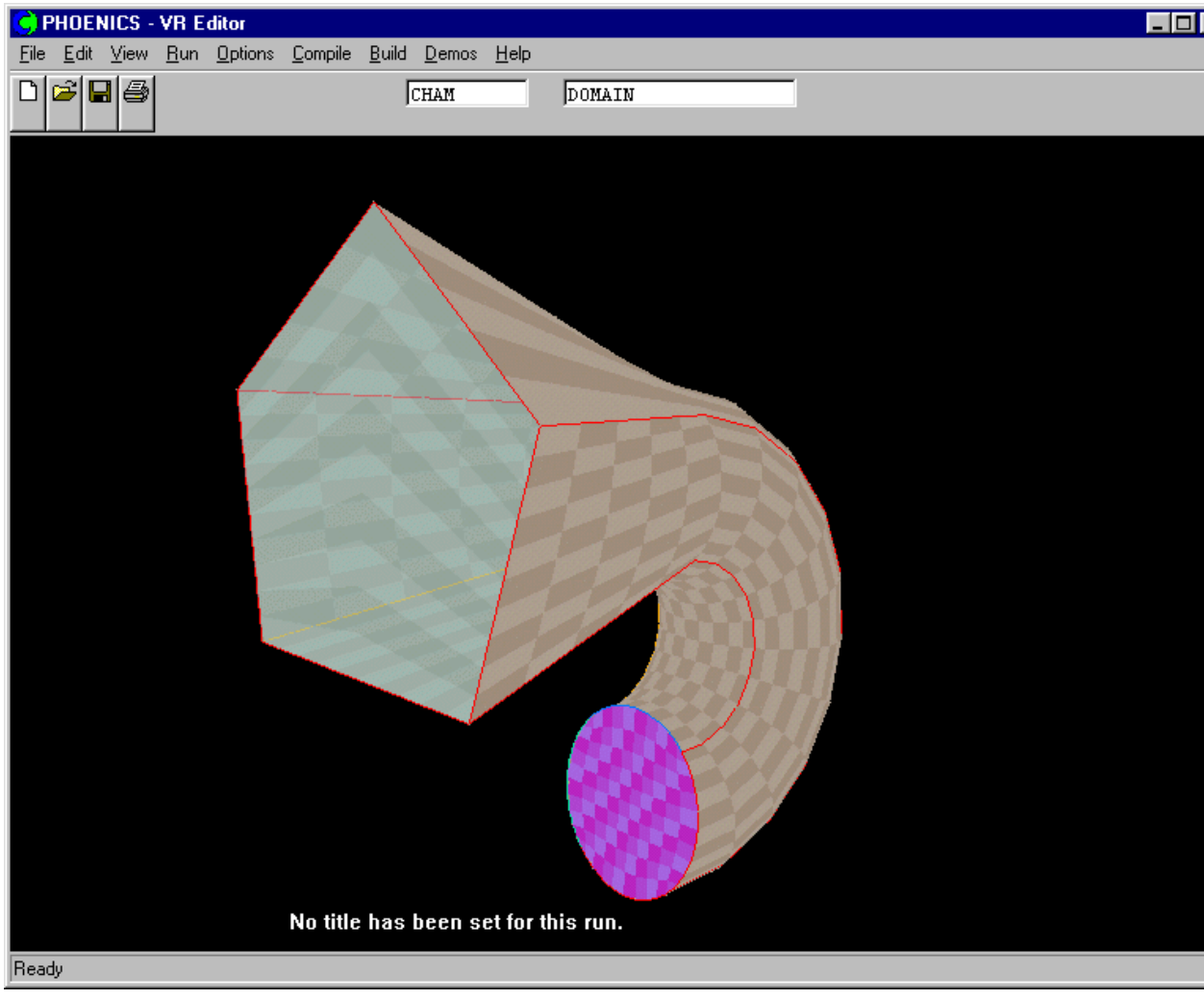
# WORKSHOP - BFC Grid Generation (Boundary Conditions)

## Introduction

In this tutorial you will learn about:

- Activating solution of variables;
- Setting boundary conditions; and
- Visualising the solution.

This tutorial deals with the creation of the geometry for a duct, where the cross-section changes from a pentangle to a circle along the length.



BFC Tutorials 1 - 5 dealt with the creation of the geometry and mesh. This tutorial activates solution of the required flow variables, and sets boundary conditions. The final outcome of this tutorial is available as Library case B115.

## Accessing PHOENICS-VR.

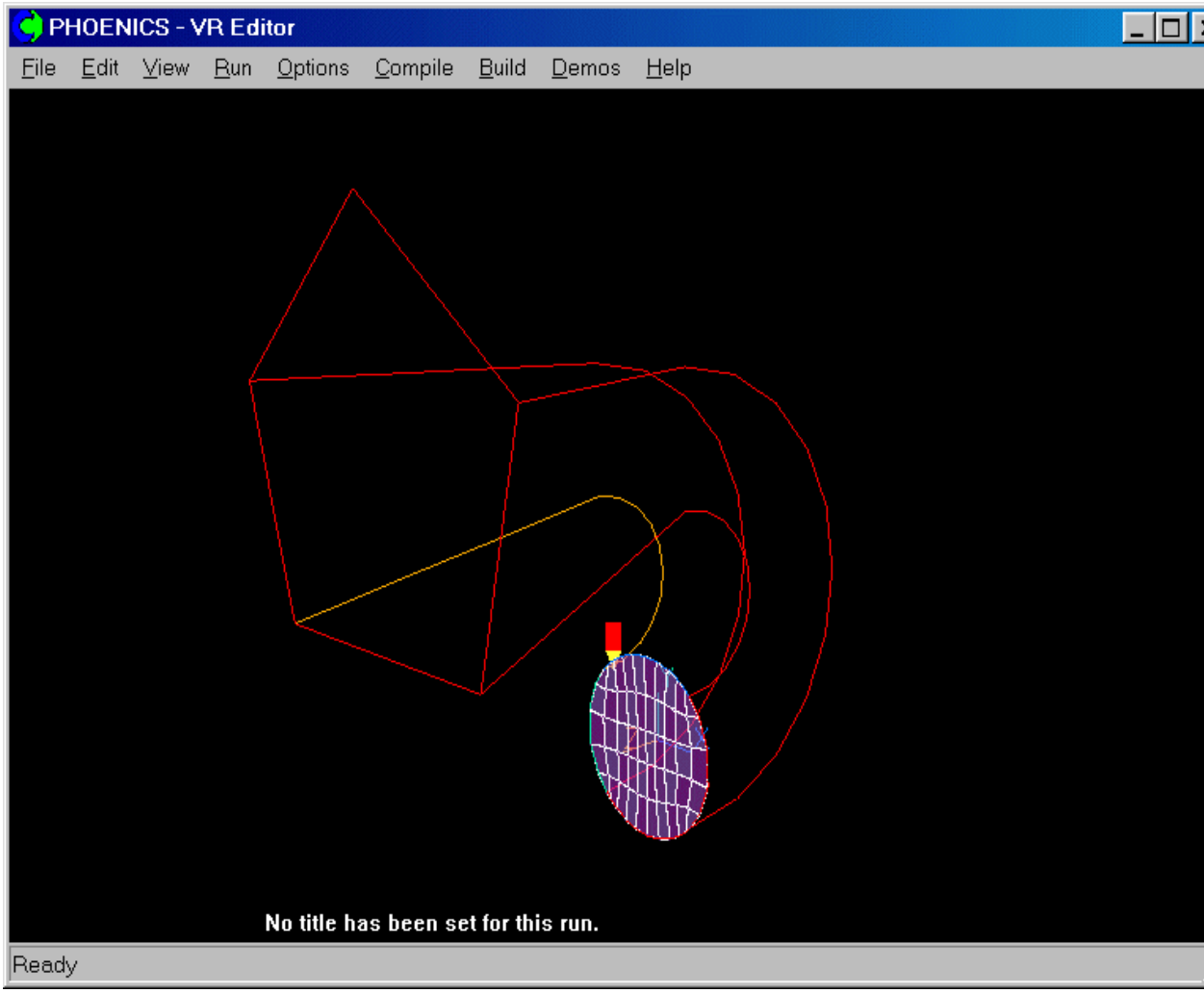
*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with Library case B114 - click on 'File' then on 'Load from libraries'. Type B114 into the Case Number box, then click OK.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

**If you are continuing BFC Tutorial 4, you will join Tutorial 5 here**.
**Activate solution of variables.**
Click on Menu / Models. Click on Equation formulation - Elliptic staggered. Select Elliptic GCV and click OK. This selects the GCV Collocated solver, which is very good at dealing with non - orthogonal (distorted) grids.
Activate solution of velocities and pressure. Select the KECHEN turbulence model.
**Set numerical controls**
Click on Numerics, and enter 100 in the 'Total number of iterations' box.
Exit the Main menu by clicking Top Menu, OK.
All the domain-related settings have been made - now set the flow boundary conditions.
**Create the inlet at K=0**
Click on New object. Set the object size as:

| | |
|---|---|
| ISize | 10 |
| JSize | 5 |
| KSize | 0 |

Note that in BFC, the object sizes are set as numbers of cells in each co-ordinate direction. Planar objects have zero thickness in one direction. Click on Type, and select INLET. Click on ATTRIBUTES, and set the Z direction velocity to -1.0. Note that the velocities are specified in the Cartesian co-ordinate system. Click OK to close the Attributes dialog, and OK to close the Object dialog.
The screen should look like this:




**Create the outlet at K=16**

Click on New Object. Set the size and position as follows:

| | |
|---|---|
| IPos | 0 |
| JPos | 0 |
| KPos | 15 |
| ISize | 10 |
| JSize | 5 |
| KSize | 0 |

Note that in BFC, the object positions are set as cell corners in each co-ordinate direction. The numbering starts at zero. Click on Type, and select OUTLET. Click on Attributes, and switch 'External turbulence' to User-set. We need to do this as the exit cuts a vortex - leaving the 'External turbulence' at the default 'In-cell' leads to excessive turbulence levels in the diffuser section. Click OK twice to close the Object dialog.

**Create four plates**

We now need four plates to provide friction conditions on the insides of the duct.

Create the first plate at J=5 (the last cell corner in J). Click on New Object.

Set the size and position as:

| | |
|---|---|
| IPos | 0 |
| JPos | 5 |
| KPos | 0 |
| ISize | 10 |
| JSize | 0 |
| KSize | 15 |

Click on Type, and select PLATE. Click OK to close the Object dialog.

Make the second plate at J=0 by duplication. Click the Duplicate object or group button.

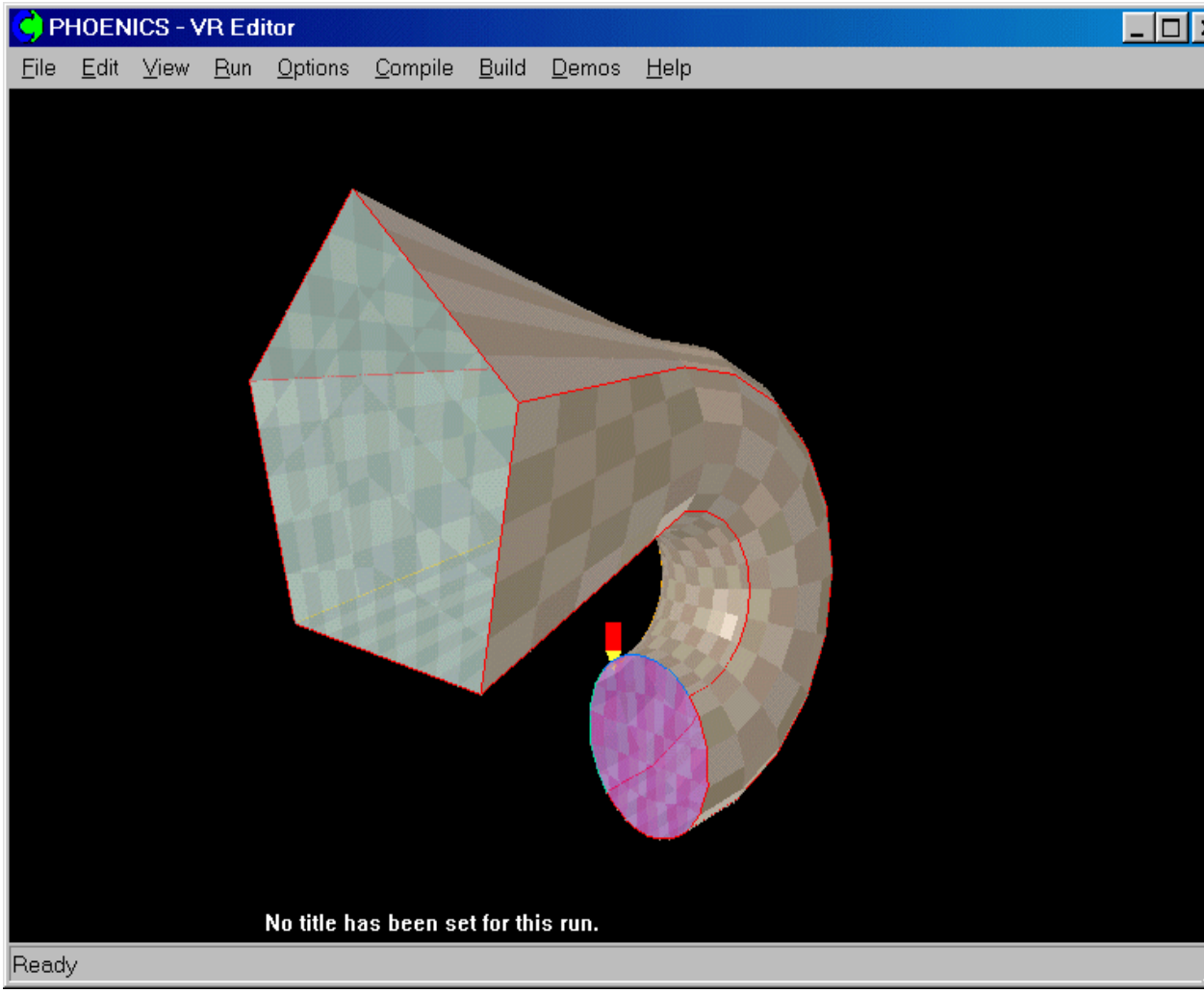
Create the third plate at I=10 (the last cell corner in I). Click on New Object. Set the size and position as:

| | |
|---|---|
| IPos | 10 |
| JPos | 0 |
| KPos | 0 |
| ISize | 0 |
| JSize | 5 |
| KSize | 15 |

Click on Type, and select PLATE. Click OK to close the Object dialog.

Make the last plate at I=0 by duplication. Click the Duplicate object or group button.

The final geometry should look like this:




The plates are hiding the inside of the duct, so we can hide them. Right-click on the Hide and show toggle, and set the entry for PLATE to 'Hide all'. Click OK. The plates will all be hidden from view, but will still remain active in the solution.

**Set monitor location**

The last action before running the Earth solver is to place the monitoring point (probe) in a sensible location.
Click on the image background, to ensure that no object is active. In this mode, the Position buttons on the hand-set move the probe. Use these to place the probe at (5,3,10).

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

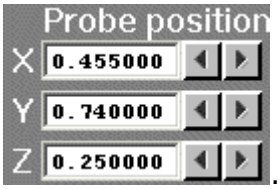
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

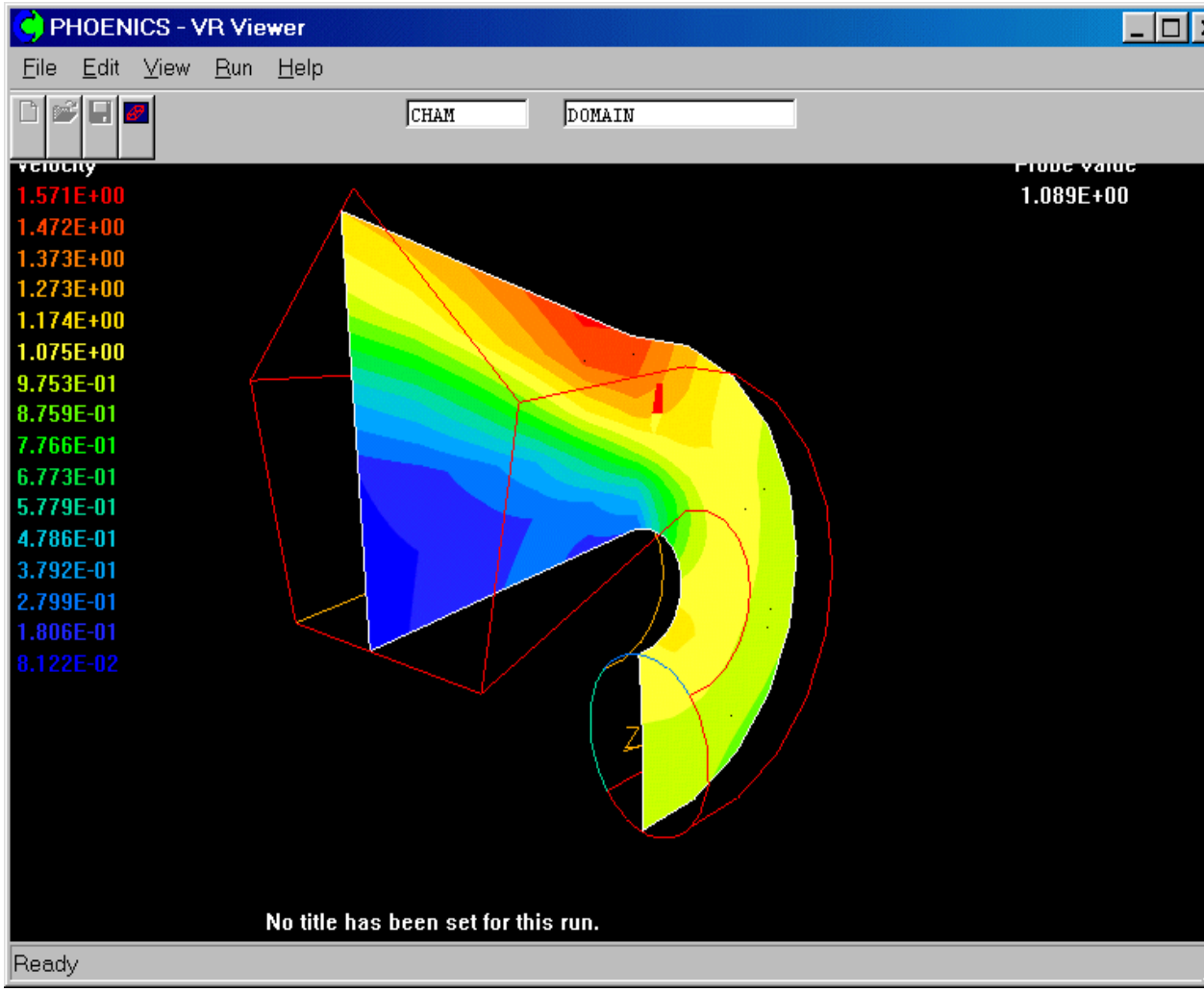
To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons






## Saving the results.

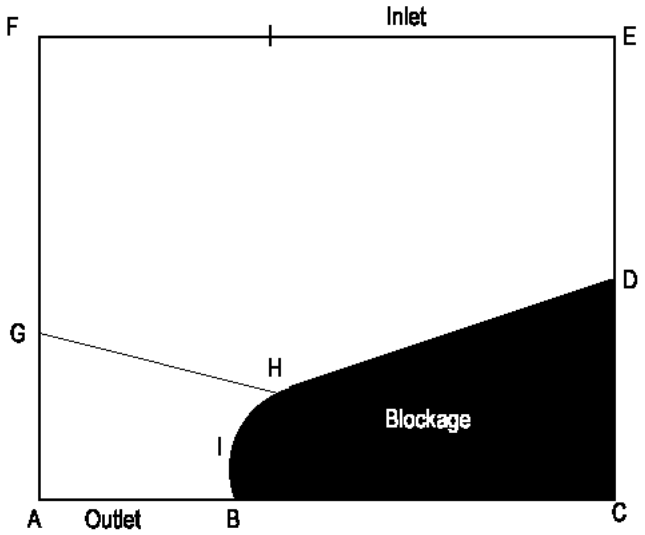
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'BFC ' (e.g.) and save as 'CASE5' (e.g.).

Home Page

# WORKSHOP - BFC Grid Generation

## Introduction

This workshop deals with the creation of a simple BFC grid, sets the flow boundary conditions, and visualises the solution. The case considered is the flow past a non-rectangular blockage, shown here:




All the lines are straight, except line B-I-H, which is an arc.
Three frames will be required. Their corner points will be:
Frame F1 - A, B, H, G
Frame F2 - B, C, D, H
Frame F3 - G, D, E, F
The line G-H is only required for the definition of frame F3, otherwise there would be no connection between G and D.
The co-ordinates of the points can be taken to be:

| Point | Xpos | Ypos | Zpos |
|---|---|---|---|
| A | 0.0 | 0.0 | 0.0 |
| B | 0.4 | 0.0 | 0.0 |
| C | 1.0 | 0.0 | 0.0 |

| | | | |
|---|---|---|---|
| D | 1.0 | 0.4 | 0.0 |
| E | 1.0 | 0.8 | 0.0 |
| F | 0.0 | 0.8 | 0.0 |
| G | 0.0 | 0.25 | 0.0 |
| H | 0.45 | 0.25 | 0.0 |
| I | 0.35 | 0.125 | 0.0 |

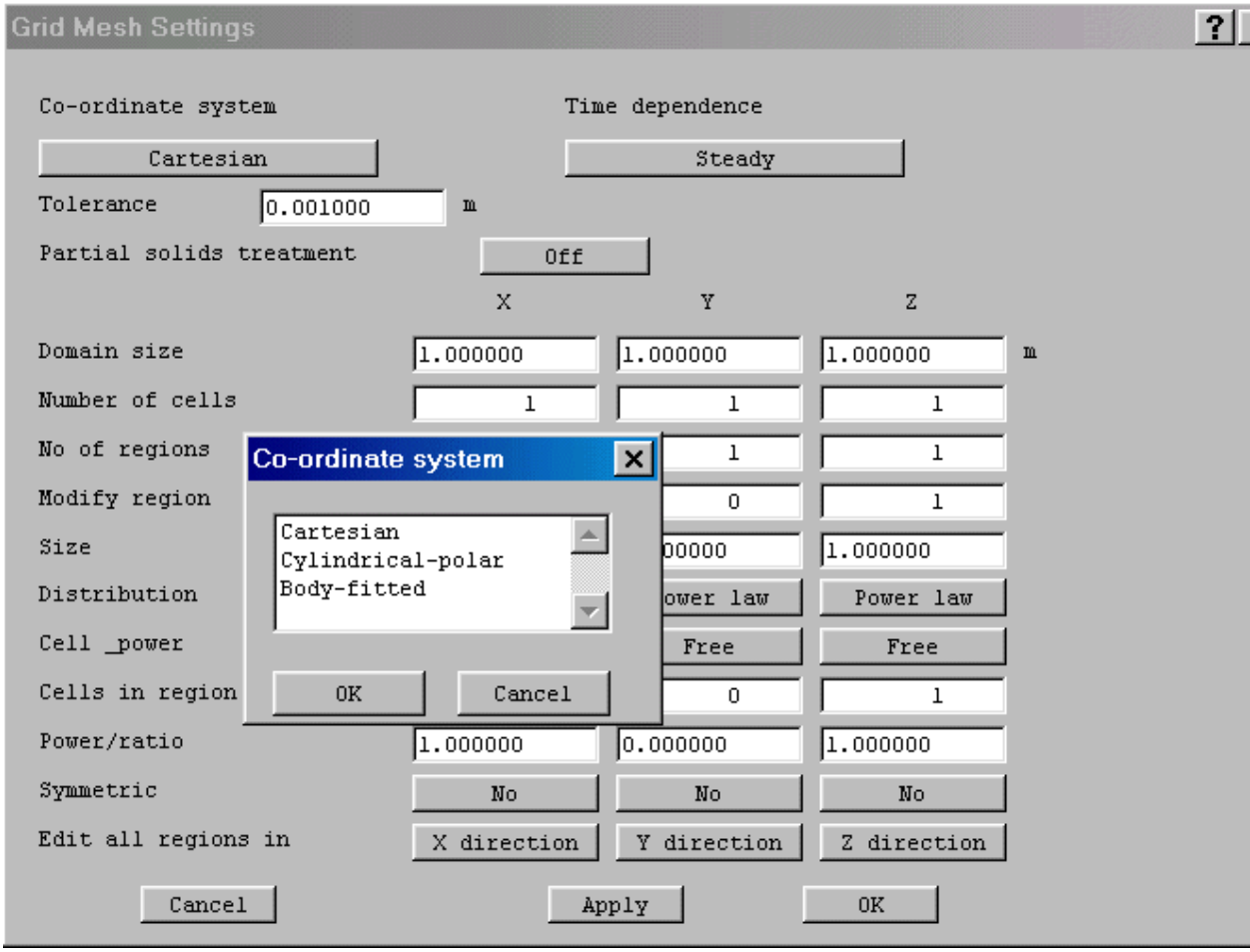
The final outcome of this tutorial is available as Library case B116.

## Accessing PHOENICS-VR.

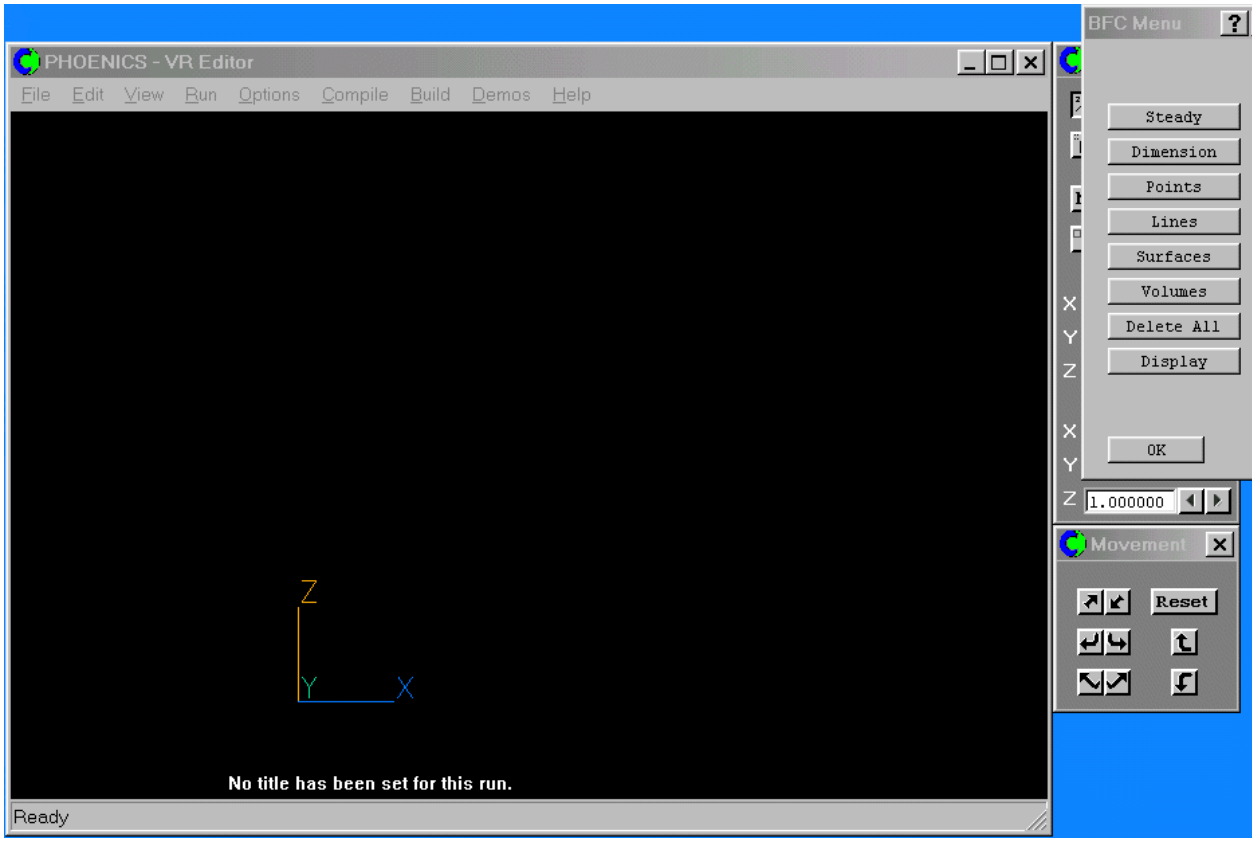
*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

Enter BFC mode by clicking on the Grid Mesh toggle  on the hand-set, then clicking anywhere within the domain to bring up the 'Grid Mesh Settings' dialog. Click on **Coordinate System - Cartesian**, and select **Body-fitted**.




Confirm the change to BFC. Activate the Grid Generator by clicking anywhere in the domain again. A new dialog will appear, overlaid over the hand-set.




This is the main BFC Menu dialog.

**Mesh generation in the Grid Generator**

The geometry is to be created on the Cartesian X-Y plane. Click the 'View down' button on the Movement control panel, and keep it pressed until the view is more or less along Z, with Y pointing up and X to the right.

To ensure that the axis orientation is absolutely as desired, click the 'Reset' button to jump the view to be along the nearest axis.

**Create the points**. Click on 'Points', then 'New Points', then 'Type x,y,z'. Enter the co-ordinates of point A as given in the table above. Change the name of the point to A. Click 'Apply' to create the point. Continue for the remaining points. After creating point I, click Cancel to quit the point creation menu. Click 'OK' twice to get back to the main BFC Menu dialog.

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

**Create the straight lines**. Click on 'Lines', then 'New lines' and then 'New Straights'. Click on A, then B. Line L1 will be created. Now click on B then C to create line L2. Continue with C-D, D-E, E-F, F-G, G-A, D-H and G-H.

**Create the arc**. Click on 'New Arcs'. Click on B, then I, then H. Arc A10 will be created. Click 'OK' to stop creating arcs.

**Set the numbers of cells along each line**. Click on 'Modify Lines'. Click on line L1. In the line modification dialog, set the number of cells to 5. Click OK. Click on line L2, and set 10 cells. Continue setting cells as follows:

| Line | Cells |
|---|---|
| L1 | 5 |
| L2 | 10 |
| L3 | 5 |
| L4 | 10 |
| L5 | 15 |
| L6 | 10 |
| L7 | 5 |
| L8 | 10 |
| L9 | 5 |
| A10 | 5 |

Click 'OK' twice to get back to the main BFC Menu dialog.

**Set the grid dimension**. Click on Dimension, and set NX to 15, NY to 15. This tells the mesh generator how many cells in total it has to set. Click 'OK'.

**Create a surface:**

**Step 1 - Create the frames**. Click on 'Surfaces', then 'New Frames'. Click on points A, B, H , G and A again to close frame F1. Click on points B, C, D, H and B to make frame F2. Click on points G, H, D, E, F and G to make frame F3. F3 has more than four points, so you must indicate which are the corners. When you click on G the second time, it will turn blue. Now click on D to indicate that it is the next corner, then E. The frame will now be made.

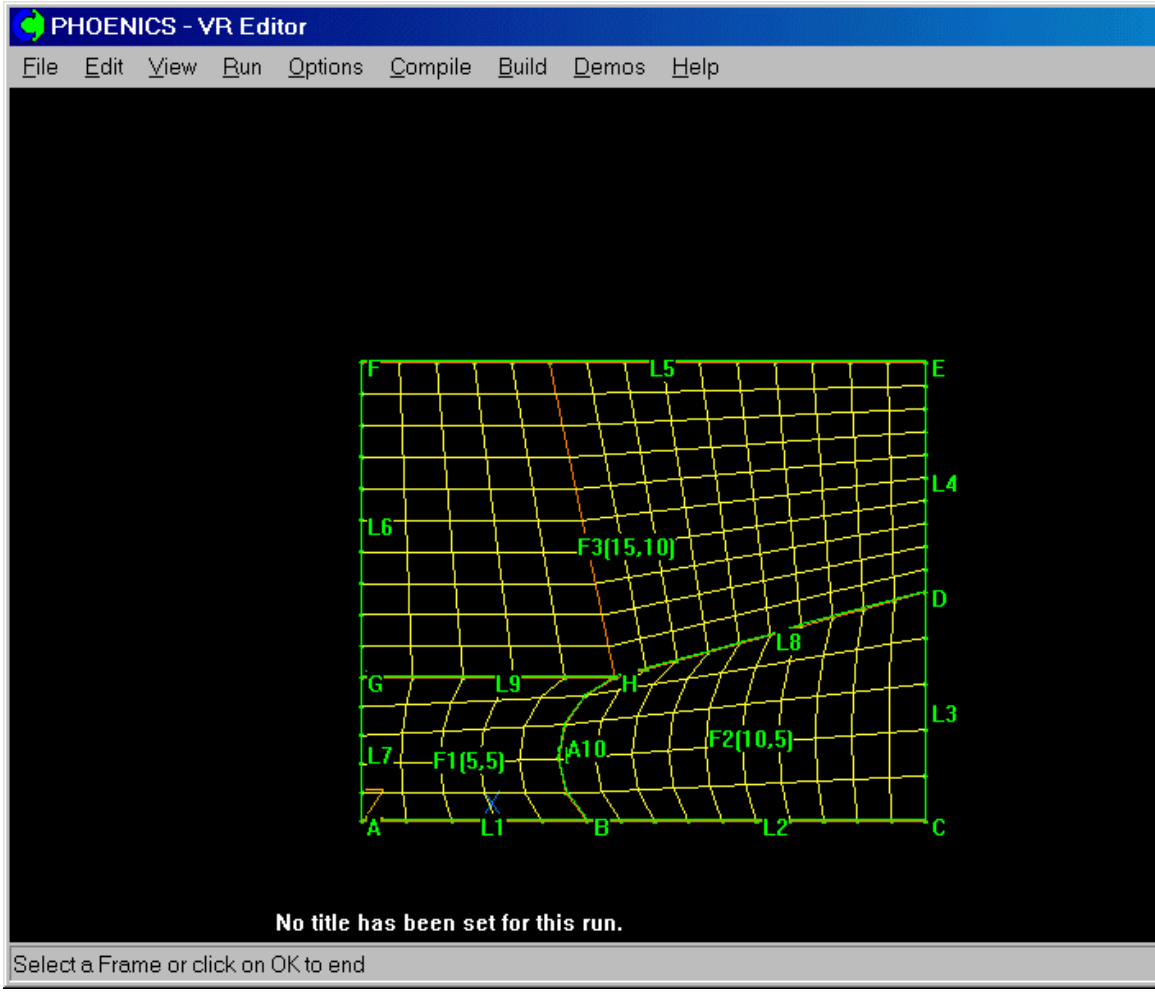
**Step 2 - Match the frames**. Click on 'Match Grid', then on the label of frame F1. The frame matching dialog will be displayed. In the 'match point A to' boxes, set I = 1, J = 1, K = 1. The direction from A -> B is +I, and B

-> C is +J, so no changes are required here. Click 'OK' to set the grid in frame F1. Click 'OK' on the Smoothing dialog to close it.

Now click on the label of frame F2. Point B should be matched to I = 6 (number of cells in L1+1), J = 1 and K = 1. The edge directions are again correct, so click 'OK' to set the grid. Click 'OK' on the Smoothing dialog to close it.

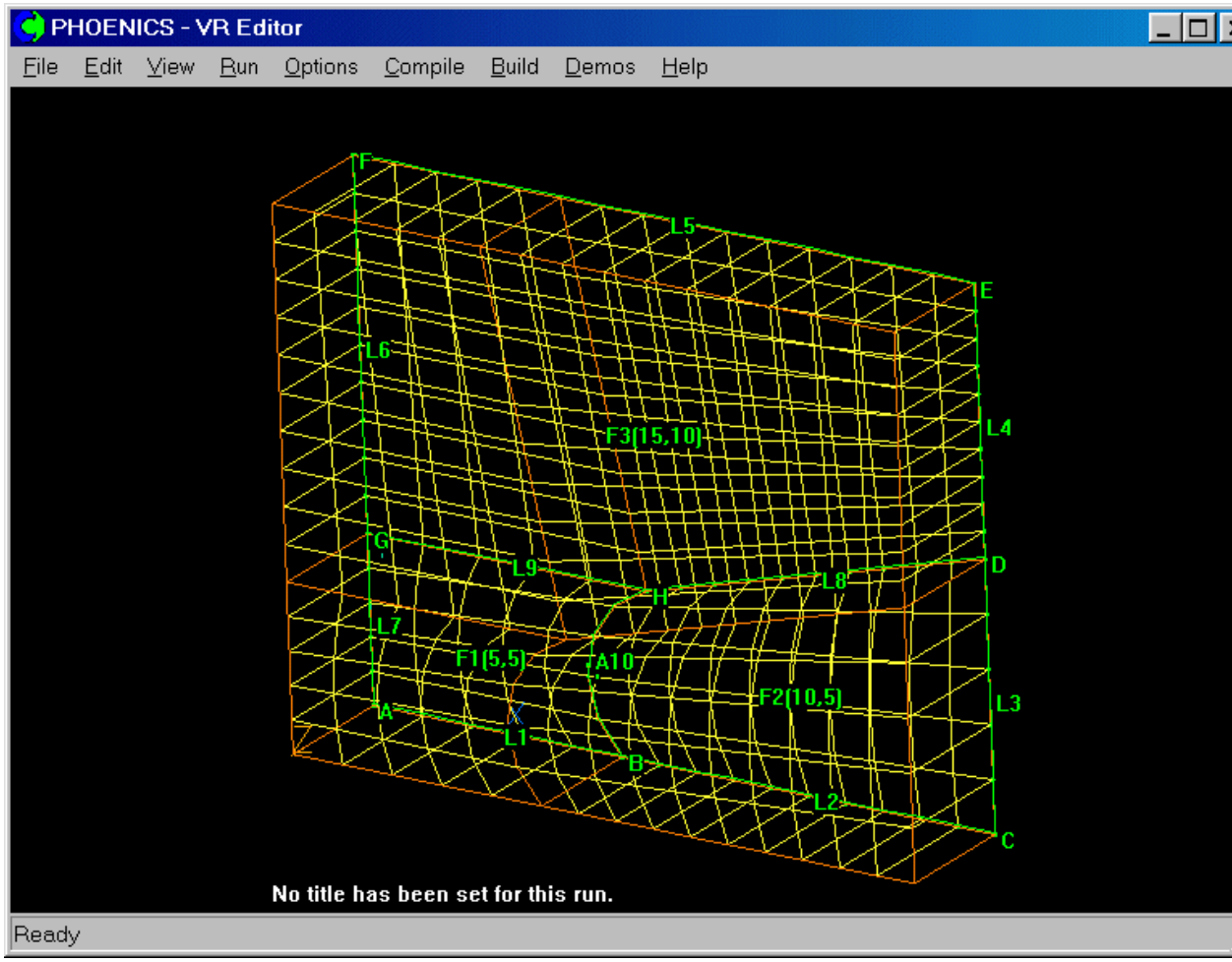
Finally click on the label of frame F3. Point G should be matched to I = 1, J = 6 (number of cells in L7+1), K = 1. The edge directions are again correct, so click 'OK' to set the grid. Click 'OK' on the Smoothing dialog to close it.

At this point, the display should look like this:




Click 'OK' twice to get back to the main BFC Menu dialog.

**Create the Volume:**
So far, the mesh has been created for the K=1 plane. PHOENICS uses a finite-volume mesh, so both faces (K=1, K=2) must be set. Click 'Volumes', then 'Extrude'. Toggle the extrude direction from I to J to K. Change the copy distance DZ from 1.0 to 0.2, then click 'OK' to close the dialog and perform the extrusion.
Click 'OK' twice to get back to the main BFC Menu dialog. Use the View control buttons to rotate the image, and see both planes of the grid.




Exit the grid generator by clicking 'OK'.
**Activate solution of variables.**
In the main menu, click on Models. Click on Equation formulation - Elliptic staggered. Select Elliptic GCV and click OK. This selects the GCV Collocated solver, which is very good at dealing with non - orthogonal (distorted) grids.
Activate solution of velocities and pressure. Select the KECHEN turbulence model.
**Set numerical controls**

Click on Numerics, and enter 100 in the 'Total number of iterations' box.
Exit the Main menu by clicking Top Menu, OK.
All the domain-related settings have been made - now set the flow boundary conditions.

**Create the blockage.**

Click on New Object. Set the size and position as:

| | |
|---|---|
| IPos | 5 |
| JPos | 0 |
| Kpos | 0 |
| ISize | 10 |
| JSize | 5 |
| KSize | 1 |

Note that in BFC, object positions are set in terms of cell corner, and sizes in terms of cells occupied. Click OK to set the blockage.

**Create the inlet at J=16**

Click on New object. Set the object position and size as:

| | |
|---|---|
| IPos | 5 |
| JPos | 15 |
| Kpos | 0 |
| ISize | 10 |
| JSize | 0 |
| KSize | 1 |

Planar objects have zero thickness in one direction. Click on Type, and select INLET. Click on ATTRIBUTES, and set the Y direction velocity to -2.0. Note that the velocities are specified in the Cartesian co-ordinate system. Click OK to close the Attributes dialog, and OK to close the Object dialog.

**Create the outlet at J=0**

Click on New Object. Set the size and position as follows:

| | |
|---|---|
| IPos | 0 |
| JPos | 0 |
| KPos | 0 |
| ISize | 5 |
| JSize | 0 |
| KSize | 1 |

Click on Type, and select OUTLET. Click on Attributes, and switch 'External turbulence' to User-set. We need to do this as the exit cuts a vortex - leaving the 'External turbulence' at the default 'In-cell' leads to excessive turbulence levels in the diffuser section. Click OK twice to close the Object dialog.

**Create three plates**

We now need three plates to provide friction conditions on the insides of the geometry.

Create the first plate at I=15 (the last cell corner in I). Click on New Object.

Set the size and position as:

| | |
|---|---|
| IPos | 15 |
| JPos | 5 |
| KPos | 0 |
| ISize | 0 |
| JSize | 10 |
| KSize | 1 |

Click on Type, and select PLATE. Click OK to close the Object dialog.

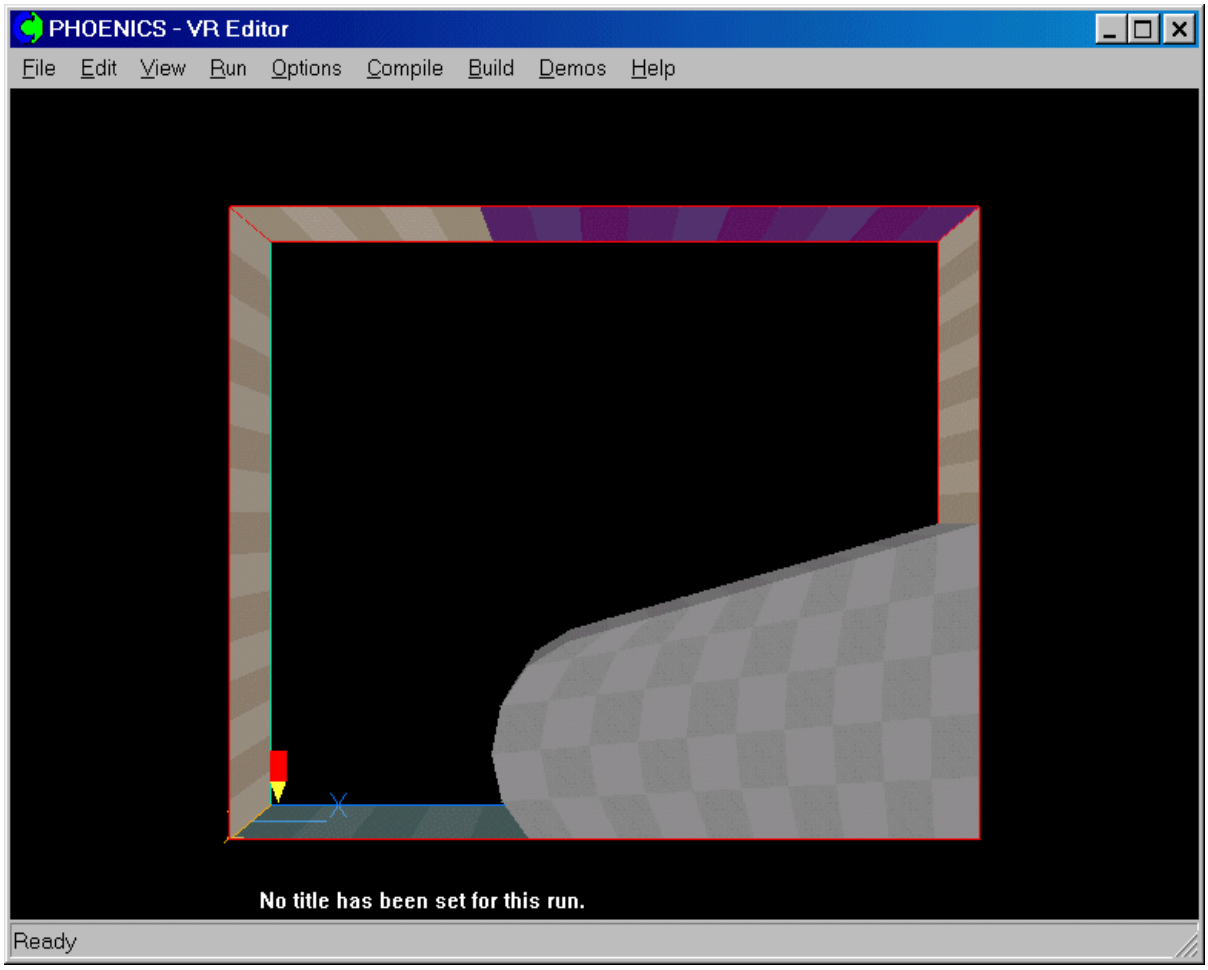
Make the second plate at I=0 by duplication. Click the Duplicate object or group button. Click the Y size up button on the hand-set until the new plate extends all the way to the top.

Create the third plate at J=15 (the last cell corner in I). Click on New Object. Set the size and position as:

| | |
|---|---|
| IPos | 0 |
| JPos | 15 |
| KPos | 0 |
| ISize | 5 |
| JSize | 0 |
| KSize | 1 |

Click on Type, and select PLATE. Click OK to close the Object dialog.

The final geometry should look like this:




**Set monitor location**

The last action before running the Earth solver is to place the monitoring point (probe) in a sensible location. Click on the image background, to ensure that no object is active. In this mode, the Position buttons on the hand-set move the probe. Use these to place the probe at (5,10,1).

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

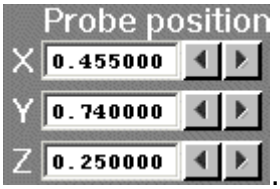
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

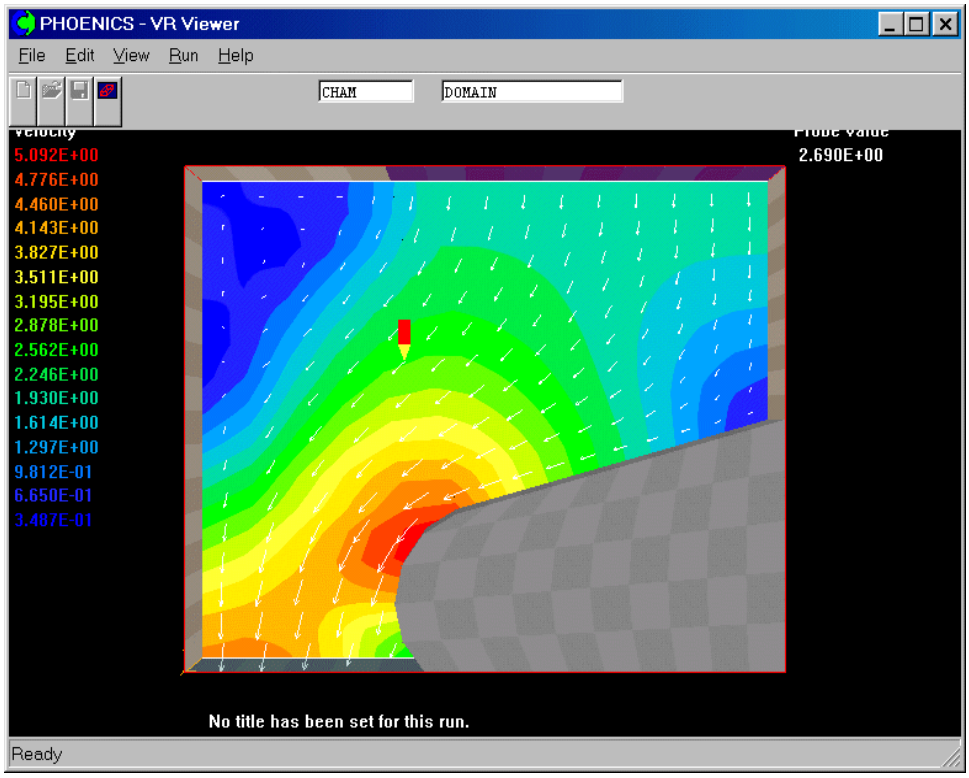
- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons



.




## Saving the results.

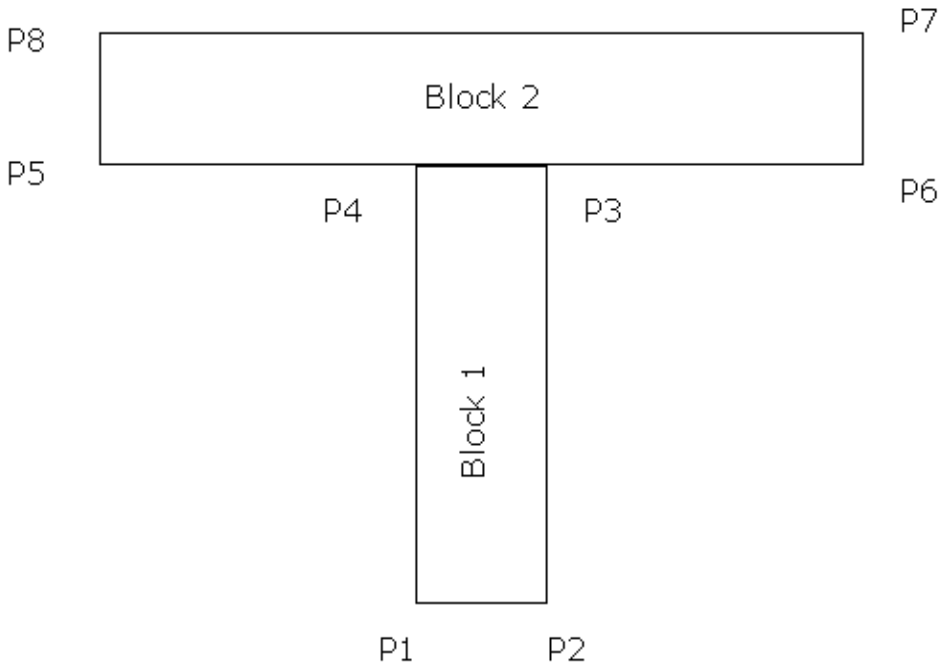
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'BFC ' (e.g.) and save as 'CASE6' (e.g.).

Home Page

# WORKSHOP - Multi-Block - 1

## "T" - Junction

This workshop goes through the steps required to create a two-block grid for a T-Junction, using the GCV solver.




It will be helpful to be familiar with the basic steps of BFC grid generation, as shown in the other BFC tutorials.

## Basic steps for Multi-Block Setup

1. Plan the decomposition of domain into individual blocks. A possible configuration is shown above. Block 1 is to be 5 x 15 x 1, and Block 2 is to be 15 x 5 x 1.
2. Generate the grid for each individual block. The grid for each block will be generated in a separate mesh-generator session.
3. Combine the grids to form the multi-block grid and define inter-block links. This step involves hand-editing the input Q1 file.

4. Define remainder of the problem (variables, boundary conditions, etc).

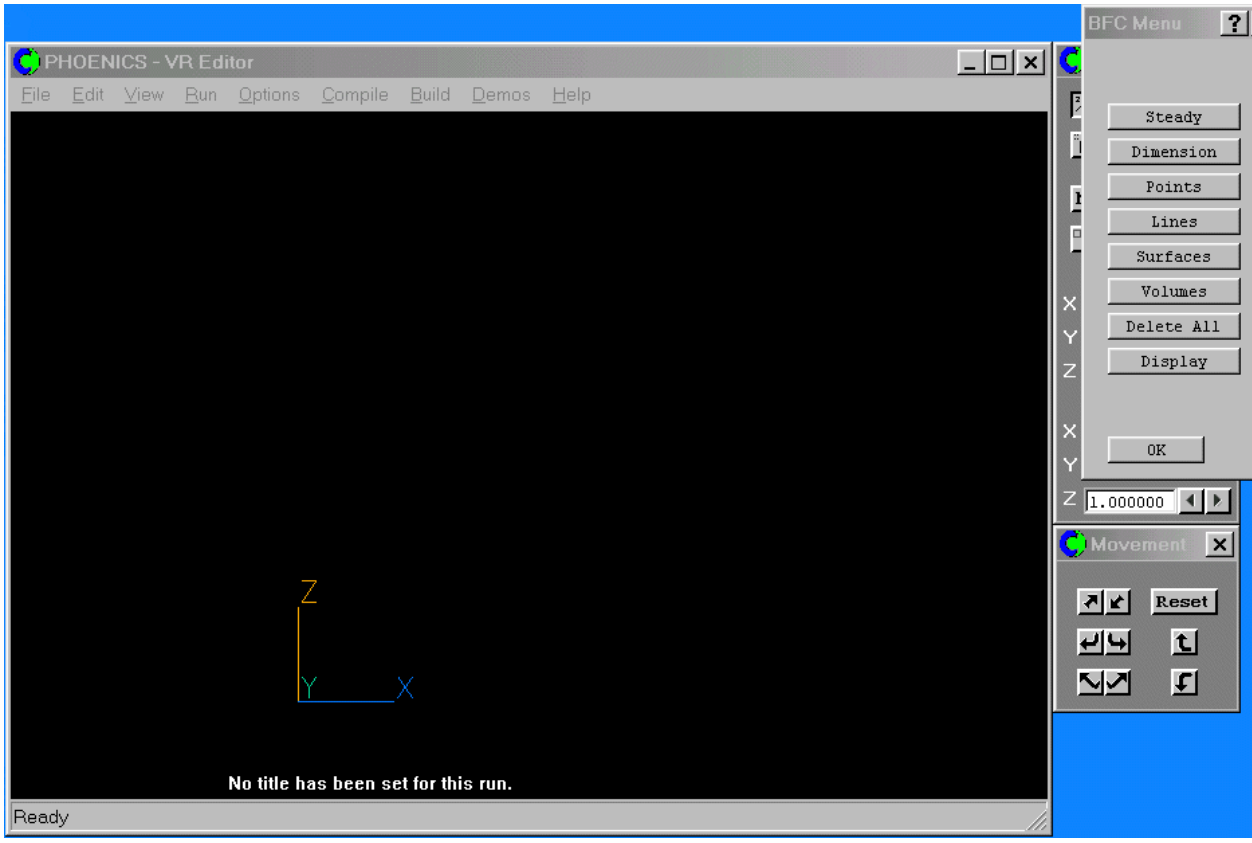
## Hints for Problem Setup

### Start

- Enter VR-Editor, go to File/New Case/Core.

### Create geometry for all blocks

- Enter BFC mode by clicking on the Grid Mesh toggle  on the hand-set, then clicking anywhere within the domain to bring up the 'Grid Mesh Settings' dialog. Click on **Coordinate System - Cartesian**, and select **Body-fitted**. Confirm the change to BFC. Activate the Grid Generator by clicking anywhere in the domain again. A new dialog will appear, overlaid over the hand-set.
- Set the view to 2-D in the x-y plane by clicking the 'View down' button on the Movement control panel, and keeping it pressed until the view is more or less along Z, with Y pointing up and X to the right. To ensure that the axis orientation is absolutely as desired, click the 'Reset' button then 'Nearest Head-on' to jump the view to be along the nearest axis.




- Create the Points and Lines for the whole geometry (i.e. both blocks). Suggested co-ordinates for the points are:

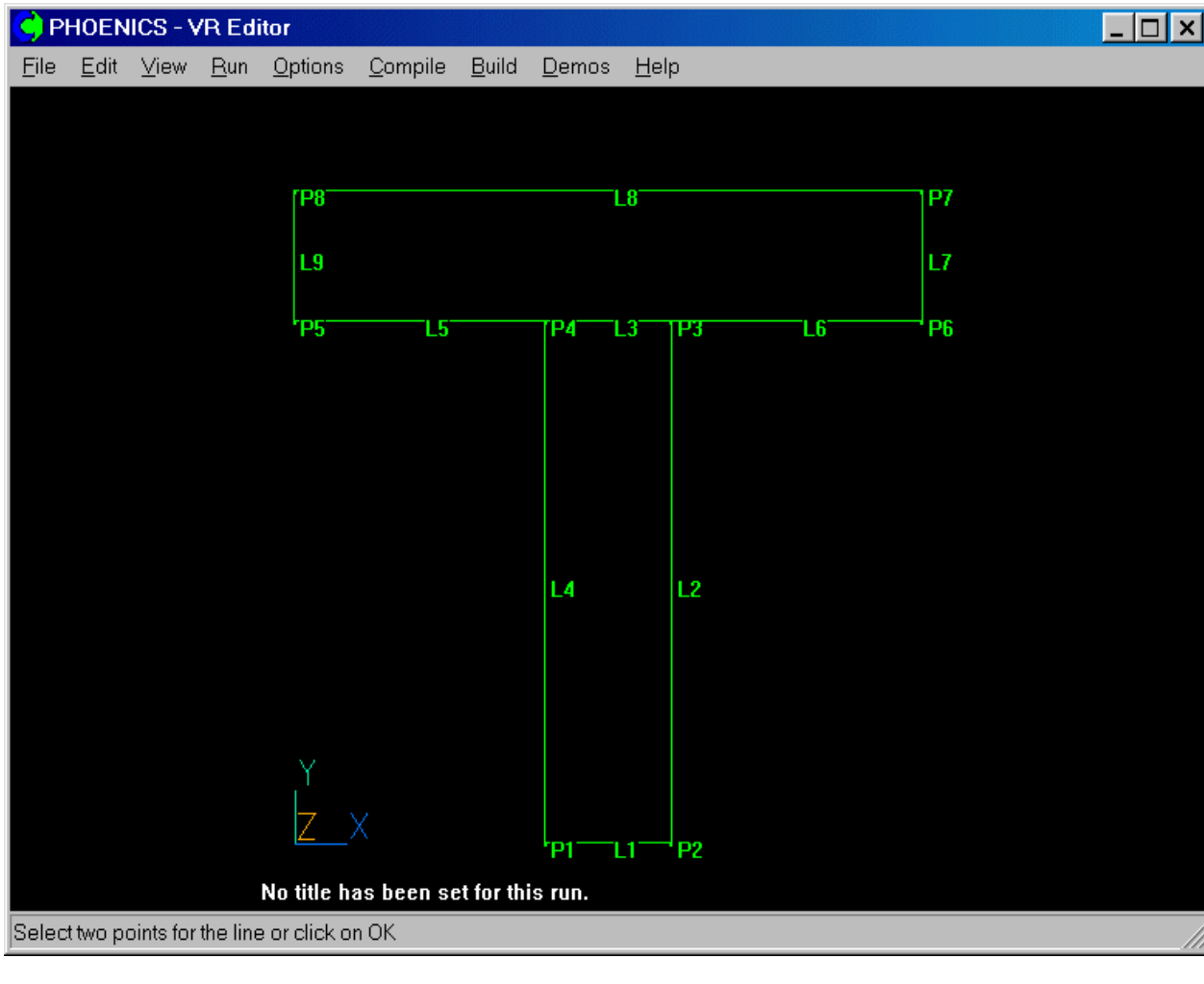
| | X | Y | Z |
|---|---|---|---|
| P1 | 1.0 | 0.0 | 0.0 |
| P2 | 1.5 | 0.0 | 0.0 |
| P3 | 1.5 | 2.0 | 0.0 |
| P4 | 1.0 | 2.0 | 0.0 |
| P5 | 0.0 | 2.0 | 0.0 |
| P6 | 2.5 | 2.0 | 0.0 |
| P7 | 2.5 | 2.5 | 0.0 |
| P8 | 0.0 | 2.5 | 0.0 |

- To create the points, on **Points, New points**, **Type x,y,z** and enter the data from the table. Click **Apply** to place each point in turn. If points appear outside the graphics window, click **Reset** on the Movement control panel to re-scale the view.
- Join the points with lines. Click on **Lines, New Lines**, **New Straights**, then on the first point of a line, then the second

point. You are in 'new line' mode until another mode is selected - there is no need to click the **New Straights** button for each line. The required lines are:

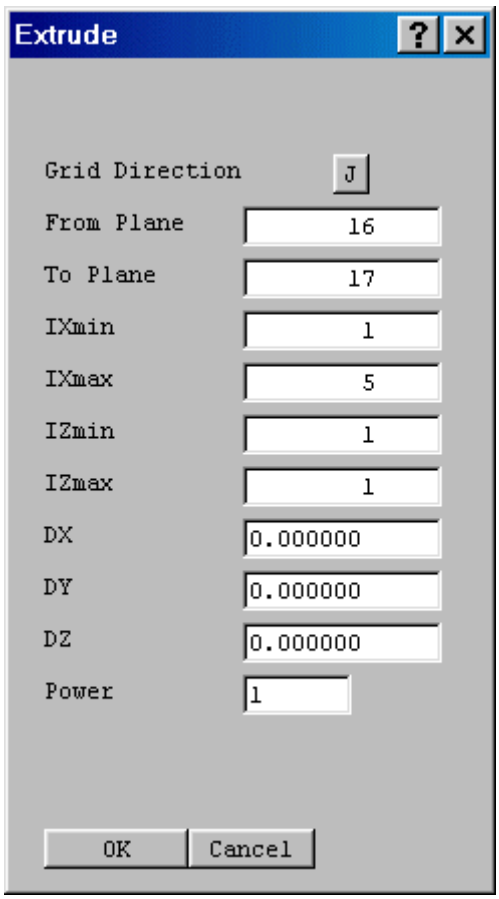
| Line | Joining points |
|---|---|
| L1 | P1 - P2 |
| L2 | P2 - P3 |
| L3 | P3 - P4 |
| L4 | P4 - P1 |
| L5 | P4 - P5 |
| L6 | P3 - P6 |
| L7 | P6 - P7 |
| L8 | P7 - P8 |
| L9 | P8 - P5 |

- After the last line, click **OK** twice to return to the main BFC Menu dialog.



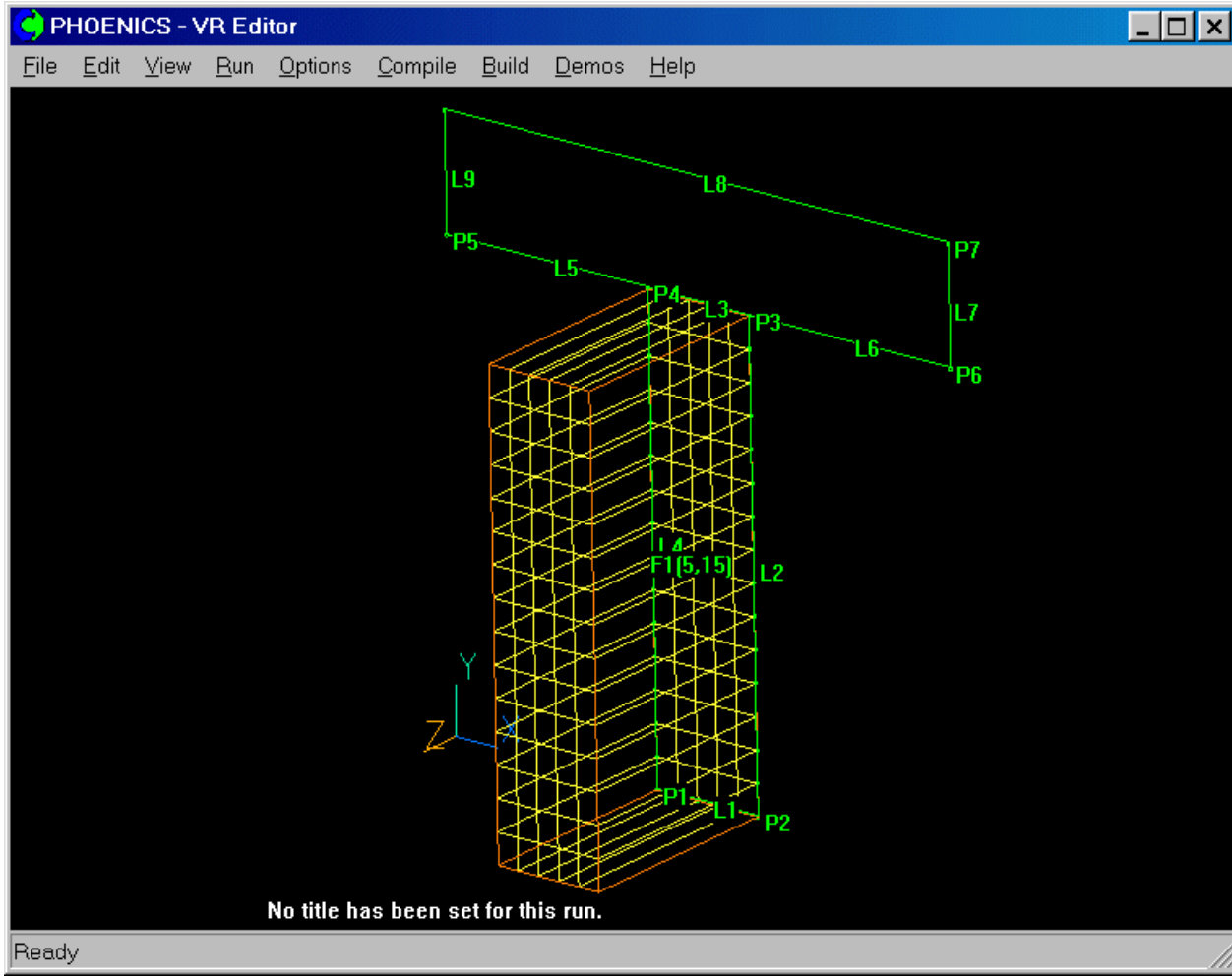
## Set grid for first block

- Click on **Dimension** to set the number of cells in Block 1 to 5 x 16 x 1 (the extra cell in Y is used because GCV uses overlapping grids between the blocks).
- Click on **Lines, Modify Lines**, then select the lines L1, L2, L3 and L4 in turn. Set the number of cells in the x-direction (L1 and L3) to 5 and the number in the y-direction (L2 and L4) to 15. After the last line, click **OK** twice to return to the main BFC Menu dialog.
- Click on **Surfaces, New Frames** to create a frame. Select the frame corner points P1, P2, P3 and P4.
- Click on **Match Grid**, then select the frame just created. Point P1 is matched to (I,J,K) (1,1,1), and the directions from P1 to P2 and P2 to P3 are +I, +J respectively. Click **OK** to set the grid, then **OK** to close the Smoothing dialog. Click **OK** to return to the main BFC Menu dialog.
- Click on **Volumes, Extrude**. Toggle 'Copy in direction' from I to K. Click **OK**. This will project the K plane in the z-direction by 1m (default).
- Click **Extrude** again. Toggle 'Copy in direction' from I to J. Change 'from plane' to 16. Set 'dy' to 0.0. This will copy J plane 17 from J plane 16 with zero offset. Click **OK**.




- The second copy creates the overlap cell required for the block link, but gives it zero thickness so that it cannot be seen. Click **OK** to perform the specified actions.

- Rotate view to check projected ok.



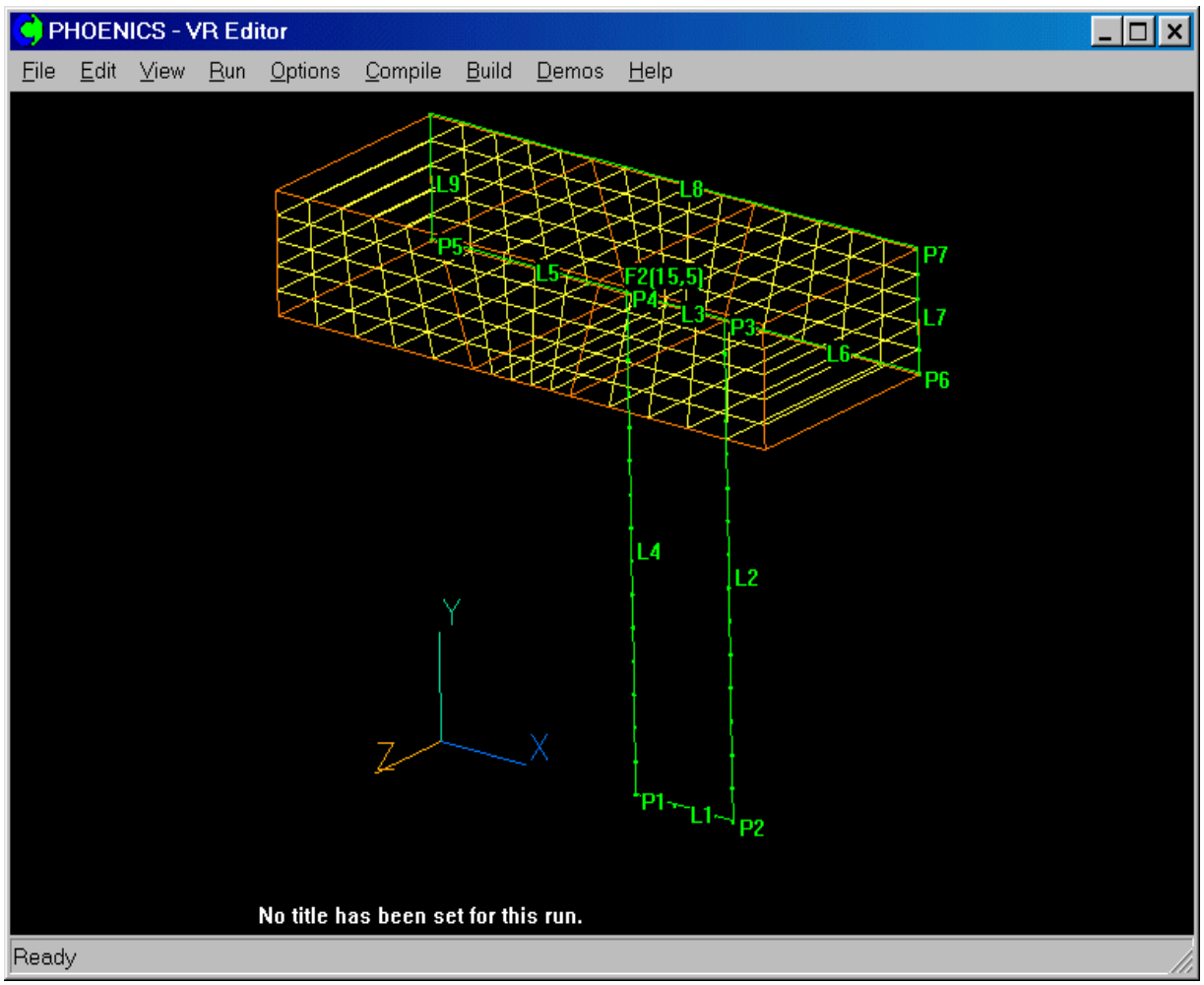

- Click on **OK** to exit the grid-generation menu.
- Click on **Menu, Geometry**. In the PIL command input box at the bottom of the geometry panel type the command DUMPC(GRI1). This will cause the grid for block 1 to be written to a file called GRI1.
- Click on **Top Menu** then OK.
- Click on File / Save as a case, and save the Q1 file as MBGRID1.

## Set grid for second block

- Enter grid generation menu again by clicking on the Grid Mesh toggle  on the hand-set, then clicking anywhere in the domain.
- Delete the two Extrude commands (Volumes / Edit / Delete / OK twice) and delete the frame (Surfaces/ Delete Frame/select frame F1/confirm)
- Reset the **Dimension** for the second block (15 x 6 x 1)
- Use **Lines, Modify Lines** to set cell numbers on Block 2

edges. Set the number of cells in the x-direction (lines L5, L3 and L6 should have 5 cells each, line l8 15 cells), and the number in the y-direction to 5 (L7 and L9).

- Create a new frame with **Surfaces, New Frames**, and match the grid with **Match Grid**. The frame points are P5, P4, P3, P6, P7 and P8. The corners are P5, P6, P7 and P8. The corners have to be selected in a second 'circuit' once the first corner (P5) has turned blue. Point P5 should be matched to (I,J,K) (1,2,1).
- Use the **Volumes, Extrude** button to extrude the K plane in the z-direction by 1m (default) - rotate view to check projected ok.
- Extrude the J plane in the y-direction from plane 2 to 1 and set dy = 0.0. This creates the overlap cell required for the block link, but gives it zero thickness so that it cannot be seen.




- Click on **OK** to exit the grid-generation menu.
- In the PIL command input box at the bottom of the geometry panel type the command DUMPC(GRI2). This will cause

the grid for block 2 to be written to a file called GRI2.

- Click on Top Menu then OK.
- Click on File / Save as a case, and save the Q1 file as MBGRID2.

## Combining the two blocks

- Click on File / Open file for editing / Q1
- In Notepad, delete the entire contents of Q1, and replace it with:

```
TALK=T;RUN( 1, 1)

BFC=T

NUMBLK=2

READCO(GRI+)

MPATCH(1,MBL1-2,NORTH,1,NX,NY-1,NY-1,1,NZ,1,1)

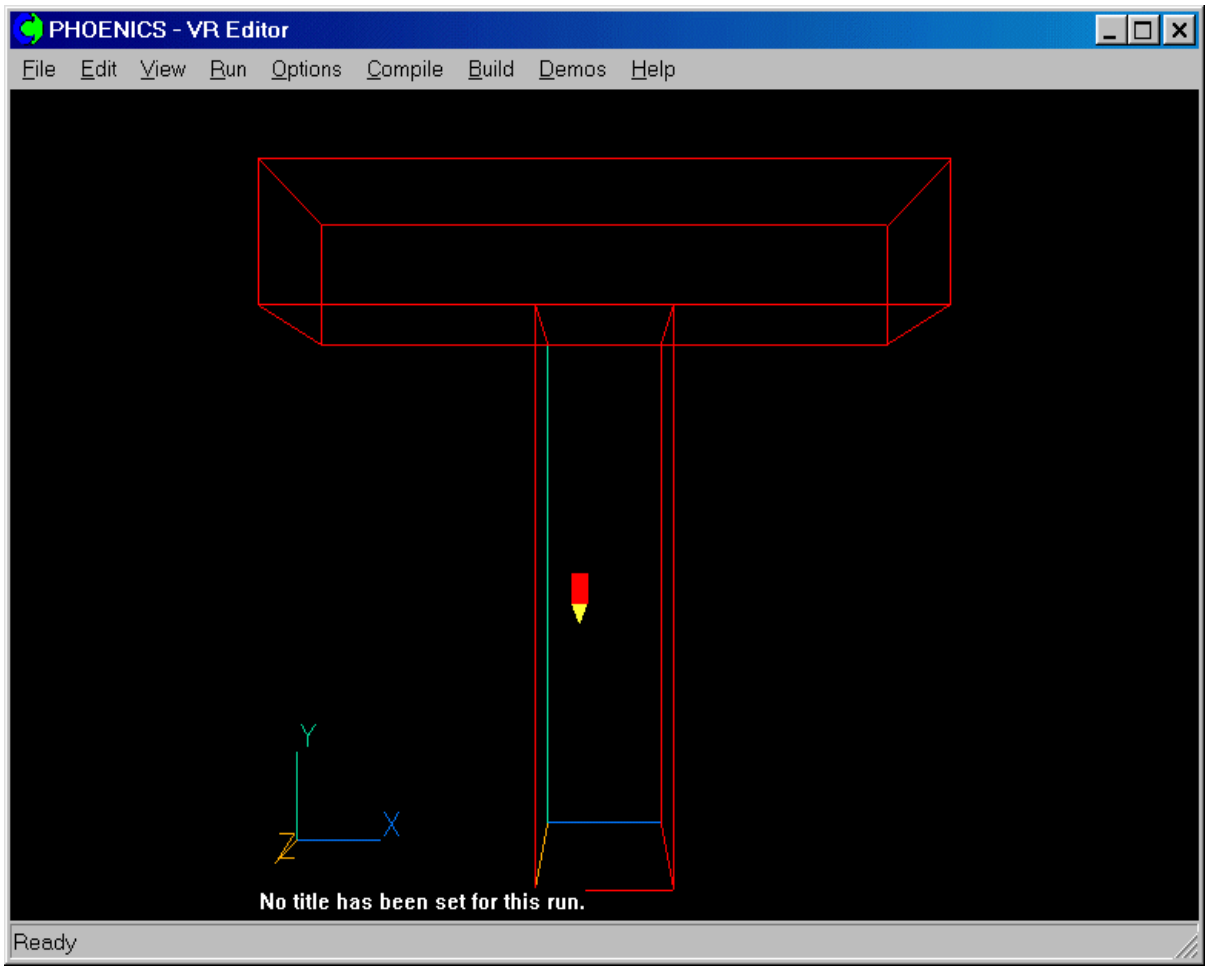
MPATCH(2,MBL2-1,SOUTH,6,10,2,2,1,NZ,1,1)

STOP
```

Save and exit Notepad.

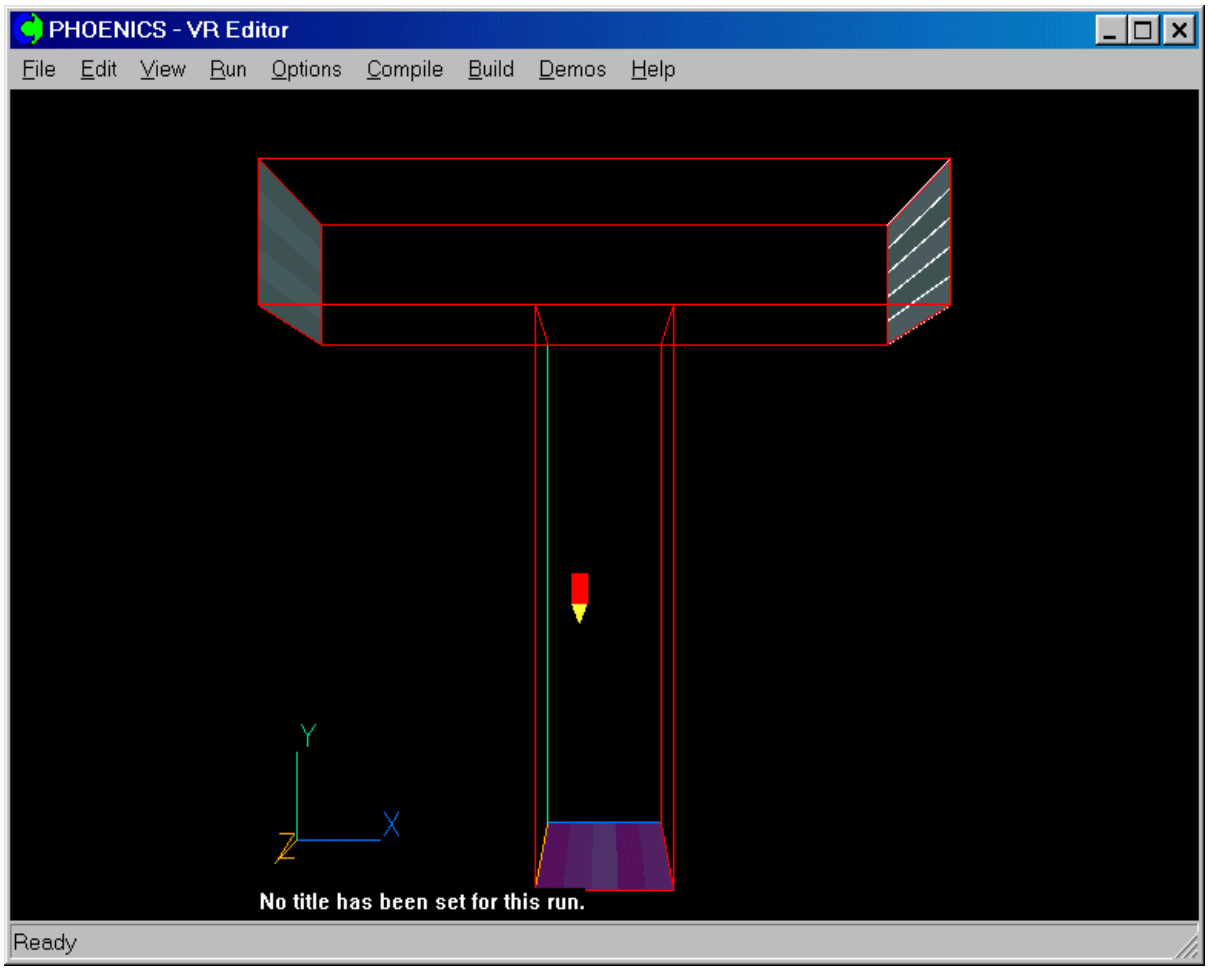
The READCO command reads the two block files and assembles them. The MPATCH commands link the two blocks. These are the only operations which cannot be performed from a dialog box.

- Allow VR-Editor to re-load the Q1. This will display your multi-block domain!!




## Setting up the remainder of the problem

- Click on **Menu** on the hand-set. Click on **Models**. Turn **Solution for velocities and pressure** ON.
- Click on **Numerics**. Set **Total number of iterations** to 100.
- Click **Top menu** and **OK** to quit the Main menu.
- Create the inlet: click on **Obj** on the hand-set. Set the **Isize** to 5 and the **Jsize** to 0. Click on **Type**, and select **Inlet**. Click on **Attributes**, and set the **Y direction** velocity to 1.0 m/s. Click **OK** to close the Attributes dialog, and **OK** to create the new inlet.
- Create the left outlet: click on **Obj** on the hand-set. Change the **Current block number** to 2. Set the **Isize** to 0 and the **Jsize** to 5. Click on **Type** and select **Outlet**. Click **OK** to create the new outlet.
- Create the right outlet: click on **Duplicate Object** on the hand-set, then on X position up to move the copied object to the right-hand side of the domain. The screen image should now show:




## Solution

- Run Earth

- Save as case MBGRID3. Should you wish to modify the grid for either block, open case MBGRID1 or MBGRID2, and enter the mesh generator. On exit, issue the DUMPC command. Re-load MBGRID3, and the new grids will be read in. At that point, you may have to re-size and re-position the inlets and outlets.

## Results

- Run VR-Viewer to view the results.

  In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.

  To view:

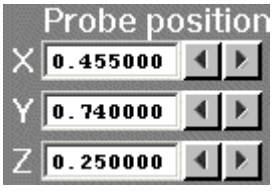
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

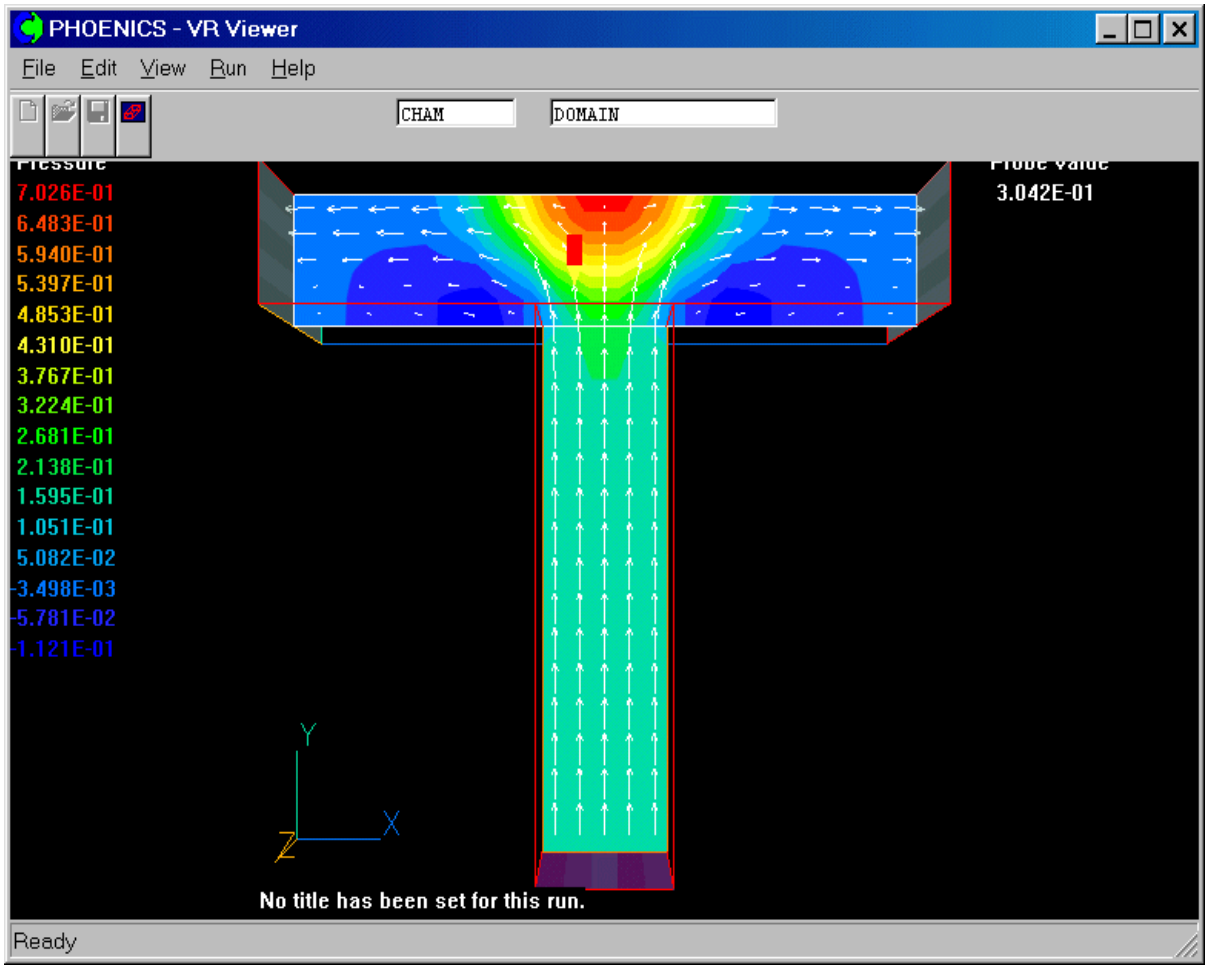
- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons



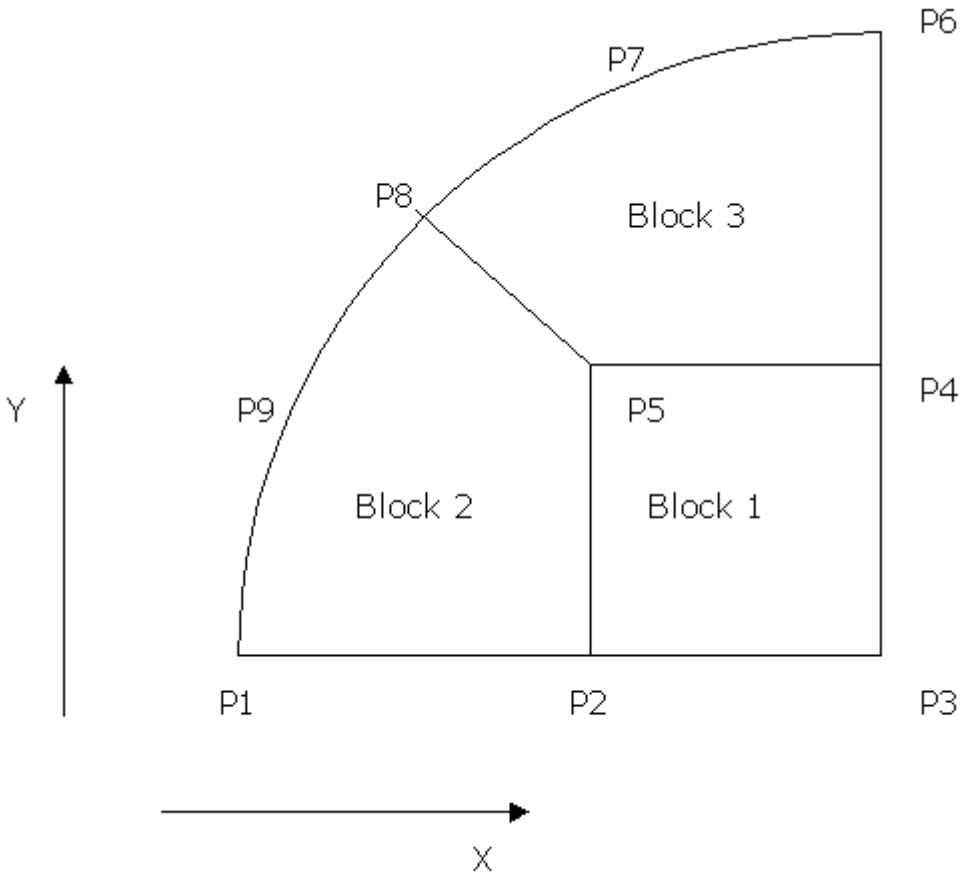

- To get contours and vectors in both blocks, click on the **Create slice** toggle when the probe is in Block 1. Now move the probe up into Block 2. The saved slice will display the contours or vectors selected for Block 2.


PHOENICS - VR Viewer
File Edit View Run Help
CHAM
DOMAIN
Pressure
7.026E-01
6.483E-01
5.940E-01
5.397E-01
4.853E-01
4.310E-01
3.767E-01
3.224E-01
2.681E-01
2.138E-01
1.595E-01
1.051E-01
5.082E-02
-3.498E-03
-5.781E-02
-1.121E-01
Probe Value
3.042E-01
Y
X
Z
No title has been set for this run.
Ready

# WORKSHOP - Multi-Block - 2

## Un-natural Links

This workshop goes through the steps required to create a three-block grid with an un-natural link, using the GCV solver.




It will be helpful to be familiar with the basic steps of BFC grid generation, as shown in the other BFC tutorials.

## Basic steps for Multi-Block Setup

1. Plan the decomposition of domain into individual blocks. A possible configuration is shown above. Each block is to be 5 x 5 x 1.
2. Generate the grid for each individual block. The grid for each block will be generated in a separate mesh-generator

session.

3. Combine the grids to form the multi-block grid and define inter-block links. This step involves hand-editing the input Q1 file.
4. Define remainder of the problem (variables, boundary conditions, etc).

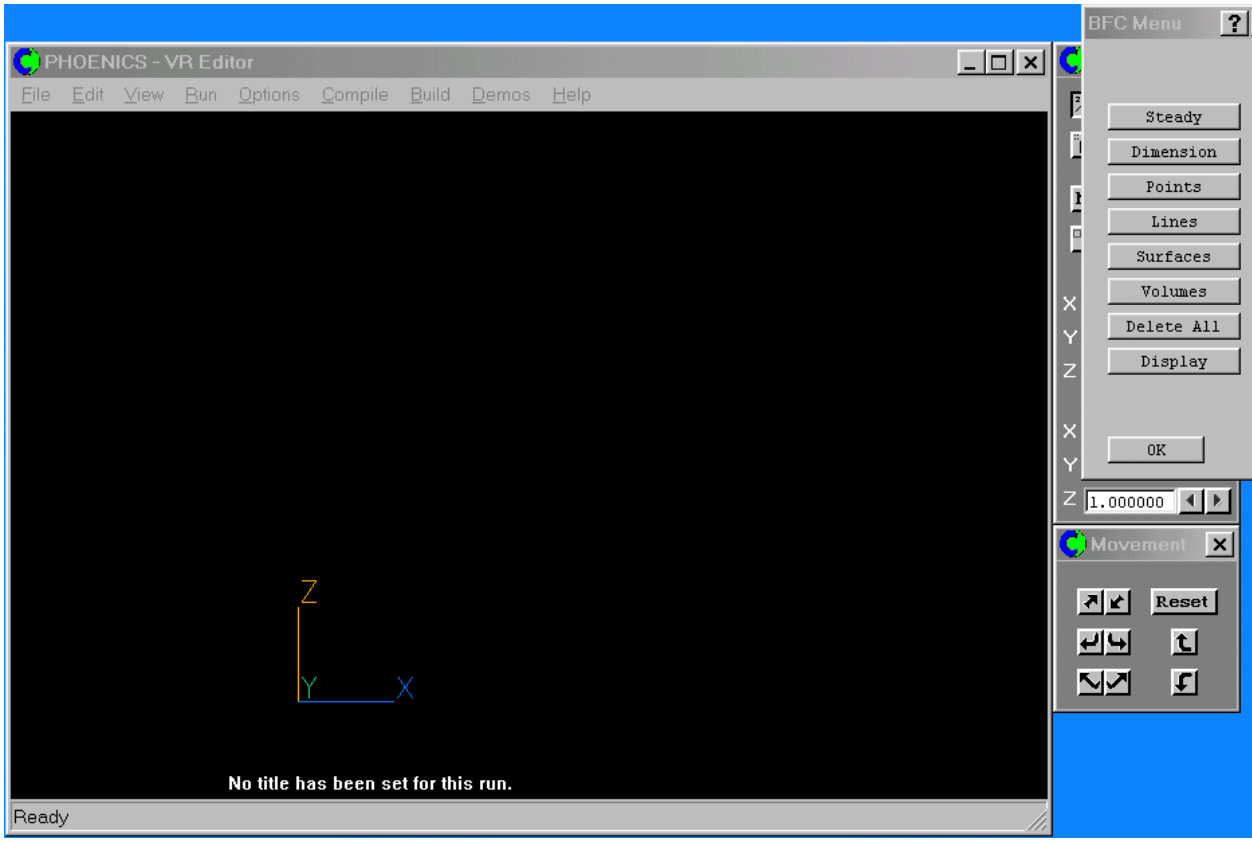
## Hints for Problem Setup

### Start

- Enter VR-Editor, go to File/New Case/Core.

### Create geometry for all blocks

- Enter BFC mode by clicking on the Grid Mesh toggle  on the hand-set, then clicking anywhere within the domain to bring up the 'Grid Mesh Settings' dialog. Click on **Coordinate System - Cartesian**, and select **Body-fitted**. Confirm the change to BFC. Activate the Grid Generator by clicking anywhere in the domain again. A new dialog will appear, overlaid over the hand-set. Set the view to 2-D in the x-y plane by clicking the 'View down' button on the Movement control panel, and keeping it pressed until the view is more or less along Z, with Y pointing up and X to the right. To ensure that the axis orientation is absolutely as desired, click the 'Reset' button to jump the view to be along the nearest axis.




- Create the Points and Lines for the whole geometry (i.e. both blocks). Suggested co-ordinates for the points are:

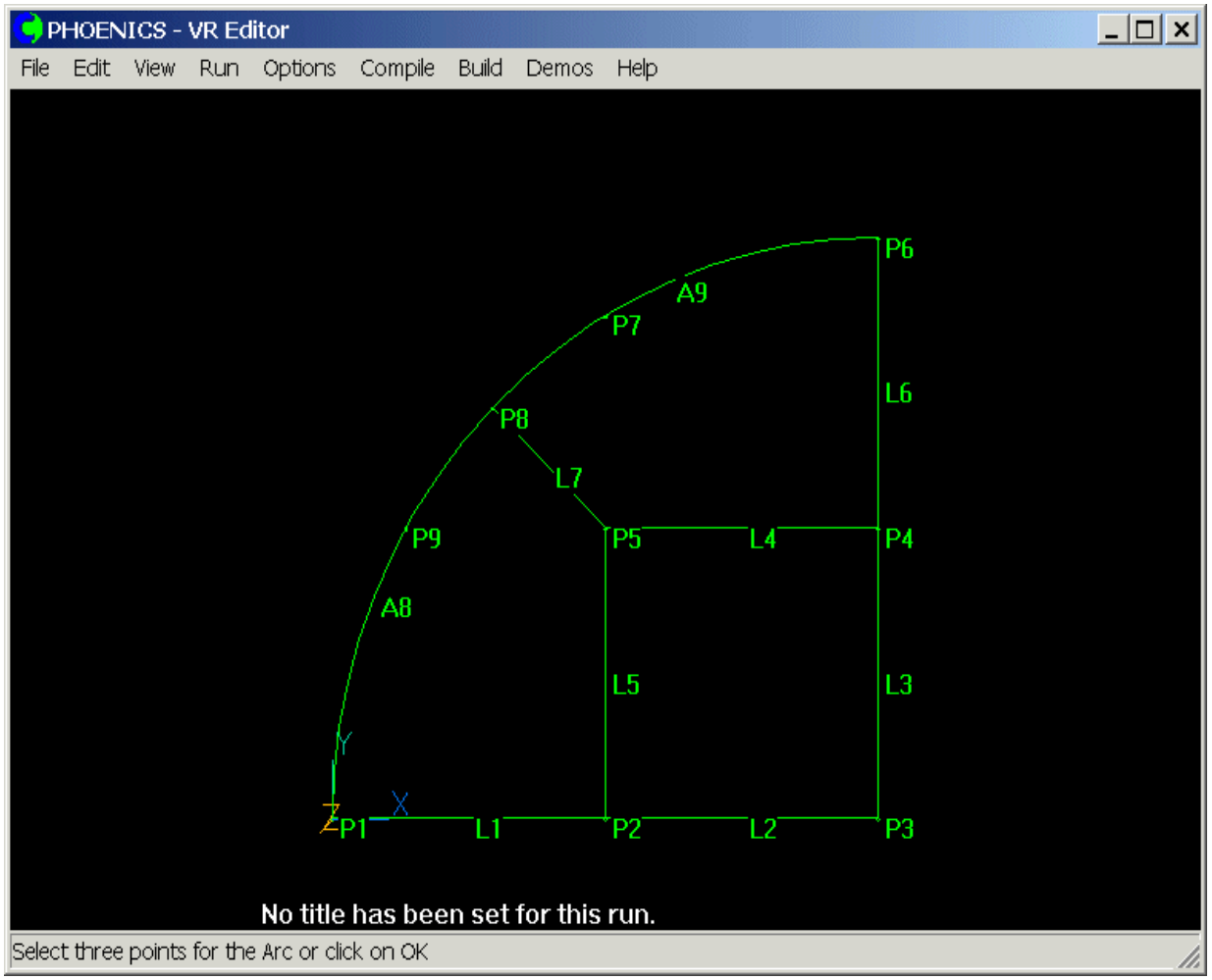
| Point | x | y | z |
|---|---|---|---|
| P1 | 0 | 0 | 0 |
| P2 | 1 | 0 | 0 |
| P3 | 2 | 0 | 0 |
| P4 | 2 | 1 | 0 |
| P5 | 1 | 1 | 0 |
| P6 | 2 | 2 | 0 |
| P7 | 1 | 1.732 | 0 |
| P8 | 0.5858 | 1.414 | 0 |
| P9 | 0.2679 | 1 | 0 |

- To create the points, on **Points, New points**, **Type x,y,z** and enter the data from the table. Click **Apply** to place each point in turn. If points appear outside the graphics window, click **Reset** on the Movement control panel to re-scale the view.
- Joint the points with lines. Click on **Lines, New Lines**, **New**

**Straights**, then on the first point of a line, then the second point. You are in 'new line' mode until another mode is selected - there is no need to click the **New Straights** button for each line. Once the straight lines have been entered, click on 'New Arcs' and add the two arcs. The required lines are:

| Name | Type | Joining points |
|---|---|---|
| L1 | Straight | P1 - P2 |
| L2 | Straight | P2 - P3 |
| L3 | Straight | P3 - P4 |
| L4 | Straight | P4 - P5 |
| L5 | Straight | P5 - P2 |
| L6 | Straight | P4 - P6 |
| L7 | Straight | P5 - P8 |
| A8 | Arc | P1 - P9 - P8 |
| A9 | Arc | P8 - P7 - P6 |

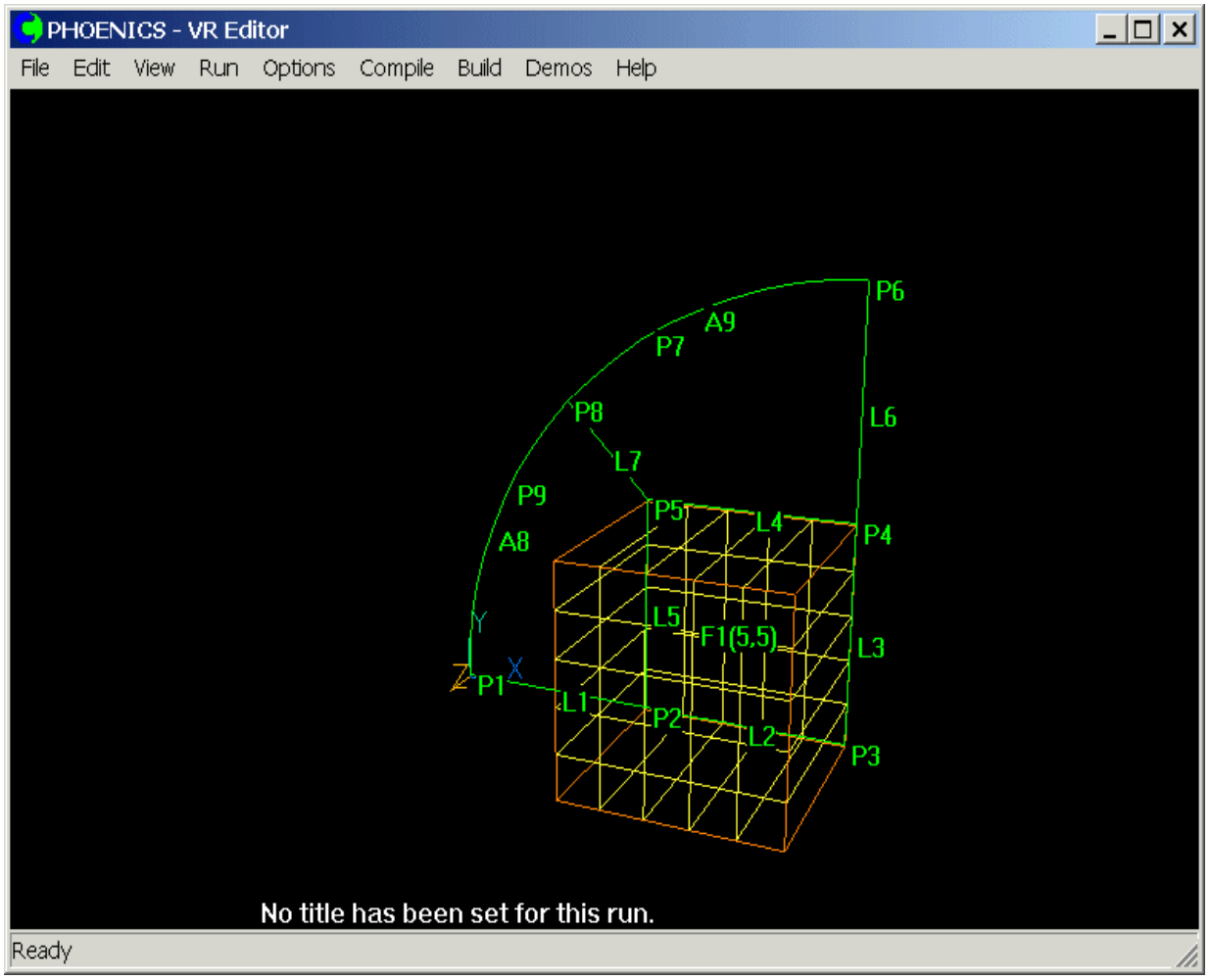
After the last line, click **OK** twice to return to the main BFC Menu dialog.




## Set grid for first block

- Click on **Dimension** to set the number of cells in Block 1 to 6 x 6 x 1 (the extra cells in X and Y are used because GCV uses overlapping grids between the blocks).
- Click on **Lines, Modify Lines**, then select the lines L2, L3, L4 and L5 in turn. Set the number of cells in the x-direction (L3 and L5) to 5 and the number in the y-direction (L2 and L4) to 5. After the last line, click **OK** twice to return to the main BFC Menu dialog.
- Click on **Surfaces, New Frames** to create a frame. Select the frame corner points P2, P3, P4 and P5.
- Click on **Match Grid**, then select the frame just created. Point P2 is matched to (I,J,K) (2,1,1), and the directions from P2 to P3 and P3 to P4 are +I, +J respectively. Click **OK** to set the grid, then **OK** to close the Smoothing dialog. Click **OK** to return to the main BFC Menu dialog.
- Click on **Volumes, Extrude**. Toggle 'Copy in direction' from I to K. Click **OK**. This will project the K plane in the z-direction by 1m (default).

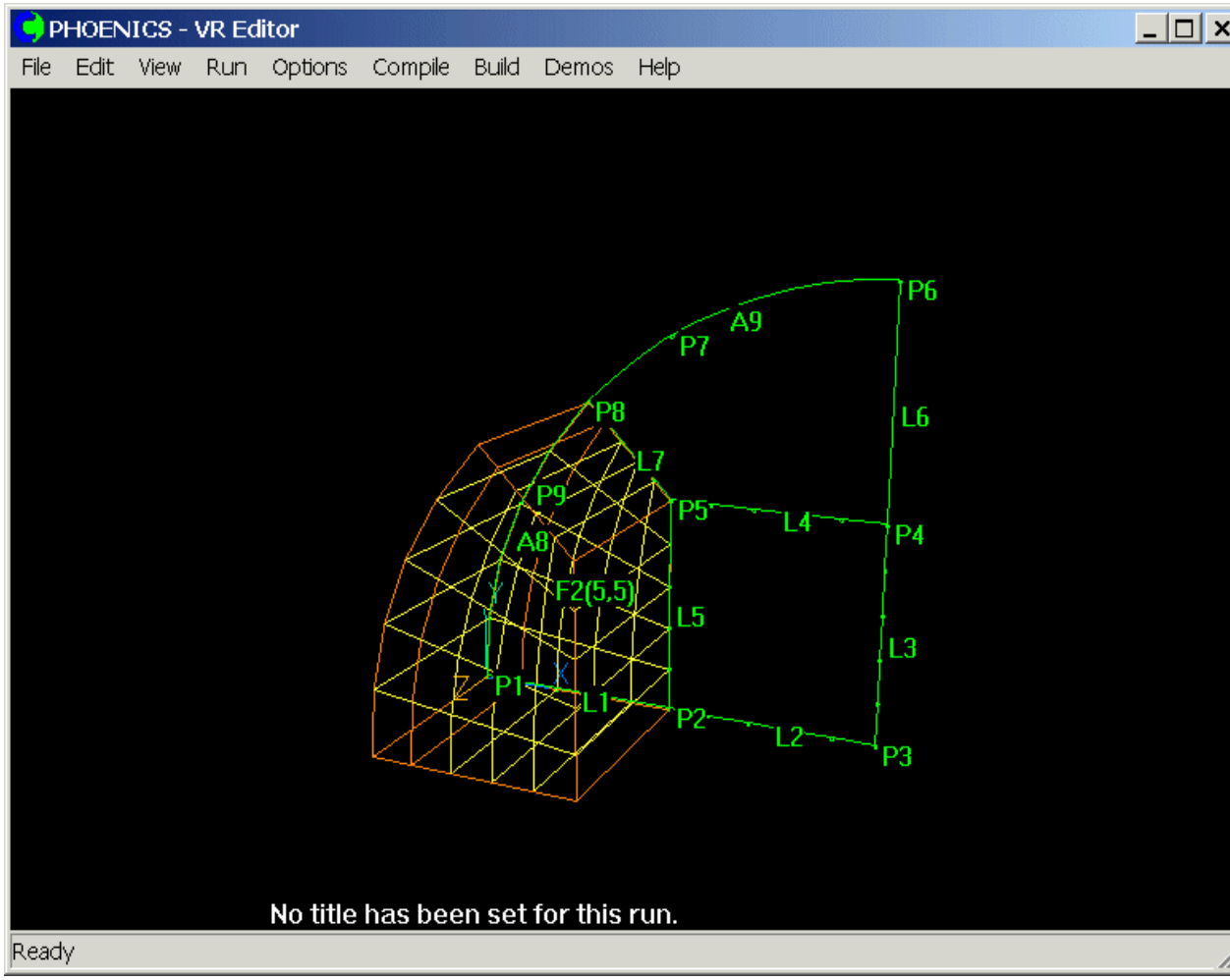
- Click **Extrude** again. Toggle 'Copy in direction' from I to J. Change 'from plane' to 6. Set 'dy' to 0.0. This will copy J plane 7 from J plane 6 with zero offset. Click **OK**.
- Click **Extrude** again. Leave 'Copy in direction' at I. Change 'from plane' to 2. Change 'to plane' to 1. Set 'dx' to 0.0. This will copy I plane 1 from I plane 2 with zero offset. Click **OK**.
- The last two copies create the overlap cells required for the block link, but give them zero thickness so that they cannot be seen. Click **OK** to perform the specified actions.
- Rotate view to check projected ok.




- Click on **OK** to exit the grid-generation menu.
- Click on **Menu, Geometry**. In the PIL command input box at the bottom of the geometry panel type the command DUMPC(UGRI1). This will cause the grid for block 1 to be written to a file called UGRI1.
- Click on Top Menu then OK.
- Click on File / Save as a case, and save the Q1 file as UGRID1.

## Set grid for second block

- Enter grid generation menu again by clicking on the Grid Mesh toggle  on the hand-set, then clicking anywhere in the domain.
- Delete the three Extrude commands (Volumes / Edit / Delete / OK three times) and delete the frame (Surfaces/ Delete Frame/select frame F1/confirm).
- Use **Lines, Modify Lines** to set cell numbers on Block 2 edges. Set the number of cells in the x-direction (L1 and L7) to 5 and the number in the y-direction (A8 and L5) to 5.
- Create a new frame with **Surfaces, New Frames**, and match the grid with **Match Grid**. The frame corner points are P1, P2, P5 and P8. Point P1 should be matched to (I,J,K) (1,1,1).
- Use the **Volumes, Extrude** button to extrude the K plane in the z-direction by 1m (default) - rotate view to check projected ok.
- Extrude the J plane in the y-direction from plane 6 to 7 and set dy = 0.0. Extrude the I plane in the x-direction from plane 6 to 7 and set dx = 0.0. This creates the overlap cells required for the block link, but gives them zero thickness so that they cannot be seen.
- Rotate view to check projected ok.



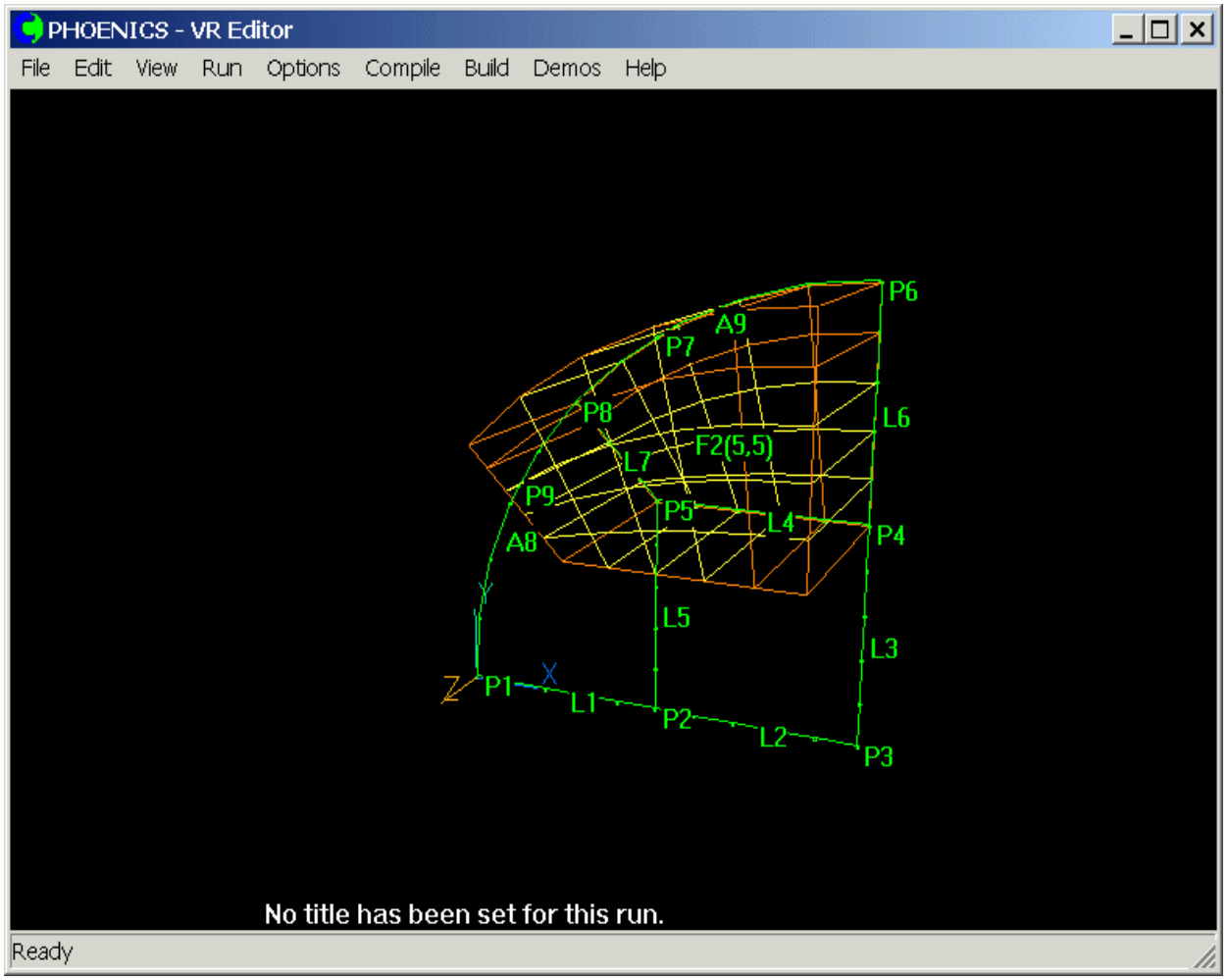

- Click on **OK** to exit the grid-generation menu.
- In the PIL command input box at the bottom of the geometry panel type the command DUMPC(UGRI2). This will cause the grid for block 2 to be written to a file called UGRI2.
- Click on Top Menu then OK.
- Click on File / Save as a case, and save the Q1 file as UGRID2.

## Set grid for third block

- Enter grid generation menu again by clicking on the Grid Mesh toggle  on the hand-set, then clicking anywhere in the domain.
- Delete the three Extrude commands (Volumes / Edit / Delete / OK three times) and delete the frame (Surfaces/ Delete Frame / select frame F2 / confirm)
- Use **Lines, Modify Lines** to set cell numbers on Block 3 edges. Set the number of cells in the x-direction (L4 and A9) to 5 and the number in the y-direction (L7 and L6) to

5.

- Create a new frame with **Surfaces, New Frames**, and match the grid with **match Grid**. The frame corner points are P5, P4, P6 and P8. Point P5 should be matched to (I,J,K) (2,2,1).
- Use the **Volumes, Extrude** button to extrude the K plane in the z-direction by 1m (default).
- Extrude the J plane in the y-direction from plane 2 to 1 and set dy = 0.0. Project the I plane in the x-direction from plane 2 to 1 and set dx = 0.0. This creates the overlap cells required for the block link, but gives them zero thickness so that they cannot be seen.
- Rotate view to check projected ok.




- Click on **OK** to exit the grid-generation menu.
- In the PIL command input box at the bottom of the geometry panel type the command DUMPC(UGRI3). This will cause the grid for block 3 to be written to a file called UGRI3.
- Click on Top Menu then OK.
- Click on File / Save as a case, and save the Q1 file as

UGRID3.

## Combining the three blocks

- Click on File / Open file for editing / Q1
- In Notepad, delete the entire contents of Q1, and replace it with:

```
TALK=T;RUN(1,1)

BFC=T

GCV=T

NUMBLK=3

READCO(UGRI+)

MPATCH(1,MBL1.2,WEST,2,2,1,NY-1,1,NZ,1,1)

MPATCH(2,MBL2.1,EAST,NX-1,NX-1,1,NY-1,1,NZ,1,1)

MPATCH(1,MBL1.3,NORTH,2,NX,NY-1,NY-1,1,NZ,1,1)

MPATCH(3,MBL3.1,SOUTH,2,NX,2,2,1,NZ,1,1)

MPATCH(2,MBL2.3,NORTH,1,NX-1,NY-1,NY-1,1,NZ,1,1)

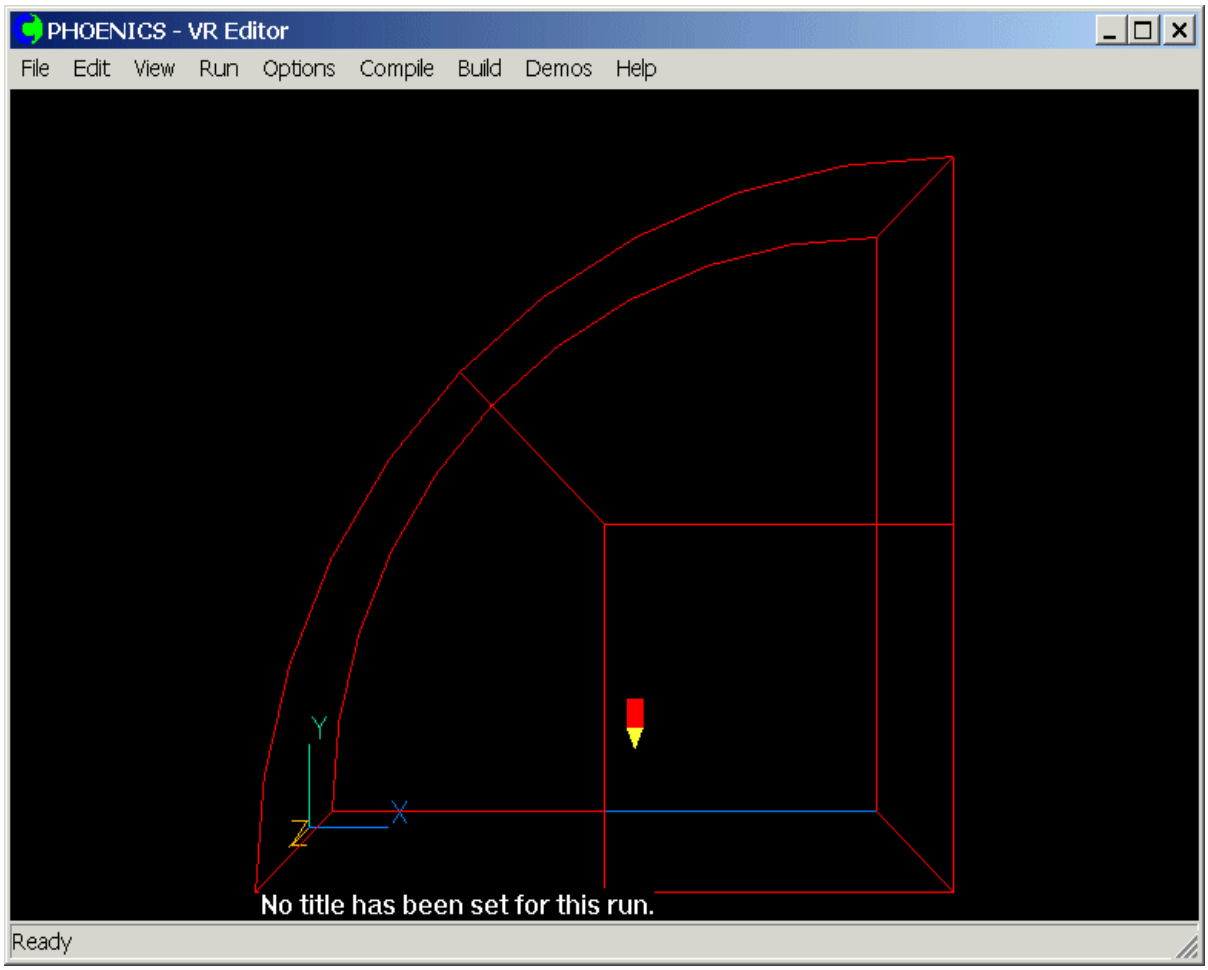
MPATCH(3,MBL3.2,WEST,2,2,2,NY,1,NZ,1,1)

SPEDAT(SET,GCV,MBL3.2,C,WNL)

STOP
```

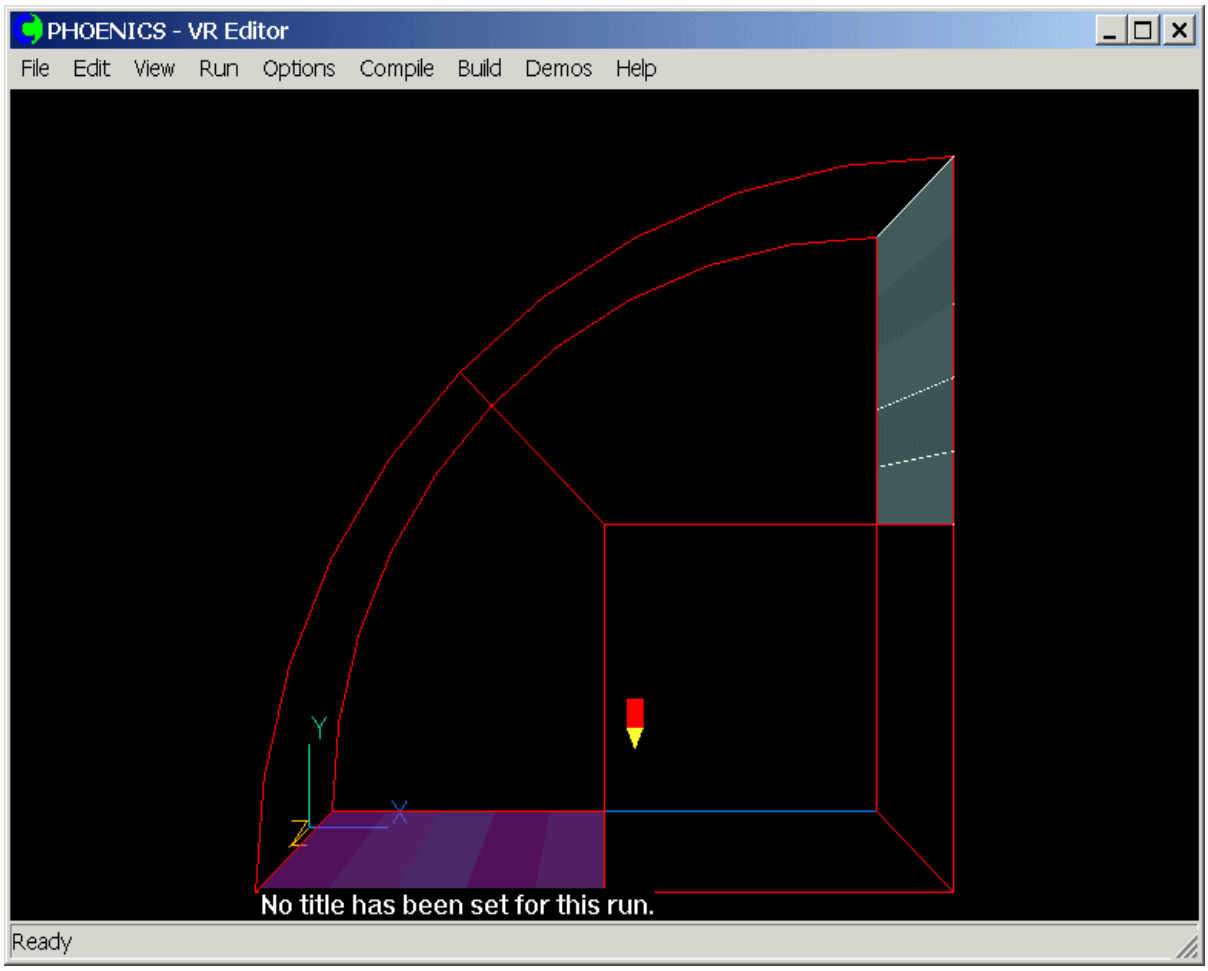
Save and exit Notepad.
The READCO command reads the three block files and assembles them. The MPATCH commands link the three blocks.
These are the only operations which cannot be performed from a dialog box.

- Allow VR-Editor to re-load the Q1. This will display your multi-block domain!!




## Setting up the remainder of the problem

- Click on **Menu** on the hand-set. Click on **Models**. Turn **Solution for velocities and pressure** ON.
- Click on **Numerics**. Set **Total number of iterations** to 50.
- Click on **Relaxation control**. Set the relaxation time for U1 and V1 to 1.0.
- Click **Previous panel**, **Top menu** and **OK** to quit the Main menu.
- Create the inlet: click on **Obj** on the hand-set. Change the **Current block number** to 2. Set the **Isize** to 5 and the **Jsize** to 0. Click on **Type**, and select **Inlet**. Click on **Attributes**, and set the **Y direction** velocity to 1.0 m/s. Click **OK** to close the Attributes dialog, and **OK** to create the new inlet.
- Create the outlet: click on **Obj** on the hand-set. Change the **Current block number** to 3. Set the **Isize** to 0 and the **Jsize** to 5. Set the **IPos** to 5. Click on **Type** and select **Outlet**. Click **OK** to create the new outlet. The screen image should now show:




## Solution

- Run Earth
- Save as case UGRID4. Should you wish to modify the grid for either block, open case UGRID1, 2 or 3, and enter the mesh generator. On exit, issue the DUMPC command. Re-load UGRID4, and the new grids will be read in. At that point, you may have to re-size and re-position the inlets and outlets.

## Results

- Run VR-Viewer to view the results.

  To view:

  - Vectors - click on the 'Vector toggle' 
  - Contours - click on the 'Contour toggle' 

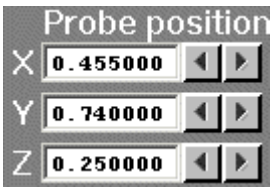
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

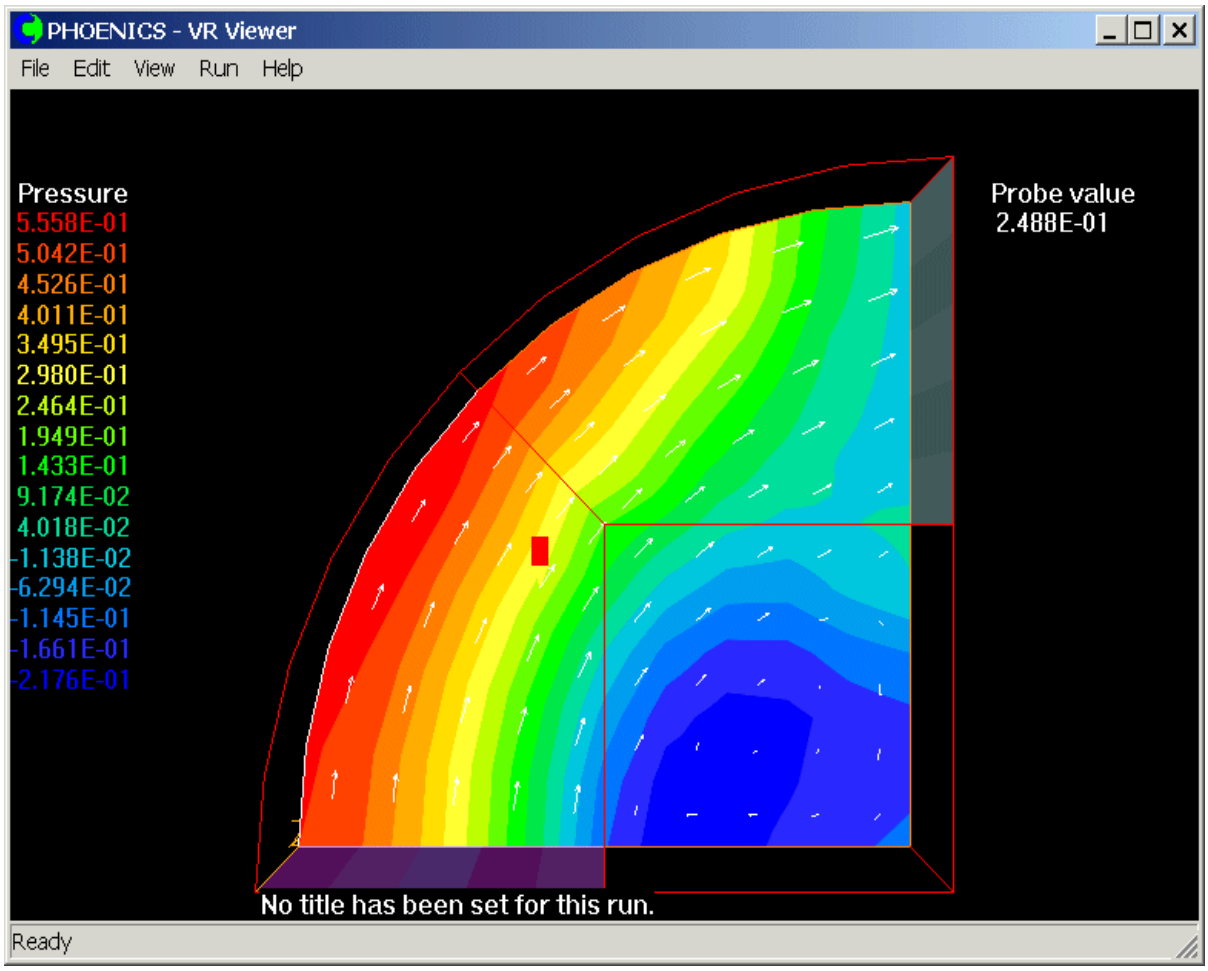
- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons




- To get contours and vectors in all three blocks, click on the **Create slice** toggle when the probe is in Block 1. Now move the probe up into Block 2. Click the **Create slice** toggle again. Move the probe into Block 3. The saved slices will display the contours or vectors selected for Block 3.

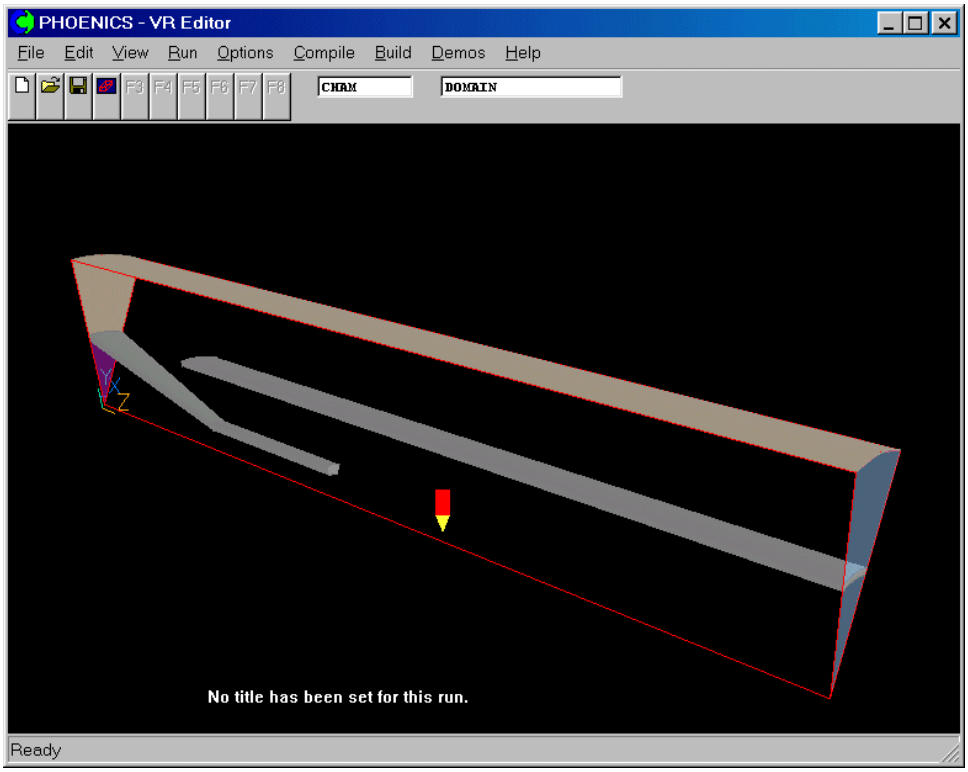

PHOENICS - VR Viewer
File Edit View Run Help
Pressure
5.558E-01
5.042E-01
4.526E-01
4.011E-01
3.495E-01
2.980E-01
2.464E-01
1.949E-01
1.433E-01
9.174E-02
4.018E-02
-1.138E-02
-6.294E-02
-1.145E-01
-1.661E-01
-2.176E-01
Probe value
2.488E-01
No title has been set for this run.
Ready

Home Page

# WORKSHOP - 2D Axi-symmetric Jet Pump

## Polar Co-ordinates

This example shows how to set up and run the simulation of a Jet Pump using VR in cylindrical-polar coordinates. It shows how to rotate objects. The flow is two-dimensional and its geometry is shown below. Velocity and pressure will be solved.




## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*

Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

### **Set the domain size, and activate solution of variables**:

Click on 'Main Menu' and then on 'Geometry'.

Click on Co-ordinate system 'Cartesian', and select 'Cylindrical-polar'. In Cylindrical-polar co-ordinates, X is always the angle, measured in radians, Y is always the radius in metres, and Z is always the axis, also in metres.

Change the X-Domain Size to 0.5 radians.

Leave the Y-Domain size at 1.0 m.

Change the Z-Domain Size to 5.0 m.

Set the number of cells:

- 1 cell in the X-direction,
- 30 cells in the Y-direction,
- 60 cells in the Z-direction.

Click 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Click on 'solution for velocities and pressure' to set it to ON.

Select the KECHEN turbulence model in 'Turbulence models'.

Click on 'Top menu' and then on 'OK'.

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry. Use the movement control buttons to get a view similar to that shown at the start of the tutorial.

**Create the fluid inlet:**

Click on 'New Object'.

Change name to INLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.5

Ypos: 0.0 Ysize: 0.45

Zpos: 0.0 Zsize: 0.0

Define Type: Inlet.

Click on 'Attributes' and set velocity in Z-direction to 5.0 m/s.

Click on 'OK' twice to exit from the Object Dialogue Box.

**Create the fluid outlet:**

Click on 'New Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.5

Ypos: 0.0 Ysize: 0.5

Zpos: 5.0 Zsize: 0.0

Define Type: Outlet.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper secondary inlet** (as an opening):

Click on 'New Object'.

Change name to OPEN1 (which stands for first opening).

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.5

Ypos: 0.53 Ysize: 0.47

Zpos: 5.0 Zsize: 0.0

Define Type: Outlet.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper-left boundary wall:**

Click on 'New Object'.

Change name to WALL1.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.5

Ypos: 0.45 Ysize: 0.55

Zpos: 0.0 Zsize: 0.0

Define Type: Plate.

Click on 'OK' twice to exit from the Object Dialogue Box.

**Create the upper inner horizontal wall:**

Click on 'New Object'.

Change name to WALL2.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.5

Ypos: 0.5 Ysize: 0.03

Zpos: 1.0 Zsize: 4.0

Define Type: Blockage.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper angled wall:**

Click on 'New Object'.

Change name to BAFFLE1.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.5

Ypos: 0.45 Ysize: 0.05

Zpos: 0.0 Zsize: 1.25

Define Type: Blockage

Click on ROTATION, set the 'Angle about X' angle to 15 degrees.

Click on 'OK' twice to exit from the Object Dialogue Box.

**Create the upper horizontal baffle:**

Click on 'New Object'.

Change name to BAFFLE2.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.5

Ypos: 0.13 Ysize: 0.05

Zpos: 1.2 Zsize: 1.0

Define Type: Blockage.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the upper boundary wall:**

Click on ‘New Object’.

Change name to T-WALL (which stands for top wall).

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.5

Ypos: 1.0 Ysize: 0.0

Zpos: 0.0 Zsize: 5.0

Define Type: Plate.

Click on 'OK' to exit from the Object Dialogue Box.

**Set the remaining solution control parameters:**

Click on 'Menu', then on 'Output'.

Change the monitoring point by setting

XMON to 0.25,

YMON to 0.05,

ZMON to 3.0

Click on 'Numerics'.

Set the number of iterations to 300 (this is not sufficient for complete convergence, but it gives a reasonable flow field).

Click on 'Relaxation Control'.

Set the Auto calculation of false-time step factor, SARAH, to 0.005.

Click on 'Previous Panel' to return to 'Numerics'.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

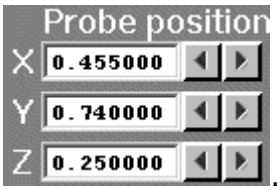
In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.
To view:

- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

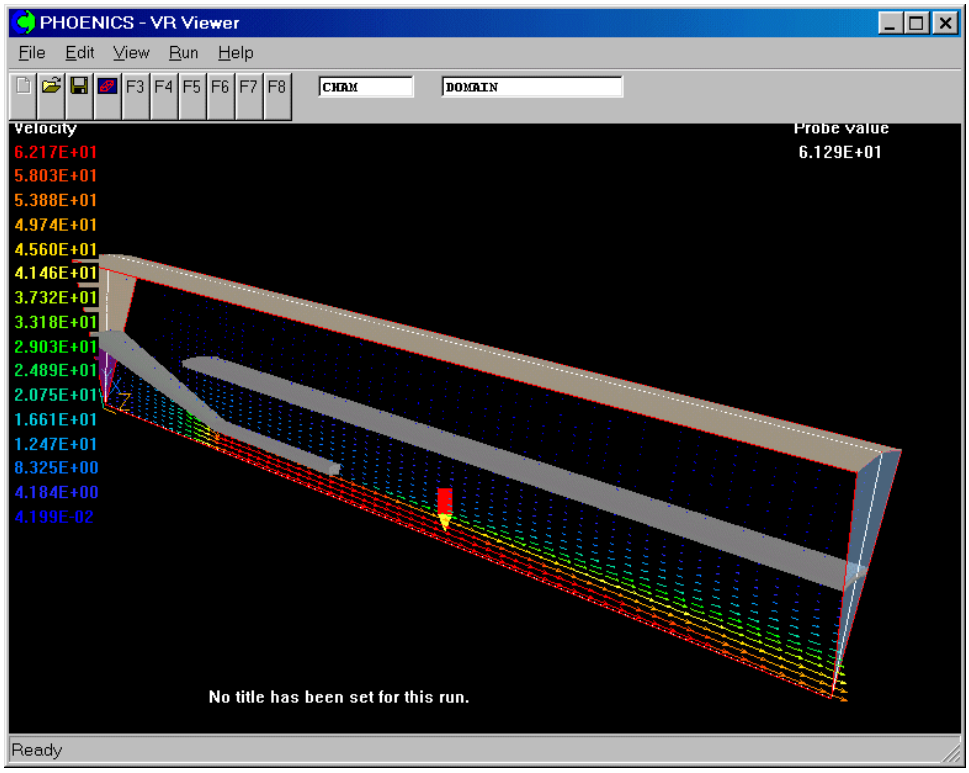
To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' 

To change the direction of the plotting plane, set the slice direction to X, Y or Z 
To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:




## Saving the results.

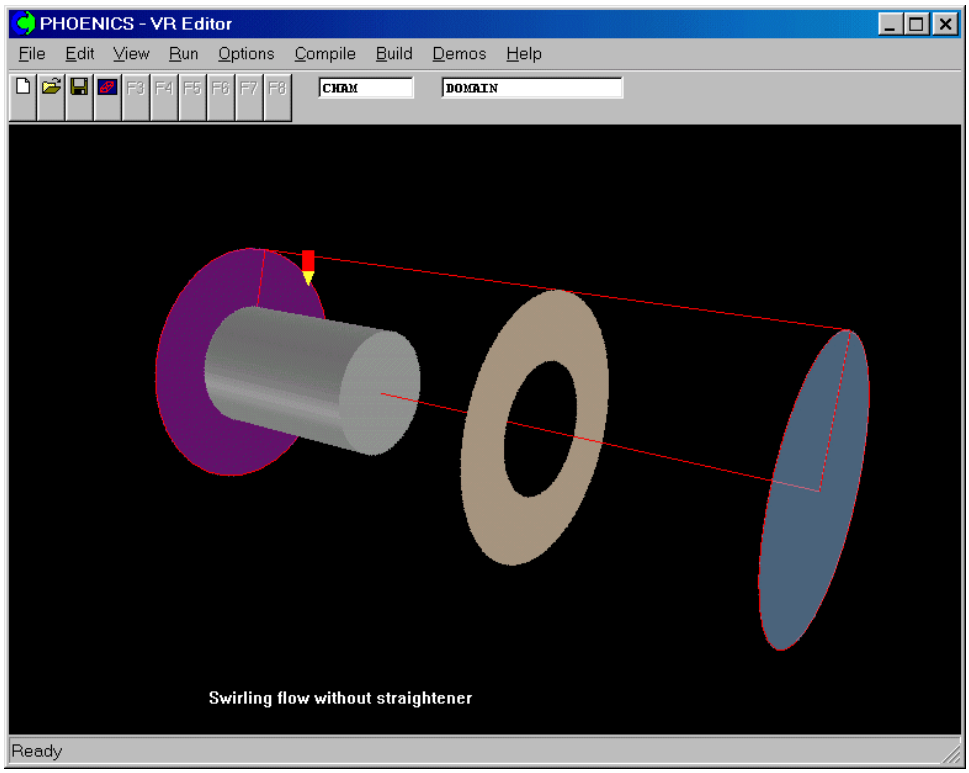
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'JETPUMP' (e.g.) and save as 'POLAR1' (e.g.).

Home Page

# WORKSHOP - 3D Swirling Flow Through an Orifice Plate

## Polar Co-ordinates, Cyclic Boundary Conditions

This example shows how to set up and run the simulation of a swirling flow through an orifice plate using VR in cylindrical-polar coordinates.
The flow is three-dimensional and its geometry is shown below. Velocity and pressure will be solved. Cyclic boundary conditions are used to allow the flow to swirl.




At the inlet, the flow velocity is 1 m/s axially, swirling at 20 radians/s.
As a second exercise, a flow-straightener is introduced down-stream of the orifice in order to remove the swirl component.

## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

### Set the domain size, and activate solution of variables:

Click on 'Main Menu' and then on 'Geometry'.

Click on Co-ordinate system 'Cartesian', and select 'Cylindrical-polar'. In Cylindrical-polar co-ordinates, X is always the angle, measured in radians, Y is always the radius in metres, and Z is always the axis, also in metres.

Change the X-Domain Size to 6.28319 radians (2 PI).

Change the Y-Domain size to 0.2 m.

Leave the Z-Domain Size at 1.0 m.

Set the number of cells:

- 20 cells in the X-direction,
- 15 cells in the Y-direction,
- 15 cells in the Z-direction.

Click 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Click on 'solution for velocities and pressure' to set it to ON.

Select the KECHEN turbulence model in 'Turbulence models'.

Click on 'Sources'. Click on 'Velocity' next to 'U values are', and select 'Angular Velocity'. From now on, all references to U velocity inlet and initial values are interpreted as radians per second.

Click on 'All slabs off' next to 'Cyclic boundaries', and select 'All slabs on'. This allows fluid to pass freely through the boundaries at $X=0$ and $X=X_{max}$.

Click on 'Top menu' and then on 'OK'.

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry. Use the movement control buttons to get a view similar to that shown at the start of the tutorial.

**Create the fluid inlet:**

Click on 'New Object'.

Change name to INLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 6.28319

Ypos: 0.1 Ysize: 0.1

Zpos: 0.0 Zsize: 0.0

Define Type: Inlet.

Click on 'Attributes' and set velocity in Z-direction to 1.0 m/s.

Set the X-direction velocity to 20 radians/s.

Click on 'OK' twice to exit from the Object Dialogue Box.

**Create the fluid outlet:**

Click on 'New Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 6.28319

Ypos: 0.0 Ysize: 0.2

Zpos: 1.0 Zsize: 0.0

Define Type: Outlet.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the central blockage:**

Click on 'New Object'.

Change name to CENTRE.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 6.28319

Ypos: 0.0 Ysize: 0.1

Zpos: 0.0 Zsize: 0.3

Define Type: Blockage.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the orifice plate:**

Click on 'New Object'.

Change name to ORIFICE.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 6.28319

Ypos: 0.1 Ysize: 0.1

Zpos: 0.6 Zsize: 0.0

Define Type: Plate

Click on 'OK' to exit from the Object Dialogue Box.

**Set the remaining solution control parameters:**

Click on 'Menu', then on 'Output'.

Change the monitoring point by setting

IXMON to 1,

IYMON to 12,

IZMON to 2

Click on 'Numerics'.

Set the number of iterations to 300.

Click on 'Relaxation Control'.

Set the Auto calculation of false-time step factor, SARAH, to 0.005.

Click on 'Previous Panel' to return to 'Numerics'.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.
To view:

- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

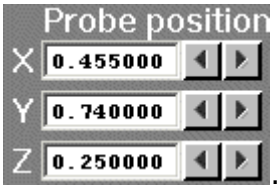
To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .

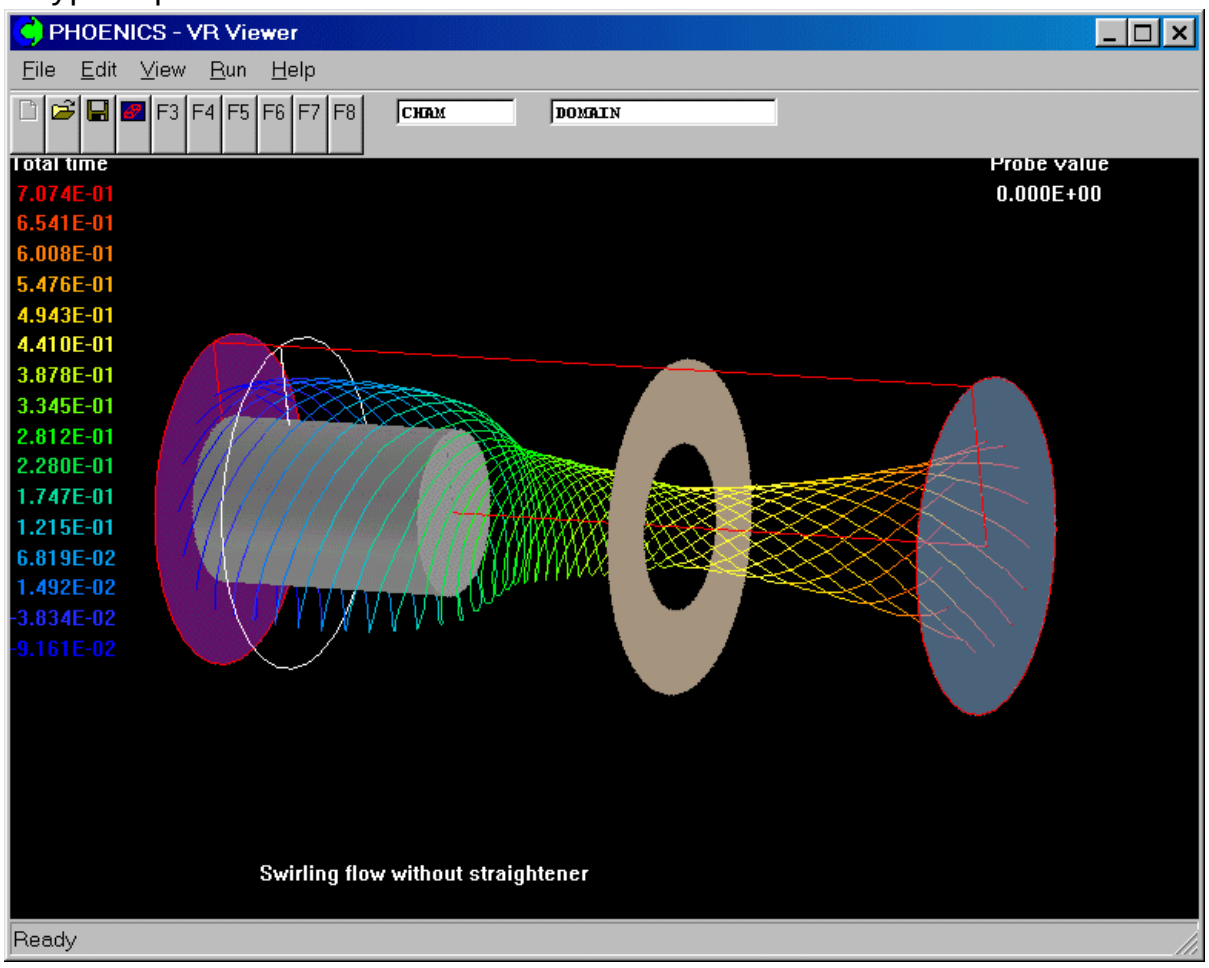
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' 

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:



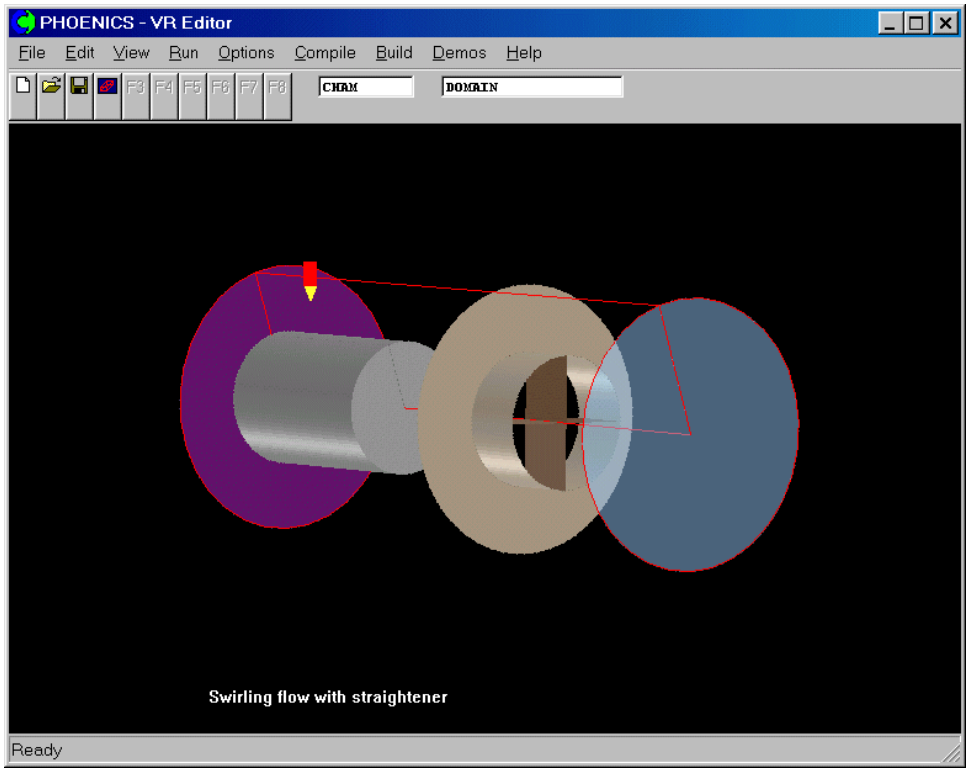
To create this plot, right-click on the 'Create Streamline' button. In the Stream Options dialog, toggle the 'Streamline start' through to 'Around a circle'. Set the circle radius to 0.15. Click 'OK' to close the dialog. Click on 'Z' to set the Z-plane as the current plotting plane. Move the probe to the axis by clicking on 'Y-position down' until the Y position is shown as 0.0. Now left-click on 'Create streamline'. 15 streamlines will be started, evenly spaced around a circle of radius 0.15, in the current plotting plane, centred on the probe.

### Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'SWIRLER' (e.g.) and save as 'CASE1' (e.g.).

## Adding the Flow-Straightener

We will now add five plates to create the flow-straightener shown below.



## Within VR Editor.

Click on 'Run - Pre processor - GUI Pre processor (VR-Editor)' to return to the Editor.

**Create the first plate:**

Click on 'New object'.

Change name to PLATE1.

Set POSITION and SIZE of object as:

Xpos: 0.314 Xsize: 0.0

Ypos: 0.0 Ysize: 0.1

Zpos: 0.6 Zsize: 0.1

Define Type: Plate

Click on 'OK' to exit from the Object Dialogue Box.

**Create three more plates:**

Click on 'Duplicate using array'.

Set Dimension to 4, and Pitch to 1.5708 (2PI/4) in X. Click 'OK'.

**Create the outer plate:**

Click on 'New object'.

Change name to PLATE5.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 6.28319

Ypos: 0.1 Ysize: 0.0

Zpos: 0.6 Zsize: 0.1

Define Type: Plate

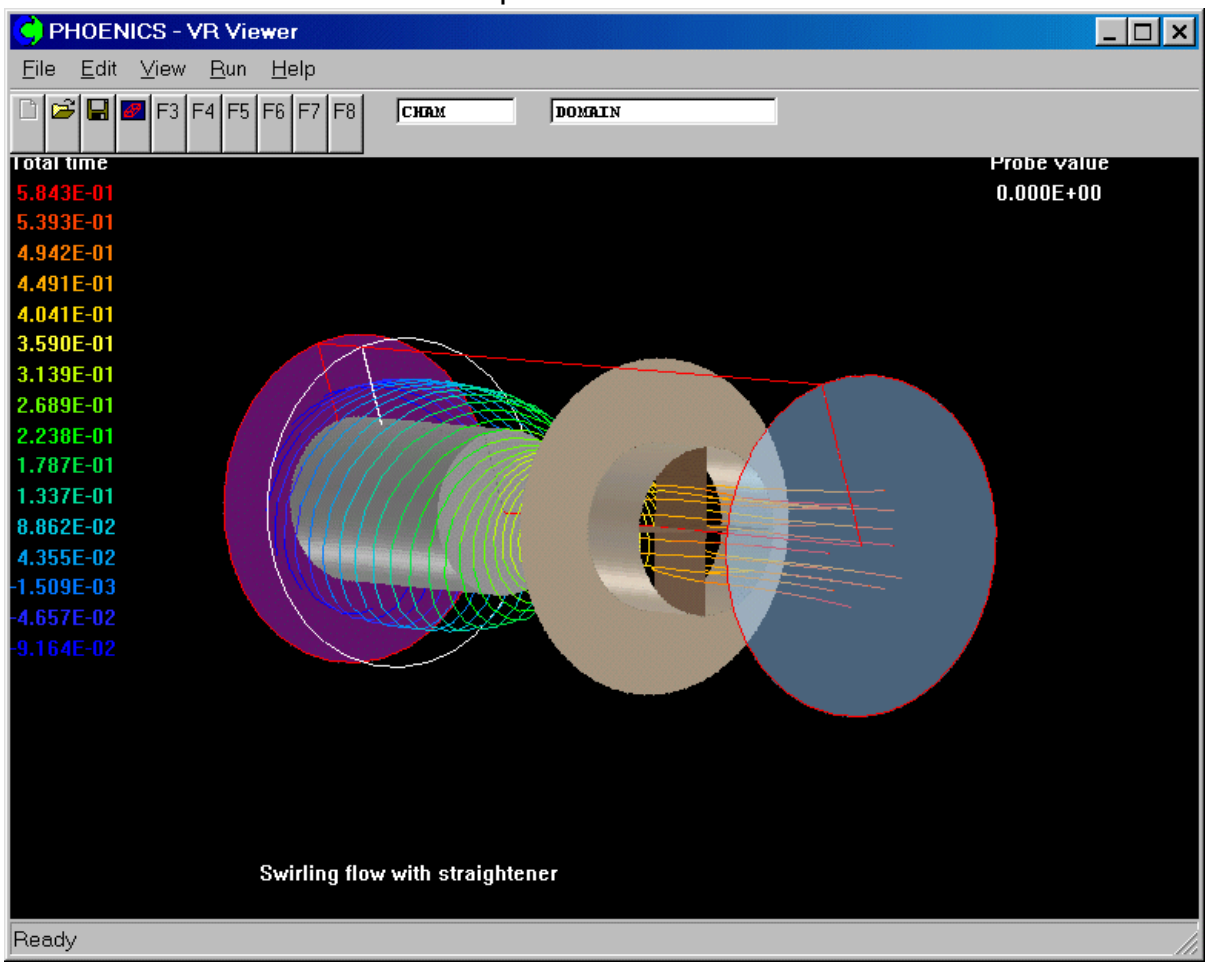
Click on 'OK' to exit from the Object Dialogue Box.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.
Recreate the same streamline plot as before.



Note how the swirl at the exit is much reduced. As an experiment, you may care to see what happens if the outer plate is removed.

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', and save as 'CASE2' (e.g.) in the SWIRLER folder.

Home Page

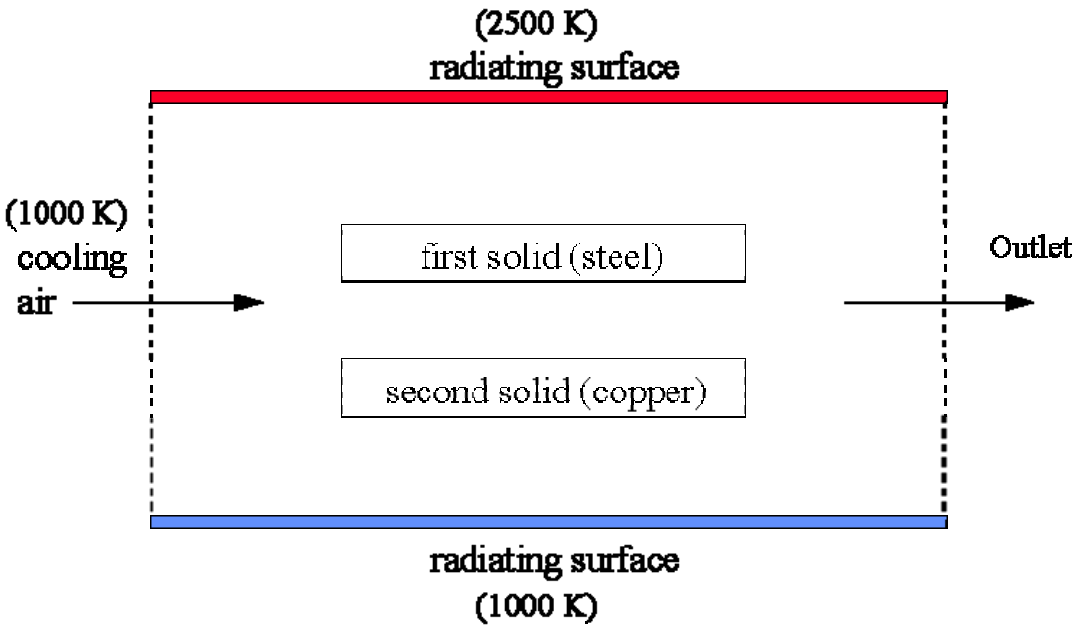
# WORKSHOP - IMMERSOL

# Radiative Heat exchange in a duct flow

This workshop calculates the heat exchange in a duct flow with fixed wall temperatures. Two solid plates of different materials are placed in the duct.

- The flow is steady and turbulent
- TEM1 and T3 are solved for

The flow domain is shown below:-




## Accessing PHOENICS-VR

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*

Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation .

## Within VR Editor.

**Set the domain size, and activate solution of the required variables**:

Click on ‘Main menu’

Click on ‘Geometry’

Change the domain size under z-direction to 0.5 m

Change the Number of cells in the x- and z-directions to 30

Click on 'OK' to close the Grid mesh settings dialog

Click on ‘Models’ and turn ‘Solution for velocities and pressure’ to ON

Click Energy Equation to ON, select Temperature and click on OK

Click on Turbulence models (LAMINAR), select KEMODL and click on OK

Click Radiation models to ON and select IMMERSOL. Click on OK

Click on IMMERSOL settings and:-

Enter 1.0 for the Absorption coefficient

Enter 0.0 for the Scattering coefficient

Switch the ‘Store of radiative energy fluxes’ to ON

Click on ‘Previous panel’

**Set material properties for the fluid**

Click on ‘Properties’. Click on domain material dialogue box/Gases/OK/2 Air using Ideal Gas Law, STP/OK

Click on Top menu/OK

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

**Create INLET object**

Click on ‘New Object’. Change Name to Inlet

Change the following to:

XPos: 0.0 XSize: 0.0

YPos: 0.0 YSize: 1.0

ZPos: 0.0 ZSize: 0.5

Click on Type dialogue box and select INLET. Click on OK

Click on Attributes and enter 727 ° C for the Temperature, 1.0 m/s for the x- direction velocity. Click on OK/OK

**Create OUTLET object**

Click on ‘New Object’. Change Name to Outlet

Change the following to:

XPos: 1.0 XSize: 0.0

YPos: 0.0 YSize: 1.0

ZPos: 0.0 ZSize: 0.5

Click on Type dialogue box/OUTLET/OK

Click on Attributes and change Temperature to 727 ° C

Click on OK/OK

**Create Lower WALL object**

Click on ‘New Object’. Change Name to Walll

Change the following to:

XPos: 0.0 XSize: 1.0

YPos: 0.0 YSize: 1.0

ZPos: 0.0 ZSize: 0.0

Click on Type dialogue box/PLATE/OK

Click on Attributes.

Click on 'Adiabatic', and select 'Surface temperature', 'OK'. Set the surface temperature value to 727.

Set the emissivity to 0.9.

Click on OK/OK

**Create upper WALL object by duplication from lower wall**

Click on the Duplicate object button and use the Z Position button to move the duplicated

object to the top of the domain. Click on the object

Change the name to Wallh.

Click on Attributes, and set the surface temperature to 2227.

Click on OK/OK

**Create upper internal blockage**

Click on ‘New Object’. Change Name to PLT1

Change the following to:

XPos: 0.3 XSize: 0.4

YPos: 0.0 YSize: 1.0

ZPos: 0.35 ZSize: 0.02

Click on Attributes.

Click on the Types dialogue box/Solids/OK

Choose ‘111 STEEL at 27 deg C’/OK

Set emissivity to 0.9

Click on OK/OK

**Create lower internal blockage by duplication from upper**

Click on Duplicate object button and on the object which appears at the origin

Change Name to PLT2

Change the following to:

XPos: 0.3 ZPos: 0.13

Click on Attributes.

Click on the Material dialogue box

Choose ‘103 COPPER at 27 deg C’/OK/OK

**Set remaining solution control parameters**

Click on 'Main Menu'

Click on ‘Initialisation’

Change FIINIT Value under TEM1 to 727

Click on ‘Numerics’

Change ‘Total number of iterations to 200

Click on Top Menu/OK

Adjust the Monitoring Position to the center of the domain using the Z Position button.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files.

To view:

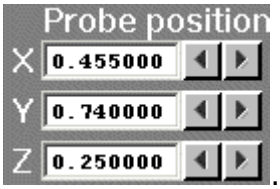
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

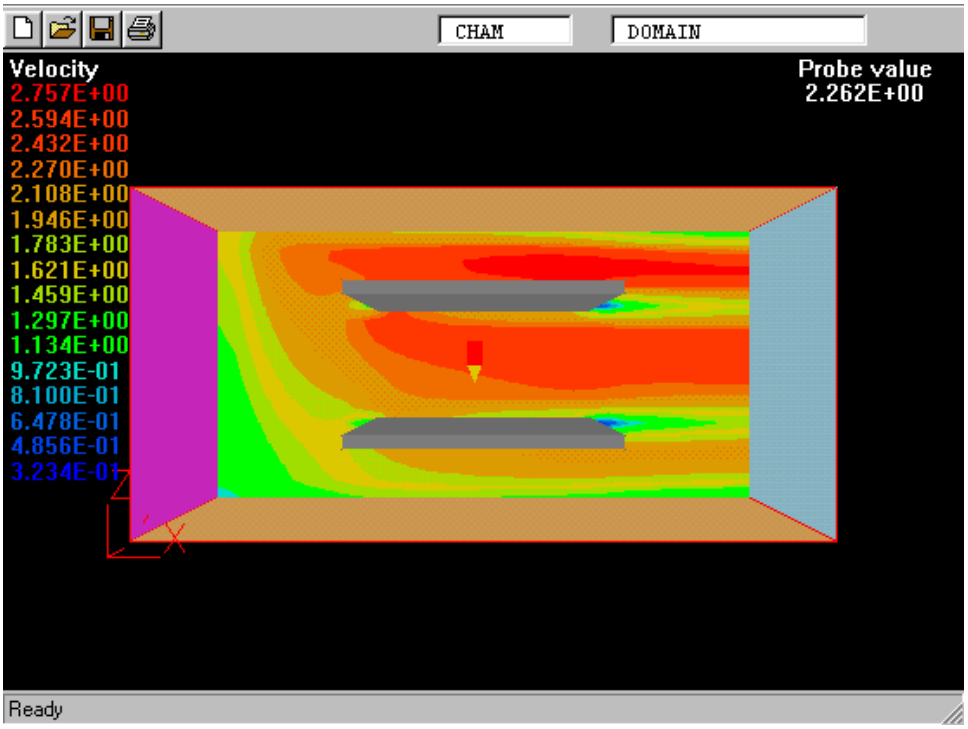
To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons



.

A typical plot from this case is:




## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'IMMERSOL ' (e.g.) and save as 'CASE1' (e.g.).

Home Page

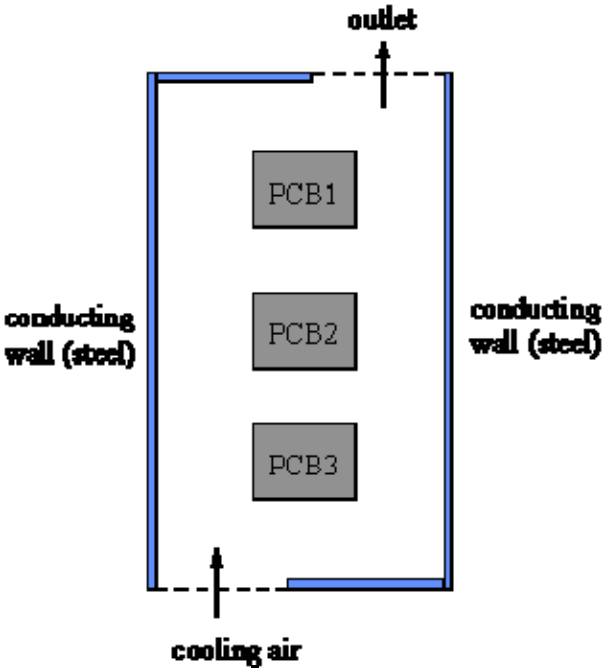
# WORKSHOP - IMMERSOL

# Radiative Heat exchange in an Electronic Box

This workshop calculates the heat exchange in an electronic box which has three printed circuit boards (PCBs) within the domain. They each radiate 50 W to the surroundings. The walls of the box are made of heat-conducting material.

- The flow is steady and turbulent, the k-E turbulence model is employed.
- TEM1 and T3 are solved for.

The flow domain is shown below:-




## Accessing PHOENICS-VR

*From the system level:*

To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation .

## Within VR Editor.

### Set the domain size, activate solution of variables and set physical properties

Click on ‘Main menu’

Click on ‘Geometry’

Change the domain size under x-direction to 0.5 m

Change 'Number of cells' in the x- and z-directions to 30

Click on 'OK' to close the Grid mesh settings dialog

Click on ‘Models’ and turn ‘Solution for velocities and pressure’ to ON

Click on Energy Equation dialogue box, select Temperature and click on OK

Click on Turbulence models (LAMINAR), select KEMODL and click on OK

Click on Radiation models dialogue box and select IMMERSOL. Click on OK

Click on IMMERSOL settings and:-

Enter 1.0 for the Absorption coefficient

Enter 0.0 for the Scattering coefficient

Switch the ‘Store of radiative energy fluxes’ to ON

Click on ‘Previous panel’

Click on ‘Properties’.

Click on Domain material box/Gases/OK/2 Air using Ideal Gas Law, STP/OK.

Click on Top menu/OK

Click 'Reset' on the Movement control panel, then 'Fit to window' to re-scale the view to fit the geometry.

**Create INLET object**

Click on ‘New Object’. Change Name to Inlet

Change the following to:

XPos: 0.0 XSize: 0.2

YPos: 0.0 YSize: 1.0

ZPos: 0.0 ZSize: 0.0

Click on Type dialogue box and select INLET. Click on OK

Click on Attributes and enter 20 ° C for the Temperature, 0.5 m/s for the z- direction velocity. Click on OK/OK.

**Create OUTLET object**

Click on ‘New Object’. Change Name to Outlet

Change the following to:

XPos: 0.3 XSize: 0.2

YPos: 0.0 YSize: 1.0

ZPos: 1.0 ZSize: 0.0

Click on Type dialogue box/OUTLET/OK.

Click on Attributes and change Temperature from 'In-cell' to 'User-set', and set $T_{ext}$ to 20 ° C.

Click on OK/OK.

**Create left wall, WWALL**

Click on ‘New Object’. Change Name to Wwall

Change the following to:

XPos: 0.0 XSize: 0.002

YPos: 0.0 YSize: 1.0

ZPos: 0.0 ZSize: 1.0

Click on Attributes. Click on the Types dialogue box/Solids/OK

Choose ‘111 STEEL at 27 deg C’/OK

Set emissivity to 0.9.

Click on OK/OK.

**Create right wall, EWALL, by duplication from WWALL**

Click on Duplicate object button.

Use X Position button to move copied object to the right hand edge of the domain.

Click on the copied object.

Change Name to Ewall

**Create the top wall, HWALL**

Click on ‘New Object’. Change Name to Hwall.

Change the following to:

XPos: 0.0 XSize: 0.3

YPos: 0.0 YSize: 1.0

ZPos: 0.998 ZSize: 0.002

Click on Attributes.

Click on the Types dialogue box/Solids/OK

Choose ‘111 STEEL at 27 deg C’/OK

Set the emissivity to 0.9

Click on OK/OK.

**Create the lower wall, LWALL, by duplication from HWALL**

Click on Duplicate object button and click on the object which appears at the origin (object B6). Change Name to Swall.

Change the following to:

XPos: 0.2 XSize: 0.3

Click on OK.

### Create bottom pcb, PCB3

Click on ‘New Object’

Change Name to PCB3 (which stands for Printed Circuit Board 3).

Change the following to:

XPos: 0.2 XSize: 0.1

YPos: 0.0 YSize: 1.0

ZPos: 0.15 ZSize: 0.1

Click on Type dialogue box and select PCB. Click on OK

Click on Attributes, then Adiabatic, select Fixed heat Flux, and click OK. Set the heat source to 50.0 W.

Set the emissivity to 0.8.

Click on OK/OK.

### Create the remaining PCB objects by Arraying the first

Click on the Array button

Change the following to:

| | Dimension | Pitch |
|---|---|---|
| X | 1 | 0.0 |
| Y | 1 | 0.0 |
| Z | 3 | 0.3 |

Click on OK.

Click on Top Object twice and change Name to PCB1.

Click on OK.

Click on Middle Object twice and change Name to PCB2.

Click on OK.

### Set remaining solution control parameters

Click on ‘Main menu’.

Click on ‘Initialisation’.

Change FIINIT Value under TEM1 to 20. (Click on > arrow to get there).

Click on ‘Numerics’

Change ‘Total number of iterations to 200.

Click on ‘Output’. Change monitor point to:

IXMON=16 IYMON=1 IZMON=16 (a point inside PCB2).

Click on Top menu/OK.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) . Click 'OK' on the file names dialog to accept the default files. To view:

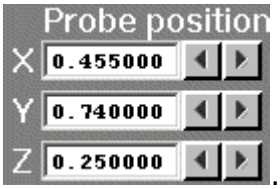
- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

To select the plotting variable:

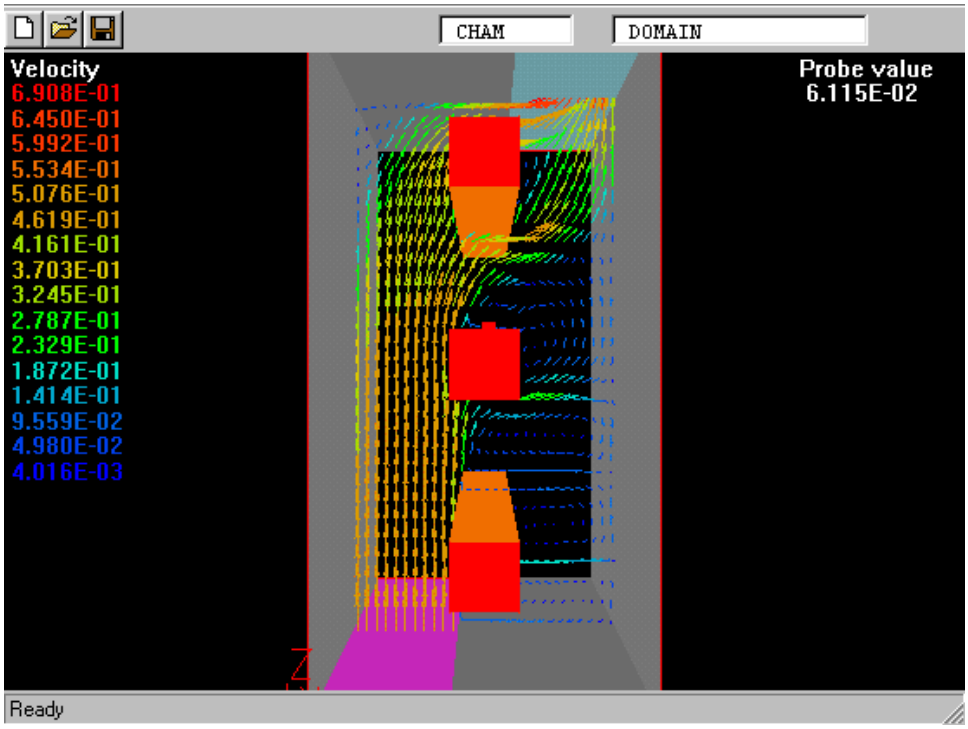
- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable button' . Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons




A typical plot from this case is:




## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'IMMERSOL' (e.g.) and save as 'CASE1' (e.g.).

## Further studies

Do the external walls heat up?
What happens if the absorption coefficient of the fluid is changed to 0.0?
To do this, click on Menu, Models, then Settings next to Immersol.
You will also have to increase relaxation on the temperature equation to obtain convergence. Click on Numerics, and increase the total number of iterations to 300. Click on Relaxation control, then click the > arrow until T3 and TEM1 are visible. Set the DTFALS for T3 and TEM1 to 100.0.
Run Earth, and inspect the results with the Viewer. Save the results as CASE2 in the IMMERSOL folder.

Home Page

# WORKSHOP - Turbulence Modeling

## Overview

The cases below exemplify a variety of turbulence models. The case considered involves flow over a backward facing step at a Reynolds Number of 40,000. The choice of turbulence model can significantly affect the length of the predicted recirculation zone.
The workshop also exemplifies modification of the mesh, and the use of AUTOPLOT to compare different solutions.

# Standard K-ε Model (Core 290)

The KE-EP turbulence model proposed by Harlow and Nakayama in 1968 is by far the most widely-used two-equation eddy-viscosity turbulence model, mainly because the EP was long believed to require no extra terms near walls. The popularity of the model, and its wide use and testing, has thrown light on both its capabilities and its short-comings, which are well-documented in the literature.

## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

### Load Library case 290 and run the base case.

Click on 'File', then 'Load from Libraries'. Enter 290 in the 'case number' data entry box, and click OK.

Click on 'Reset View parameters' then View +Z to get the view exactly along the Z axis.

Click on 'Main menu', 'Models'. The selected turbulence model is the standard high-Reynolds number k-$\varepsilon$ model.

Click on 'Output', and toggle Monitor display from ASCII to GRAPHICS. Set the Monitor update frequency to 1.

Click on 'Numerics'. Set the Total number of iterations to 200, and the Global convergence criterion to 0.01%.

Click on 'Top menu' and 'OK'.

### Increase the number of cells in the X direction behind the step.

Click on the 'mesh toggle' button on the hand-set. The grid will now be drawn. The blue lines are 'ordinary' grid lines, the red lines are region boundaries. The grid can be modified within each region.

Click on the image of the domain, just to the right of the step, in X region 2. In the Grid mesh Settings dialog box, set Cells in region to 16 and Power/ratio to 1.3 for X in region 2.

Click OK. You should see the number of cells in X region 2 change.

Click on the image of the domain toward the right-hand end, in X region 3. In the Grid mesh Settings dialog box, set Cells in region to 16 and Power/ratio to 1.3 for X in region 3. Click OK. You should see the number of cells in X region 3 change.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.
To view:

- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

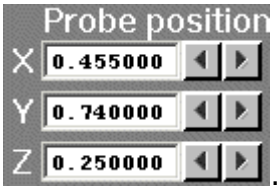
To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button' 
- To select Temperature - click on the 'Select Temperature button' 
- To select any other variable - click on the 'Select a Variable

button' . Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z 

To change the position of the plotting plane, move the probe using the probe position buttons




In particular, look at the length of the recirculation zone and the contours of turbulent viscosity, ENUT.

### Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'TURBUL', select the new folder, and save as 'KE'.

## RNG K-ε Model

The RNG model is a variant of K-ε based on ReNormalisation Group theory. It uses different constants, and has an addition term in the ε equation. The effect of the changes is to reduce the turbulent viscosity in regions of high shear - e.g. in recirculation zones. Hence it should predict a longer recirculation zone, in agreement with experimental evidence.

### Within VR Editor.

Click on 'Run - Pre processor - GUI Pre processor (VR-Editor)' to return to the Editor.
**Change the turbulence model to RNG.**

Click on 'Main menu', 'Models'.

Click on 'KEMODL' next to Turbulence models. From the list of models select KERNG and click 'OK'.

Click on 'Top menu', 'OK'.

### Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK'; to confirm running Earth.

### Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run' then 'VR Viewer', and view the velocity vectors, contours and isosurfaces.

### Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', select the folder 'TURBUL' and save as 'RNG'.

## CHEN-KIM K-ε Model

The CHEN-KIM model is a variant of K-ε based on comparison with experimental data. It uses different constants, and has an addition term in the ε equation. The effect of the changes is to reduce the turbulent viscosity in regions of high shear - e.g. in recirculation

zones. Hence it should predict a longer recirculation zone, in agreement with experimental evidence.

## Within VR Editor.

Click on 'Run - Pre processor - GUI Pre processor (VR-Editor)' to return to the Editor.

**Change the turbulence model to CHEN-KIM.**

Click on 'Main menu', 'Models'.

Click on 'KERNG' next to Turbulence models. From the list of models select KECHEN and click 'OK'.

Click on 'Top menu', 'OK'.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)', and view the velocity vectors, contours and isosurfaces.

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', select the folder 'TURBUL' and save as 'CHEN'.

# Comparing the results in AUTOPLOT

AUTOPLOT is a 2D graph-plotting program. It will read a PHOENICS PHI file, and plot any variable against any coordinate, at a given location.
In this case, we will compare the results of the three turbulence models by plotting the X-directed velocity, U1, along the floor of the duct (IY=1). The point at which U1 is zero is the attachment point.

**Accessing AUTOPLOT**
In the PHOENICS-VR Environment, click on 'Run', 'Post processor' then 'X-Y Graph plotter (AUTOPLOT)'.

**Attaching the data files.**
To tell AUTOPLOT which files are to be plotted, enter the command

```
FILES
```

Enter the name of the first file as TURBUL/KE.PHI and press RETURN. AUTOPLOT will ask for the file type. Enter 5 (for sequential PHI file) and press RETURN. Now enter the name of the second file, TURBUL/RNG.PHI and press RETURN. Finally, enter the last file name, TURBUL/CHEN.PHI and press RETURN twice to signal the end of file attachment.

**Reading the data**
We must now read data from the attached files into AUTOPLOT. We want to plot U1 as the Y axis, X distance as the X axis. We want to pick up the U1 values along the floor, that is in the first row of cells at IY=1.
The command to read this data from the first file (KE.PHI) is:

```
DATA 1 U1 Y 1
```

The command to read this data from the second file (RNG.PHI) is:

```
DATA 2 U1 Y 1
```

The command to read this data from the third file (CHEN.PHI) is:

```
DATA 3 U1 Y 1
```

Each time a DATA command is executed, AUTOPLOT reads the specified values from the specified file, and creates a 'data element' in memory.

**Plotting the data**

The command to plot individual data elements is PLOT n, where n is the number of the data element. PLOT will draw the data in the default colour and linestyle, which is a solid white line.

An alternative means of drawing data elements is the COLj n command. Here n is the element number to be drawn, and j is a colour number to use. The available colour 'numbers' are 0 - 9 and A - F. Colour 0 is black, 1 is white, 2 is dark blue and F is red.

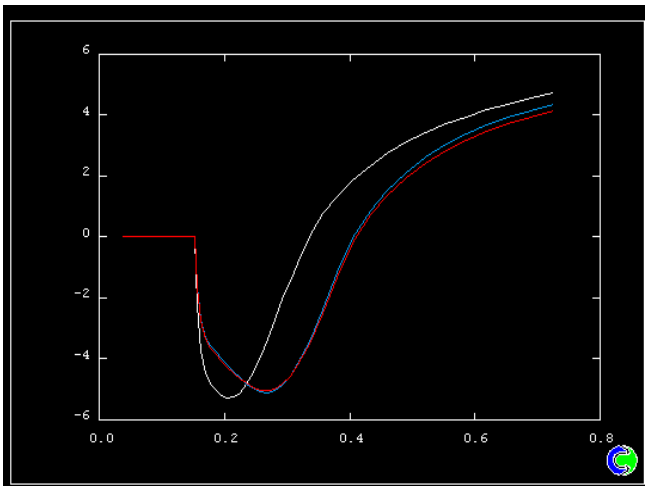
To draw the data from KE.PHI in white, RNG.PHI in light blue and CHEN.PHI in red, enter the commands:

```
PLOT 1

COL4 2

COLF 3
```

At this point, the screen should look like:




It is clear that both RNG and CHEN-KIM predict a longer recirculation zone than the standard model. This is to be expected, as both produce lower turbulent viscosity within the recirculation zone.

# Further Suggestions

If you have time, you might like to try the effect of refining the grid further. (You might need more sweeps to ensure convergence).
Alternatively, you may wish to try the effect of using a higher-order differencing scheme.
In the VR Editor click on Main Menu / Numerics / Differencing Schemes.
Click on Set schemes individually and set:

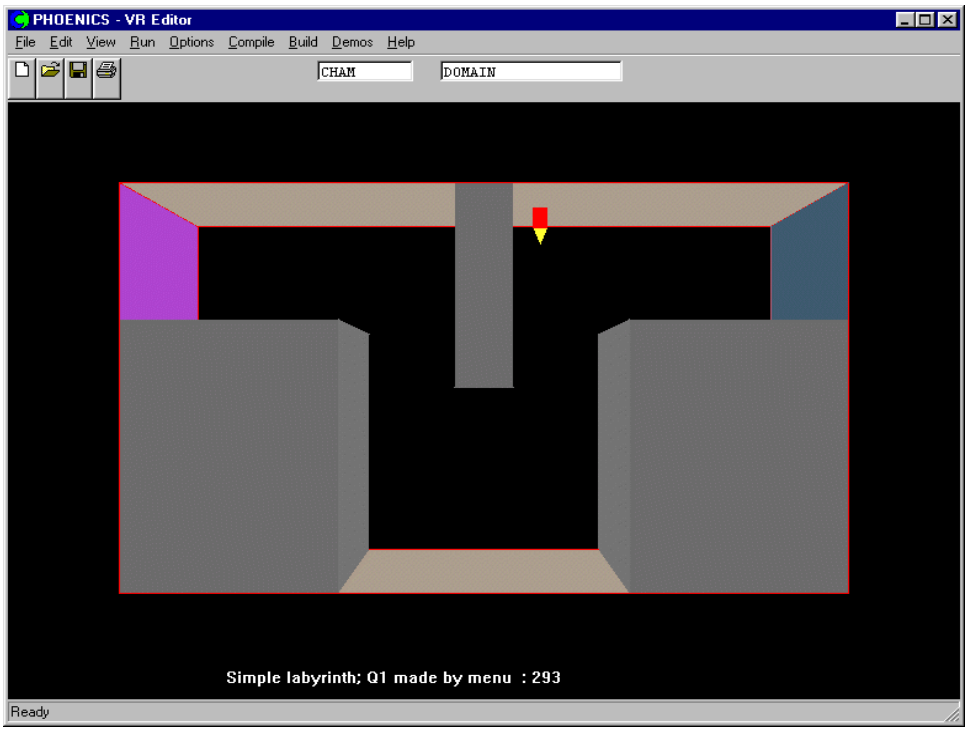
- All velocities to the OSPRE scheme
- KE, EP to the VANL2 scheme

Click on Previous panel/Relaxation control and set the values for U1 and V1 to 1.0E-3.
Click on Previous Panel/Top/OK.
Run Earth, and compare the results with those you already have.

Home page

# WORKSHOP - LVEL Turbulence Model

This workshop shows how the LVEL model can be applied to a case with several blockages, in order to calculate the distance from each cell to the nearest wall. It is based on Library Case 293.




## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with case 293 - click on 'File' then on 'Load from libraries'. Enter 293 in the Case number box, then click on 'OK'; to load the case.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

**Reset the material of the blockages**
The three blockages are made of material 199 - this allows fluid slip. It provides no boundary conditions for the wall-distance calculations.
Click on each blockage in turn, to bring up its Object dialog box. Click on Attributes, and change the material to 198 - Solid with smooth-wall friction.
**Activate the LVEL model**
Click on Main Menu, then on Models. From the list of turbulence models, select LVEL. Click on Top menu and OK to quit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.
To view:

- Vectors - click on the 'Vector toggle' 
- Contours - click on the 'Contour toggle' 
- Streamlines - click on the 'Create Streamline' button 
- Iso-surfaces - click on the 'Iso-Surface toggle' 

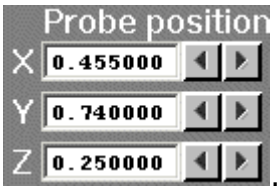
To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .

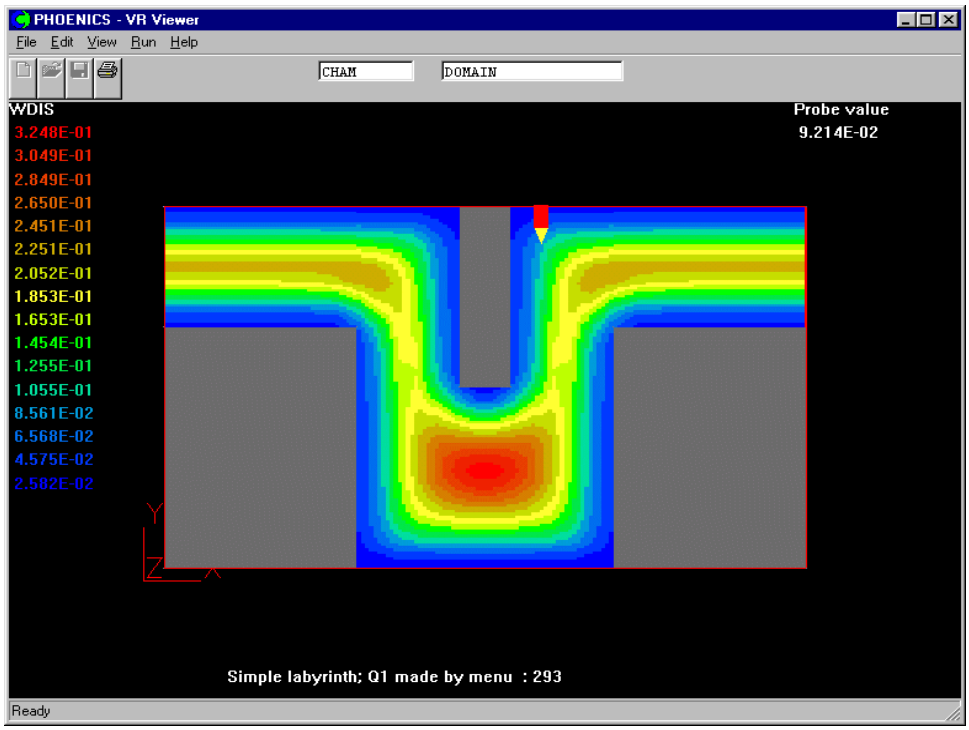
- To select Velocity - click on the 'Select Velocity button' V
- To select Temperature - click on the 'Select Temperature button' T
- To select any other variable - click on the 'Select a Variable button' C. Select DEN1 for density, or ENUL for laminar viscosity values.

To change the direction of the plotting plane, set the slice direction to X, Y or Z X Y Z

To change the position of the plotting plane, move the probe using the probe position buttons




The plot below shows contours of the wall distance variable WDIS.




## Saving the results.

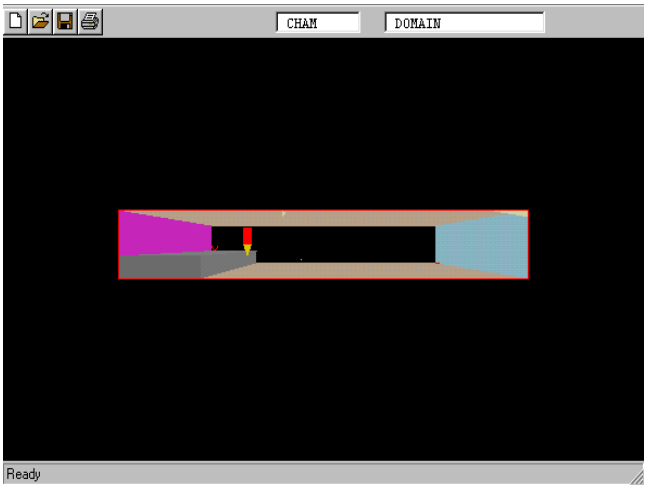
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'LVEL' (e.g.) and save as 'CASE1' (e.g.).

Home Page

# WORKSHOP - Two-Phase Flow (Solids in Air)

## Introduction

The case considered is a 2-phase solids-laden turbulent flow of air over a back-ward facing step. The basic single-phase setup is taken from Core library case 290, which is then modified to include the second, heavy, phase. The geometry is shown below:




The particles will be given a diameter of 0.1 mm and a density of 100 kg/m$^3$. The volume fraction of particles at the inlet will be 10%, and they will be travelling at the same velocity as the carrier gas.

## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*

Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

### Load Library case 290 and run the base, single-phase, case.

Click on 'File', then 'Load from Libraries'. Enter 290 in the 'case number' data entry box, and click OK.

Click on 'Reset View parameters' then View +Z to get the view exactly along the Z axis.

Click on 'Main menu', 'Output', and toggle Monitor display from ASCII to GRAPHICS. Set the Monitor update frequency to 1. Click 'Top menu' and 'OK'.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver', then 'Local solver (Earth)' and 'yes' when prompted to save data to files.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor(VR Viewer)', and view the velocity vectors, contours and isosurfaces.

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'TWOPHS' (e.g.) and save as 'CASE1' (e.g.).

## Within VR Editor.

Click on 'Run - Pre processor - GUI Pre processor (VR-Editor)' to return to the Editor.

**Activate solution of the second phase.**

Click on 'Main menu'.

Click on 'Models'.

Click on 'One_phase', and from the list of multi-phase models select 'IPSA Full'. Click 'OK'.

### Set interphase linkages

Click on 'Settings' next to IPSA_FULL.

Click on Interphase Drag law 'User', and from the list of Drag laws select 'Standard drag curve'. Click 'OK'.

Set the particle diameter to 1.0E-4 m, and the minimum slip velocity to 1.0E-3 m/s.

Click 'Previous Panel'.

### Set properties of the second phase.

Click on 'Properties'.

Click on 'Set Phase 2 properties'.

Set the density of phase 2 to 100 $kg/m^3$.

## Set initial values

Click on 'Initialisation'

Set initial value of U2 to 13 (same as U1)

Set initial value of R1 (phase 1 volume fraction) to 0.9, and that of R2 (phase 2 volume fraction) to 0.1. You will have to use the < > buttons next to Variable to scroll the list of variables.

## Set relaxation parameters

Click on 'Numerics'.

Set the Gloabal convergence criterion to 0.01%.

Click on 'Relaxation control', and set the DTFALS values for U1, U2 V1 and V2 all to 0.001.

Click 'Previous panel', 'Top Menu', 'OK'.

## Set the inlet values for the two phases

Click twice on the inlet object (called INLET). Click on 'Attributes'.

Set the inlet density to 0.9. (This is actually the phase 1 density, 1.0, multiplied by the phase 1 volume fraction, 0.9).

Click on Setting values for 'Phase 1' to switch to Phase 2. Set the inlet density for phase 2 to 10.0 (this is the phase 2 density, 100.0, multiplied by the phase 2 volume fraction, 0.1)

Set the inlet velocity for phase 2 to 13.0 m/s in the X direction.

Click on 'OK', 'OK'.

**Allow the second phase to leave the domain at the outlet.**

Click twice on the outlet object (called OUTLET). Click on 'Attributes'.

Set the Ph 1 Coefficient to 1.0.

Set the Ph 2 Coefficient to 100.0 (this is the phase 1 coefficient multiplied by the phase 2 density).

Click on 'OK', 'OK'.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver', then 'Local solver (Earth)' and 'yes' when prompted to save data to files.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor(VR Viewer)', and view the velocity vectors, contours and isosurfaces. Note how the contour of phase 2 volume fraction, R2, shows a 'shadow' behind the step. The heavy particles do not follow the fluid exactly.

## Saving the results.

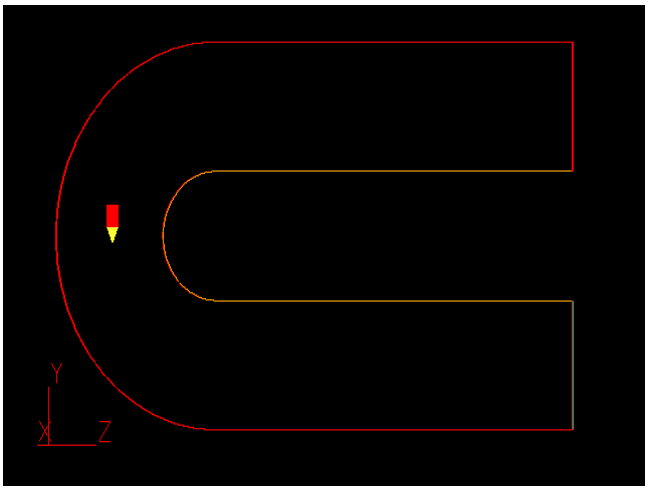
In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'TWOPHS' (e.g.) and save as 'CASE2' (e.g.).

Home Page

# WORKSHOP - Introducing non-standard interphase transport correlations

## Introduction

The case considered is a 2-phase frozen-"snow"-laden gas flow in a U-bend. The basic single-phase setup is taken from BFC library case B527, which is then modified to include the second, heavy, phase. The geometry is shown below:




The snow particles will be given a diameter, $D_p$, of 1.0 mm and a density of 110 kg/m$^3$. The volume fraction of particles at the inlet will be 0.1%, and they will be travelling at the same velocity as the carrier gas, namely 20 m/s.
The inlet temperature of gas is 0°C and snow particles have -23°C. The phase specific heats, $C_{P1}$ and $C_{P2}$, are 5000 and 4000, J/(kgK), correspondingly. The gas thermal conductivity, k, is 0.01 W/(mK) and its kinematic viscosity, $\nu$, is $10^{-5}$, m$^2$/s.
Interphase drag coefficient correlation is given as
$C_d = \min( 200., 24/Re(1.+0.15Re^{0.687}) )$
where:

$Re = V_{slip}D_p/\nu$
with $V_{slip}$ being the relative velocity between the particulate and gas phases.
Interphase heat transfer is controlled by local Nusselt Number correlation as
$Nu = hD_p/k = 2.( 1.+0.1Re^{0.3} )$
where:
h is a heat transfer coefficient, $W/(m^2K)$
The task is to calculate the distribution of phase velocities, temperatures and pressure drop.

## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

### Load Library case B527.

Click on 'File', then 'Load from Libraries'. Enter B527 in the 'case number' data entry box, and click OK.

Double click in the domain and click on 'Delete', 'Yes' to get rid of irrelevant object. Repeat for two others near-wall objects to clean-up the flow field.

### Activate solution of the second phase and enthalpy.

Click on 'Main menu'.

Click on 'Models'.

Click on 'One_phase', and from the list of multi-phase models select 'IPSA Full'. Click 'OK'.

Click 'Energy Equation "OFF"', and from the list select 'Enthalpy'. Click 'OK'.

Click on 'STATIC' next to 'ENTHALPY' to set it 'TOTAL'.

**Set interphase linkages**

Click on 'Settings' next to IPSA_FULL.

Set 'Interphase Coefficient (CFIPS)' GRND and click on 'Store' to set it ON.

Set 'CFIPB' 0.001 - particle diameter in meters.

Change 'Interphase Heat transfer' CINT for 'Variable' H2 1.e20 and for H1 GRND. Click on '>' and '<' to scroll the list of variables.

Click 'Previous Panel'.

**Provide the storage for auxiliary variables.**

Click on 'Page Dn'.

Click on 'Settings' next to 'Solution control/Extra variables'.

Set 'Activate Storage' 'STORE' CD for drag coefficient and click 'Apply'.

Repeat for KOND, NUSS, REYN and VREL to activate the storage for gas thermal conductivity, Nusselt number, Reynolds number and relative phase velocity.

Click on 'Previous panel'.

**Set properties of the first phase.**

Click on 'Properties'.

Set 'Storage' for density to 'ON'.

Set ENUL=1.e-5 for the gas kinematic viscosity and 'Storage' ON.

Click on 'Page Dn'.

Click 'Advanced settings' PIL'

Set $C_{P1}$ = 5000.

Set 'PRNDTL' for 'Variable's R1 = 1.e10 to cut-off phase diffusion.

Set PRNDTL for H1=-2.e-6 which is $-k/C_{P1}$ and

Click on 'Previous Panel'

**Set the Kelvin-to-Celsius and enthalpy-to-temperature parameters for the first phase.**

Click on 'Temperature' NOT SET, from the list of select 'Linear in H' and click 'OK'.

Set $T(H_o)$=-273 and CP=$C_{P1}$=5000. Click 'Storage' to set it 'ON'.

Click on 'Page Dn'

Set 'Reference temp'= 273.

Click on 'Page Up'

**Set properties of the second phase.**

Click on 'Set Phase 2 Properties'.

Set RHO=110 and click on 'Page Dn'

Click 'Advanced settings' PIL'

Set $C_{P2}$ = 4000.

Click on 'Page Dn' or 'Line Dn', as appropriate.

Set 'PRNDTL' for 'Variable's R2 = 1.e10 to cut-off phase diffusion.

Set PRNDTL for H2=1.e10 to cancel the 2nd phase diffusion

Click on 'Page Up'

Click on 'Previous Panel'

**Set the Kelvin-to-Celsius and enthalpy-to-temperature parameters for the second phase.**

Click on 'Temperature' NOT SET, from the list of select 'Linear in H' and click 'OK'.

Set $T(H_o)$=-273 and CP=$C_{P1}$=4000. Click 'Storage' to set it 'ON'.

Set 'Reference temp'= 273.

**Set interface properties.**

Click on 'Set interface properties'

Set 'PHINT' for 'Variable's H1 = 1.25 which is $C_{P1}/C_{P2}$.

**Set initial values**

Click on 'Initialisation'

Set initial value of W1 and W2 to 20

Set initial value of R1 (phase 1 volume fraction) to 0.999, and that of R2 (phase 2 volume fraction) to 0.001. You will have to use the < > buttons next to Variable to scroll the list of variables.

Set initial values of H1 to 1.365E+06 and initial value of H2 to 1.250E+06. These are the inlet enthalpies.

Set value of KOND to 0.01.

Click on 'Apply'

**Set relaxation parameters**

Click on 'Numerics'.

Set 'Total number of iterations' to 400.

Set the Global convergence criterion to 0.1%.

Click on 'Relaxation control', and set the DTFALS values for W1, W2, V1, V2, H1 and H2 all to 0.6.

Click 'Previous panel', 'Top Menu', 'OK'.

**Set the inlet values for the two phases**

Click twice on the inlet object (called IN). Click on 'Attributes'.

Click on 'Phase fluid'

Set the inlet density to 0.999. (This is actually the phase 1 density, 1.0, multiplied by the phase 1 volume fraction, 0.999).

Set Temperature = 0°C.

Set Z-direction velocity to -20.0 m/s.

Click on Setting values for 'Phase 1' to switch to Phase 2.

Click on 'Phase fluid'

Set the inlet density for phase 2 to 0.11 (this is the phase 2 density, 110.0, multiplied by the phase 2 volume fraction, 0.001)

Set inlet temperature to -23°C.

Set the inlet velocity for phase 2 to -20.0 m/s in the Z direction.

Click on 'OK', 'OK'.

**Allow the second phase to leave the domain at the outlet.**

Click twice on the outlet object (called OUTLET). Click on 'Attributes'.

Set the Ph 1 Coefficient to 1.

Set the Ph 2 Coefficient to 110.0 (this is the phase 1 coefficient multiplied by the phase 2 density).

Click on 'OK', 'OK'.

**Make the PLANT settings for phase relative velocity.**

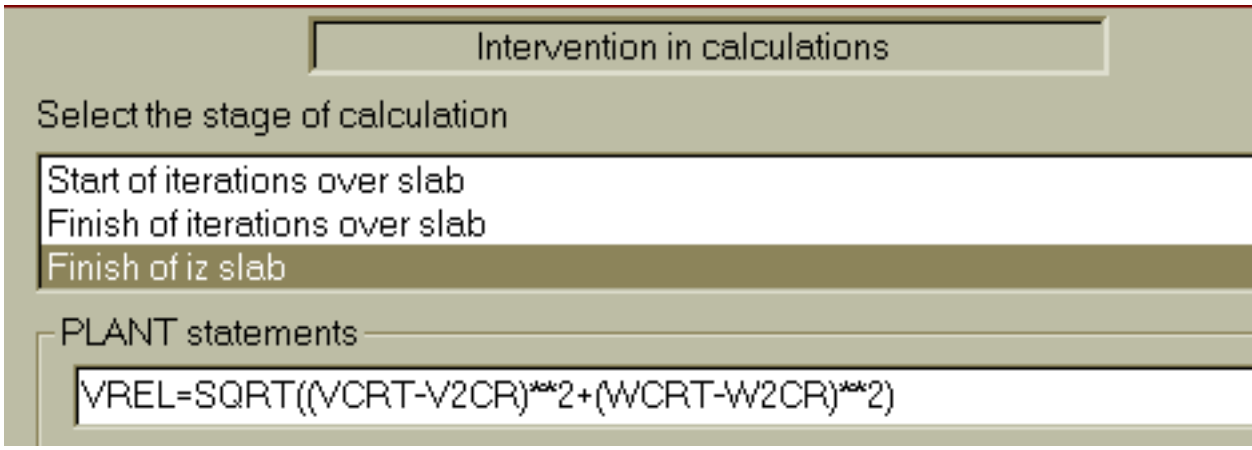
Click on 'Run, Pre processor, Fortran creator (Plant Menu)'.

Select 'Intervention in calculations', then click on 'Add'.

Select 'Finish of IZ-slab'.

Make settings for VREL as shown on the image below.

Click 'OK' to close the window.



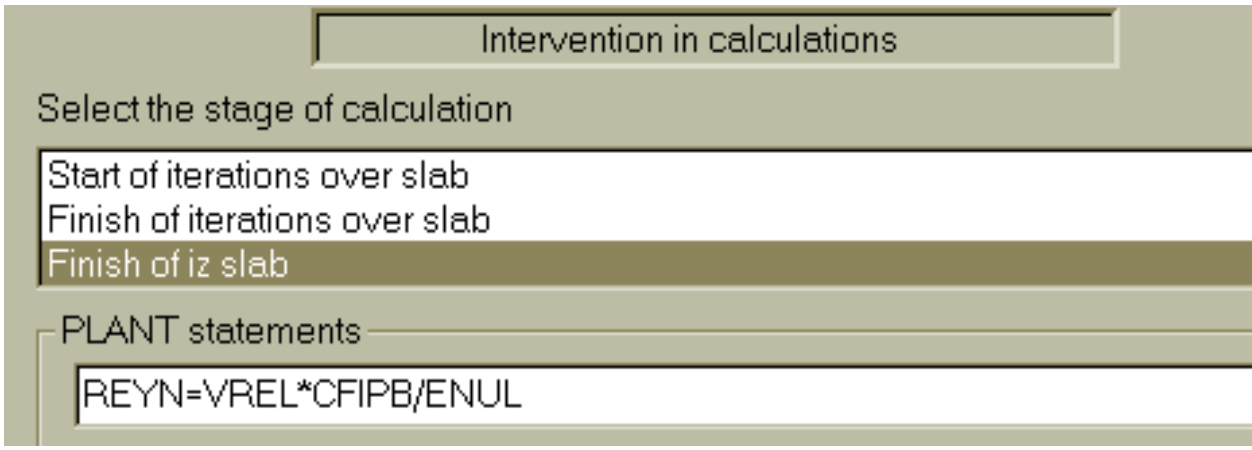

**Make the PLANT settings for particle Reynolds number.**

Click on 'Add'.

Select 'Finish of IZ-slab'.

Make settings for REYN as shown on the image below.

Click 'OK' to close the window.



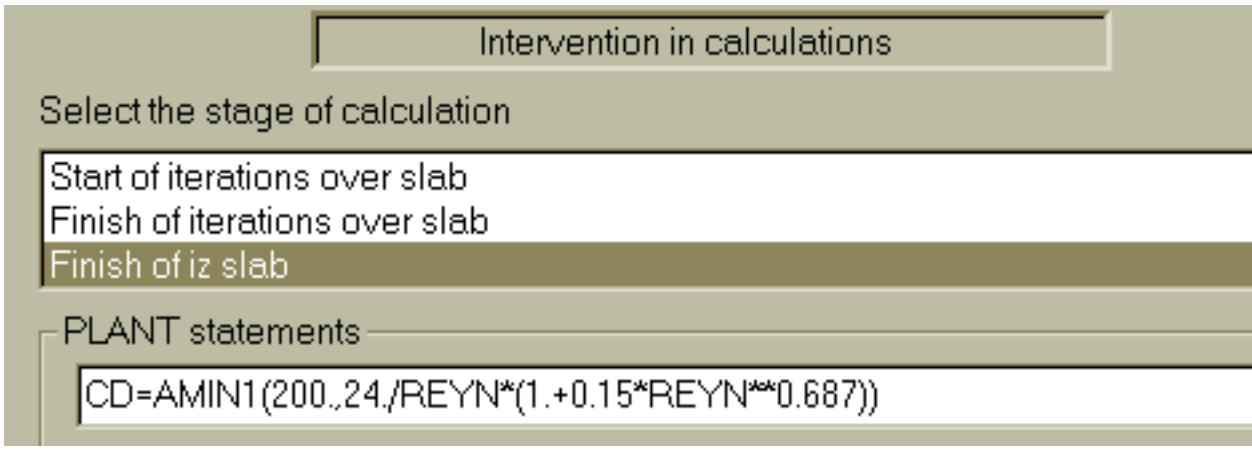

**Make the PLANT settings for particle drag coefficient.**

Click on 'Add'.

Select 'Finish of IZ-slab'.

Make settings for CD as shown on the image below.

Click 'OK' to close the window.



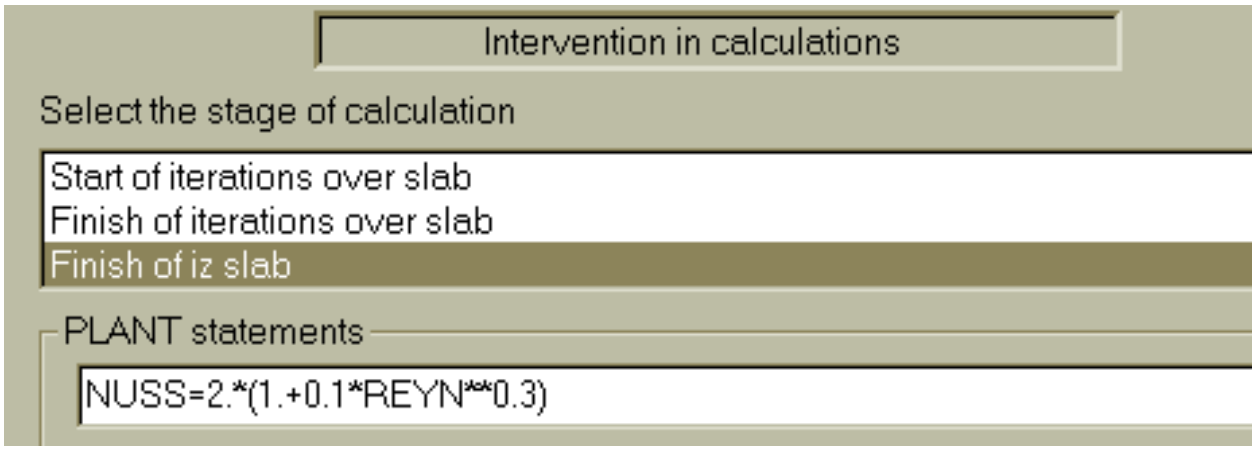

**Make the PLANT settings for particle Nusselt number.**

Click on 'Add'.

Select 'Finish of IZ-slab'.

Make settings for NUSS as shown on the image below.

Click 'OK' to close the window.



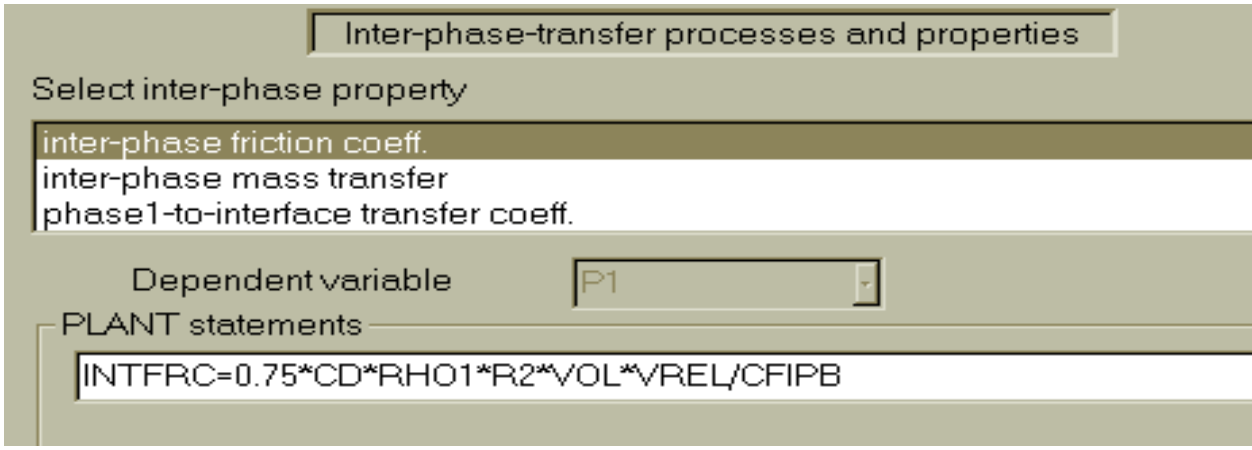

**Make the PLANT settings for interphase friction coefficient.**

Select 'Interphase-transfer processes', then click on 'Add'.

Select ''Interphase friction coefficient'.

Make settings for INTFRC as shown on the image below.

Click 'OK' to close the window.




**Make the PLANT settings for interphase heat transfer coefficient.**

Select 'Interphase-transfer processes', then click on 'Add'.

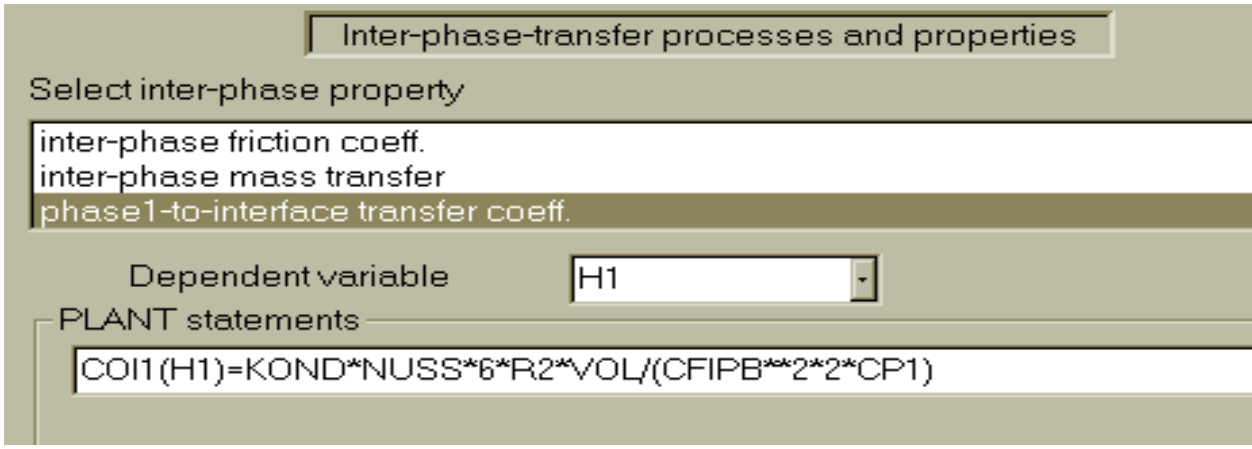
Select 'Phase1-to-interface transfer coefficient'.

Set 'Dependent variable' to H1.

Make settings for COI1(H1) as shown on the image below.

Click on 'OK' to close the window.

Click on 'Exit' then on 'OK'.




## Running the Solver.

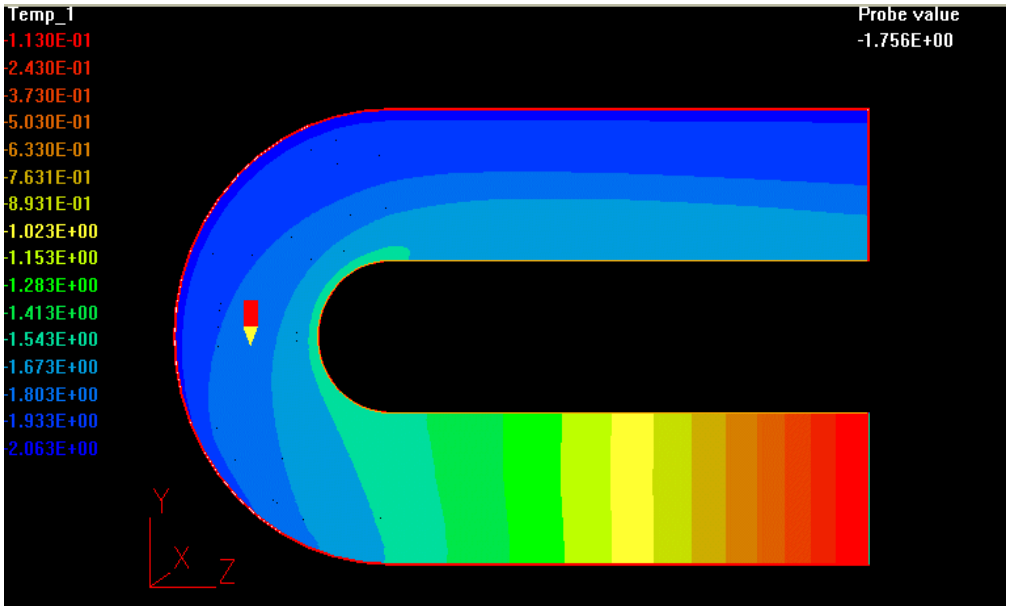
In the PHOENICS-VR environment, click on 'Options', then on 'Run Version'.
For 'Earth', select 'Prompt'.
Click on 'Run', 'Solver' then 'Local solver(Earth)'.
Select 'Private'.
Allow the compilation and re-linking to proceed.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor(VR Viewer)', and view the velocity vectors, contours and isosurfaces. Note how the contour of phase 2 volume fraction, R2, shows a 'shadow' behind the step. The heavy particles do not follow the fluid exactly.

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'TWOPHS' (e.g.) and save as 'CASE2' (e.g.).
The calculated temperature distribution in the 1st phase is shown below:

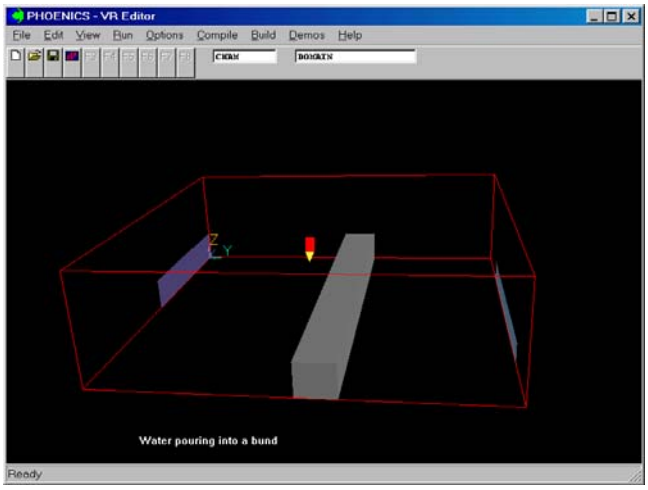



svz/331/0201 jcl/35/0702

Home Page

# WORKSHOP: - Water pouring into a bund by SEM.

This workshop example shows how to set up and run the simulation of 3D transient free surface flow of water in air by Scalar Equation Model in PHOENICS-VR.
The task is to simulate the water pouring into a bund through the bottom slot at the right side wall. The opposite side wall has an opening through which the water leaks out of the bund. Internal obstacle is located on the water path.
It is expected that the water starts to propagate into the bund along the horizontal floor until it encounters the internal beam. It re-directs the water stream until it finally reaches the leak opening over-flowing the obstacle.
The geometry is shown in the figure below.




## Accessing PHOENICS-VR

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*

Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation .

## Within VR Editor.

**Set the domain size, and activate solution of the required variables**:

Click on 'Menu' and then on 'Geometry'.

Change the Z-Domain Size to 0.3 m.

Change the Number of cells to 10 cells in X, Y and Z.

Click on Time dependence STEADY to toggle to TRANSIENT.

Click on 'Time step settings'.

Set:

- Starting time: 0.0
- Duration: 2.0
- Number of steps: 100

Click 'OK' to close the Time step settings dialog, then 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Set the 'Solution for velocities and pressure' ON.

Set the 'Free-surface model' to Scalar Equation.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

**Create the BEAM object**:

Click on 'Object'.

Change name to BEAM.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.0

Ypos: 0.5 Ysize: 0.1

Zpos: 0.0 Zsize: 0.09

Select Type: Blockage (default).

To change the material, click on 'Attributes', click on ‘Types’,

click on SOLIDS then 'OK';, and then on 'ALUMINIUM at 27 deg C' and 'OK';.

Click on 'OK' to exit the Attributes menu,

and on 'OK' to close the Object Dialogue Box.

**Create the WATER-IN object**:

Click on 'Object'.

Change name to WATER-IN.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 0.5

Ypos: 0.0 Ysize: 0.0

Zpos: 0.0 Zsize: 0.09

Define Type: Inlet.

Click on 'Attributes' and set velocity in Y-direction at 2.0 m/s.

Set 'Inlet density' at 'Heavy' for the inflowing water.

Click on 'OK' to close the Attributes menu,

and on 'OK' in the Object Dialogue Box.

**Create the OUTLET object**:

Click on 'Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 0.2 Xsize: 0.6

Ypos: 1.0 Ysize: 0.0

Zpos: 0.0 Zsize: 0.06

Define Type: Outlet.

Leave the default values in the Attributes dialogue box.

Click on 'OK' to exit the Object Dialogue Box.

**Create the FREE-TOP object**:

Click on 'Object'.

Change name to FREE-TOP.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.3 Zsize: 0.0

Define Type: Outlet.

Leave the default values in the Attributes dialogue box.

Click on 'OK' to exit the Object Dialogue Box.

**Set the initial values and fluids distributions**:

Click on 'Menu' and then on 'Initialisation'.

Set zero values for P1, U1,V1, W1 and SURN in the fields provided for 'FIINIT values', clicking > to go through the list of variables;

**Set the properties of 'heavy' and 'light' fluids**:

Click on 'Properties'.

To change the light fluid material, click on the field provided, click on GASES then 'OK';, and then on 'AIR at 20 deg C' and 'OK';.

To change the heavy fluid material, click on the field provided, click on LIQUIDS then 'OK';, and then on 'Water at 20 deg C' and 'OK';.

Click on Propert 'Storage' and set 'ON' for Density and Viscosity, then on 'Previous panel' to get back.

**Set gravity**:

Click on 'Sources'.

Set 'Gravitational forces' ON and 'OK'.

Set 'Buoyancy model' CONSTANT

Change 'Gravitational acceleration' to 0.0 in X and Y directions and -9.81 in Z.

**Set the remaining solution-output-control parameters**:

Click on 'Numerics'.

Set the number of iterations to 20.

Click on 'Relaxation control'.

Set relaxations for P1, U1,V1, W1 and SURN to be -0.5, 0.03, 0.03, 0.03 and 3.0, respectively, in the fields provided for 'DTFALS' Click on > to go through the list of variables;

Click on 'Previous panel' to get up.

Click on 'Output'.

Change the monitoring point by setting

IXMON to 5,

IYMON to 5,

IZMON to 5.

Go into 'Field dumping' settings.

Change 'First step' to 1 and 'Last step' to 100.

Change 'Start letter for PHI' CSG1 to N

Click on 'Previous panel' to get up.

Click on 'Top menu' to return to the top menu.

Click on 'OK' to exit the Main Menu.

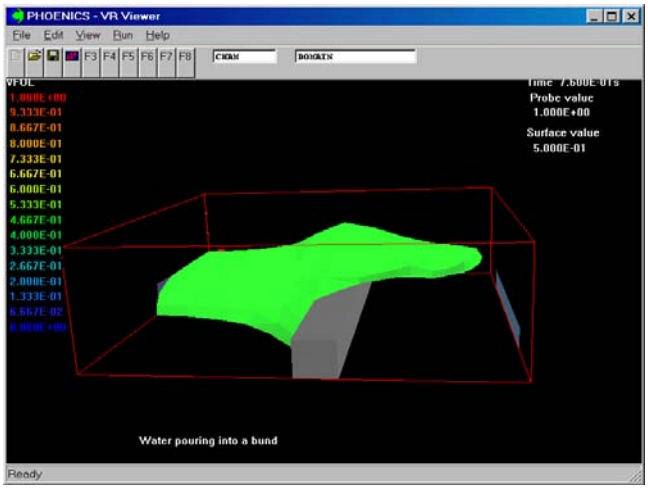
## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on ‘Yes’ when prompted to Save data to files.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) , and

view the velocity vectors, contours and isosurfaces of free surface.




## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'BUND ' (e.g.) and save as 'CASE1' (e.g.).

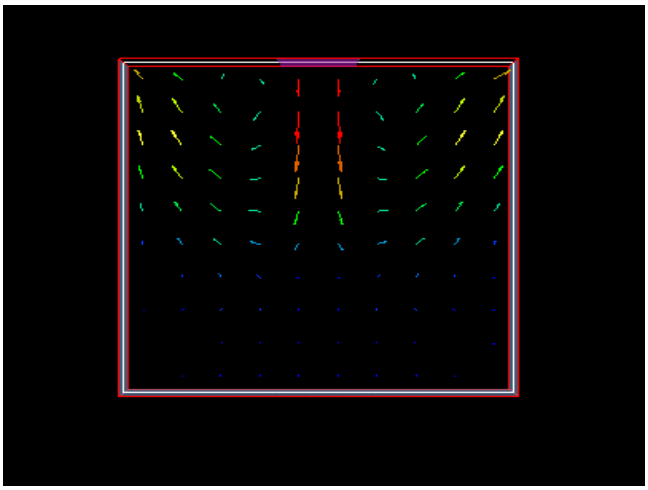
---

svz/331/0201

# WORKSHOP:-

# SEM prototype: A lighter fluid penetrating heavier liquid.

This workshop example shows how to set up and run the simulation of the lighter fluid (oil) stream penetrating, from the slot at the top wall, the domain filled with heavier (water) liquid.
The task is to simulate the velocity field and concentrations distributions.
The geometry and expected velocity vectors are shown in the figure below.



The inlet oil velocity is 2.0 m. The water density is 1000.0 and the oil density is 700, , kg/$m^3$. Two-dimensional domain has 1 x 1 meters in Y and Z directions.

## Accessing PHOENICS-VR

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation .

## Within VR Editor.

**Set the domain size**:

Leave the default sizes.

**Activate the solution settings for velocities and mass fraction of water**:

Click on 'Menu'.

Click on 'Models'

Set 'Solution for velocity and pressure' ON.

Click on 'Page down' and click 'Settings' of 'Extra variables'.

Set 'SOLVE' C1 (mass fraction of water)

and click on 'Apply'

Click on 'Previous panel'

Click on 'Properties'

Set 'Use property tables' OFF

Click on 'Density' CONSTANT, from the list of options select'Linear in scalar' and click 'OK'

Set A=700 and B=1000-700=300

Change 'Storage' ON

Change 'Viscosity' 1.e-6

Click on 'Sources'

Set 'Gravitational Forces' ON and click 'OK' to confirm.

Set 'Gravitational acceleration' 0., 0. and -9.81 in X, Y and Z directions, respectively.

Set 'Reference density' 1000.

Click 'Top menu' and click 'OK'

**Adjust the view point**:

Click on 'Move right' toggle to view in X-direction.

Click 'Reset' to line up.

**Create the OIL-IN object**:

Click on 'Object'.

Change name to OIL-IN.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.0

Ypos: 0.4 Ysize: 0.2

Zpos: 1.0 Zsize: 0.0

Define Type: Inlet.

Click on 'Attributes' and set velocity in Z-direction at -2.0 m/s.

Set 'Inlet density' at 'Domain fluid'.

Set C1 'Inlet value' 0.0

Click on 'OK' to close the Attributes menu,

and on 'OK' in the Object Dialogue Box.

**Create the FREE-N object**:

Click on 'Object'.

Change name to FREE-N.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.0

Ypos: 0.0 Ysize: 0.0

Zpos: 0.0 Zsize: 1.0

Define Type: Outlet.

Click on 'Attributes' and set C1 'External value' USER-SET.

Set 'User-set' 1.0

Click on 'OK' to exit the Object Dialogue Box.

**Create the FREE-S object**:

Click on 'Object'.

Change name to FREE-S.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.0

Ypos: 1.0 Ysize: 0.0

Zpos: 0.0 Zsize: 1.0

Define Type: Outlet.

Click on 'Attributes' and set C1 'External value' USER-SET.

Set 'User-set' 1.0

Click on 'OK' to exit the Object Dialogue Box.

**Create the FREE-LOW object**:

Click on 'Object'.

Change name to FREE-LOW.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.0

Ypos: 0.0 Ysize: 1.0

Zpos: 0.0 Zsize: 0.0

Define Type: Outlet.

Click on 'Attributes' and set C1 'External value' USER-SET.

Set 'User-set' 1.0

Click on 'OK' to exit the Object Dialogue Box.

**Set the spatial grid**:

Click on the 'Mesh toggle' button. The default mesh (shown as orange lines) will appear on the screen.

Click anywhere on the image, and the ‘Gridmesh settings’ dialog box will appear.

Change the Number of cells to 10 cells in Y and 10 cells in Z.

Click on 'OK'; to close the dialog box. The mesh will now be shown as blue and orange lines. The orange lines are ‘regions’, defined by the bounding boxes of the objects.

Click on ‘Mesh toggle’ again to turn off the mesh display.

**Set the remaining solution-output-control parameters**:

Click on'Menu'

Click on 'Numerics'.

Set the number of iterations to 300.

Click on 'Relaxation control'.

Set relaxations for V1, W1 and C1 0.1, in the fields provided for 'DTFALS'.

Click on 'Previous panel' to get up.

Click on 'Output'.

Change the monitoring point by setting

IXMON to 5,

IYMON to 5,

IZMON to 5.

Click on 'Previous panel' to get up.

Click on 'Top menu' to return to the top menu.

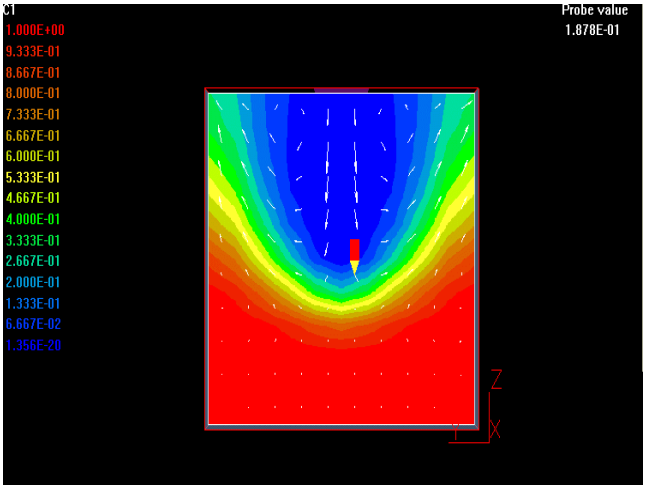
Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on ‘Yes’ when prompted to Save data to files.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) , and view the velocity vectors, contours of water concentration and mixture density.
The calculated water concentration contour is shown below:




## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'OILIN' (e.g.) and save as 'CASE1' (e.g.).

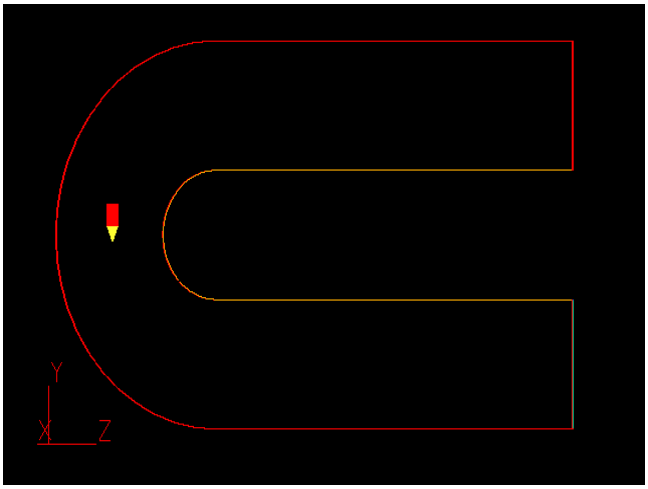
---

svz/332/0201

# WORKSHOP -Local depth in an open U-bend

## Introduction

The case considered is a steady, inviscid, free surface flow of water in an open shallow channel with U-turn. The task is to simulate the velocity field along with local depth distribution.
The geometry is shown below:




The flow depth at the inlet will be 1 m, and the inlet water velocity is 1 m/s.

## Accessing PHOENICS-VR.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

### Load Library case B527 and run the base, rigid wall, case.

Click on 'File', then 'Load from Libraries'. Enter B527 in the 'case number' data entry box, and click OK.

Double click anywhere in the domain and click 'Hide' to clean it up.

Click on 'Main menu', 'Output', and toggle Monitor display from ASCII to GRAPHICS. Set the Monitor update frequency to 1.

Click 'Top menu' and 'OK'.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', then 'Earth' and 'yes' when prompted to save data to files.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run' then 'VR Viewer', and view the velocity vectors, contours and isosurfaces.

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'U-BEND' (e.g.) and save as 'CASE1' (e.g.).

## Within VR Editor.

### Activate solution of isentropic analog of shallow-water equations.

Click on 'Menu'.

Click on 'Properties'.

Click on 'Density 'CONSTANT, and from the list of options select 'Isentropic'. Click 'OK'.

Set RHOA=(2./9.81)**0.5 and RHOB=0.5.

Change 'Storage' ON.

Set 'Reference pressure' $9.81*h_{in}^2/2$, where $h_{in}=1$ m, inlet depth.

Click 'Top Menu' and 'OK'.

### Set the inlet conditions

Click twice on the inlet object (called IN). Click on 'Attributes'.

Change 'the pressure of;' 0.0

Click on 'OK', 'OK'.

### Set the outlet conditions.

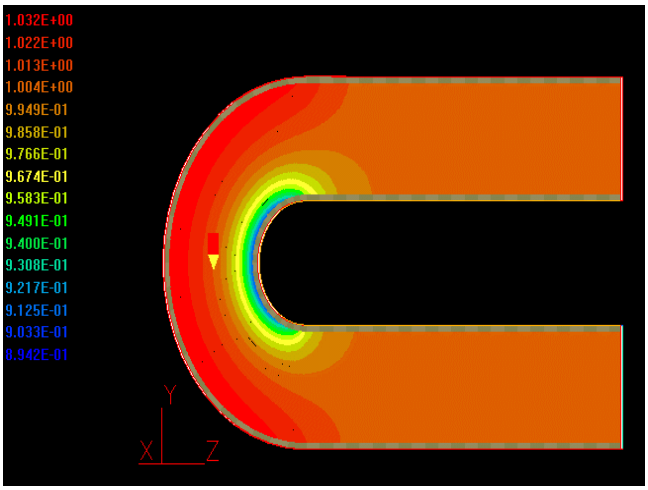
Leave the default.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK' to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor(VR Viewer)', and view the velocity vectors, contours and isosurfaces. Note that depth is represented by DEN1.
The calculated depth distribution is shown below:




## Saving the results.

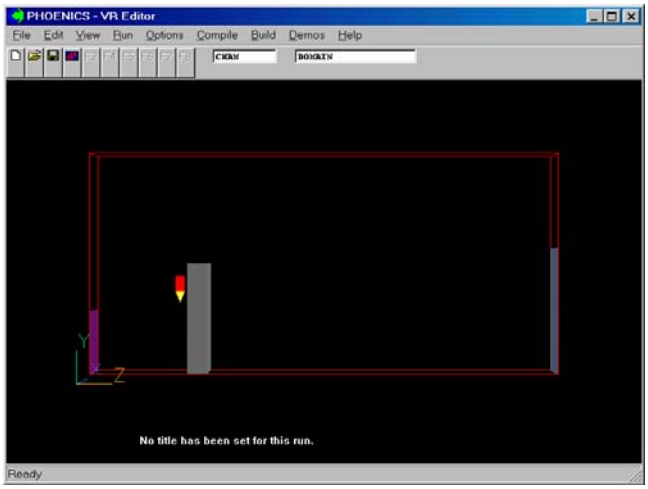
In the PHOENICS-VR environment, click on 'Save as a case', go to the folder 'U-BEND' (e.g.) and save as 'CASE2' (e.g.).

---

svz/331/0201

# WORKSHOP:-
# A whirlpool in a shallow pond:
# viscous flow

This workshop example shows how to set up and run the simulation of free surface turbulent flow of water recirculating in a shallow open pond.
The task is to simulate the velocity field along with local depth distribution.
The geometry is shown in the figure below.




The flow depth at the inlet will be 0.5 m, and the inlet water velocity is 1.0 m. The water density is 1000.0, $kg/m^3$.

## Accessing PHOENICS-VR

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment,*
Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.
*To enter VR Editor:*
This is the default mode of operation .

## Within VR Editor.

**Set the domain size, and activate solution of the required variables**:

Click on 'Main Menu' and then on 'Geometry'.

Change the Y-Domain Size to 10 m.

Change the Z-Domain Size to 20 m.

Click 'OK' to close the Grid mesh settings dialog.

Click on 'Models'.

Set the 'Solution for velocities and pressure' ON.

Click on 'Turbulence models' and select CONSTANT-EFFECTIVE

Click on 'Settings' and set 'Turbulent viscosity' 2.e-4

Click 'Previous panel' to get back

**Activate anisotropic analog of shallow-water equations**:

Click on 'Properties'.

Set 'Use property tables' OFF

Click on 'Density' CONSTANT, from the list of options select'Isentropic' and click 'OK'

Set RHOA=(2.?000./9.81)$^{1/2}$=14.278 and RHOB=0.5

Change 'Storage' ON

Change 'Viscosity' 1.e-6

Set 'Reference pressure' 1000.?.81?.$5^2$/2.= 1226.

Click 'Top menu' and click 'OK'

**Create the BEAM object**:

Use the view toggles to adjust the view point, as appropriate.

Click on 'Object'.

Change name to BEAM.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.0

Ypos: 0.0 Ysize: 5.0

Zpos: 4.17 Zsize: 0.83

Select Type: Blockage (default).

To change the material, click on 'Attributes', click on ‘Types’,

click on SOLIDS then 'OK';, and then on 'ALUMINIUM at 27 deg C' and 'OK';.

Click on 'OK' to exit the Attributes menu,

and on 'OK' to close the Object Dialogue Box.

**Create the WATER-IN object**:

Click on 'Object'.

Change name to WATER-IN.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.0

Ypos: 0.0 Ysize: 2.86

Zpos: 0.0 Zsize: 0.0

Define Type: Inlet.

Click on 'Attributes' and set velocity in Z-direction at 1.0 m/s.

Set 'Inlet density' at 'Domain fluid' at a pressure of 0.0.

Click on 'OK' to close the Attributes menu,

and on 'OK' in the Object Dialogue Box.

**Create the OUTLET object**:

Click on 'Object'.

Change name to OUTLET.

Set POSITION and SIZE of object as:

Xpos: 0.0 Xsize: 1.0

Ypos: 0.0 Ysize: 5.7

Zpos: 20.0 Zsize: 0.0

Define Type: Outlet.

Leave the default values in the Attributes dialogue box.

Click on 'OK' to exit the Object Dialogue Box.

**Set the spatial grid**:

Click on the 'Mesh toggle' button. The default mesh (shown as orange lines) will appear on the screen.

Click anywhere on the image, and the ‘Gridmesh settings’ dialog box will appear.

Change the Number of cells to 14 cells in Y and 24 cells in Z.

Click on 'OK'; to close the dialog box. The mesh will now be shown as blue and orange lines. The orange lines are ‘regions’, defined by the bounding boxes of the objects.

Click on ‘Mesh toggle’ again to turn off the mesh display.

**Set the remaining solution-output-control parameters**:

Click on 'Numerics'.

Set the number of iterations to 150.

Click on 'Relaxation control'.

Set relaxations for P1, V1, W1 and DEN1 -0.5, 10.0, 10.0 , and -0.3 respectively, in the fields provided for 'DTFALS'.

Click on 'Previous panel' to get up.

Click on 'Output'.

Change the monitoring point by setting

IXMON to 5,

IYMON to 5,

IZMON to 5.

Click on 'Previous panel' to get up.

Click on 'Top menu' to return to the top menu.

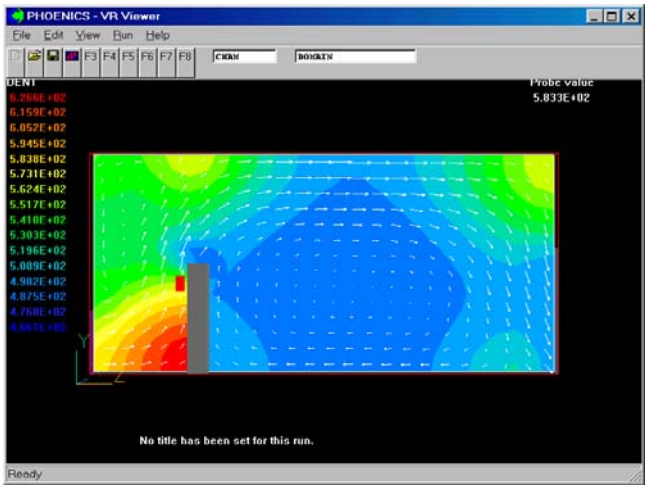
Click on 'OK' to exit the Main Menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver (Earth)', and click on ‘Yes’ when prompted to Save data to files.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) , and view the velocity vectors and contours of pressure and local depth.
Note that depth is represented by DEN1/1000.
The calculated depth?000, m, distribution is shown below:




## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'POND ' (e.g.) and save as 'CASE1' (e.g.).

---

svz/331/0201

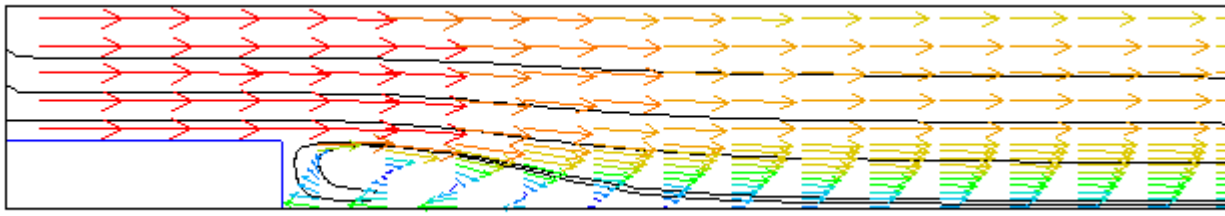
Home Page

# WORKSHOP - GENTRA

# Flow of solid particles over a backward facing step

## 1) The Problem Considered

This workshop shows the user how to set up a simple case which simulates the 2d turbulent flow of solid particles over a backward facing step.



Air, laden with solid particles, enters a 2d plane channel with a velocity of 13 m/s. The total channel width is 0.1143m and the length of the channel is 0.762m. The step is 0.1524m long by 0.0381m high. Turbulence is represented by means of the standard k-e model with equilibrium wall functions.
The density and diameter of the particles is 1500 kg/$m^3$ and 10 microns, respectively. 60% of the particles enter with the air, and the remaining 40% are released near the base of the channel about 0.05m downstream of the step. The mass loading of the particles is about 1% of the air flow.
The particulate phase is represented by 5 tracks of equal mass loading. The initial position and velocity vector of each track are defined below:

| | | |
|---|---|---|
| Track 1 | **X** = (0, 0.09) | **U** = (11.25, -6.5) |
| Track 2 | **X** = (0, 0.07) | **U** = (11.25, -6.5) |
| Track 3 | **X** = (0, 0.05) | **U** = ( 5.63, -0.4) |
| Track 4 | **X** = (0.2, 0.05) | **U** = ( 1.80, -0.6) |

| | | |
|---|---|---|
| Track 5 | **X** = (0.2, 0.01) | **U** = ( 8.50, -0.3) |

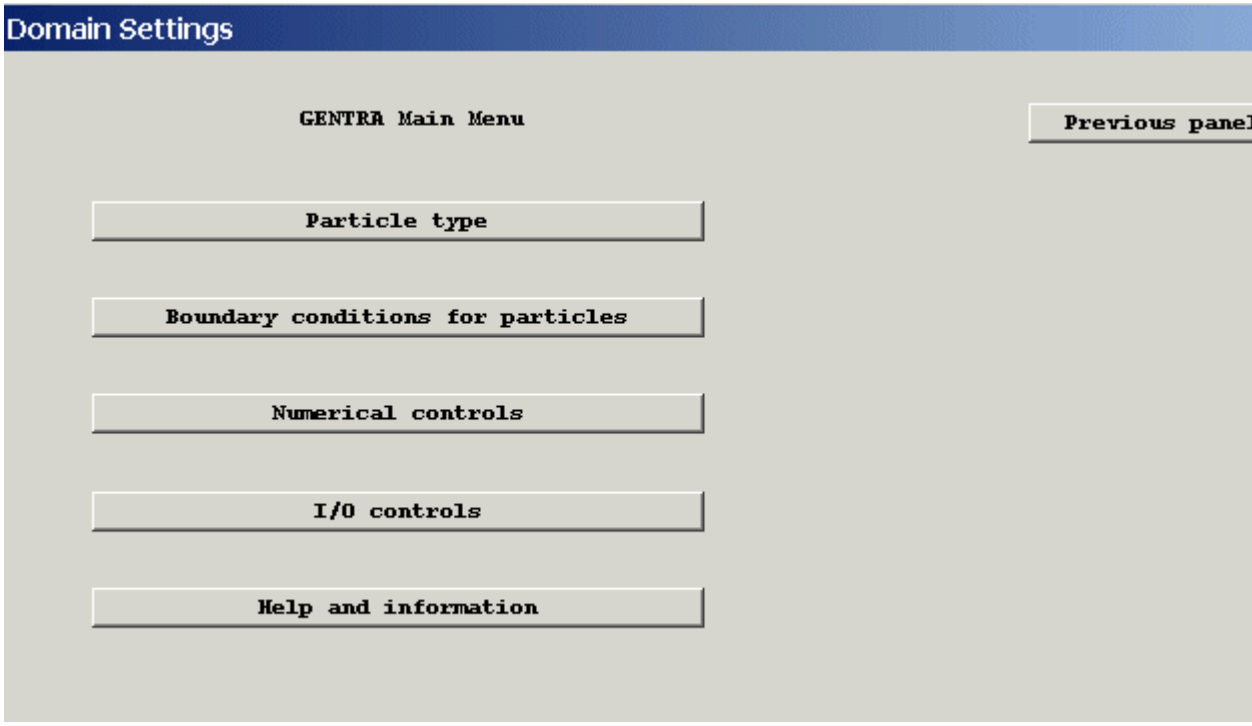
## 1) Loading the fluid-flow case from the PHOENICS Library.

*From the system level:*
To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
*In PHOENICS-VR environment:*
The first task is to load the backward-facing-step case from the PHOENICS library so as to provide a working fluid-flow Q1 file for use with GENTRA.
Click on 'File', 'Load from Libraries', enter 300 in the Case Number data entry box, and click 'OK'.
The case now needs to be modified so as to increase the number of iterations for complete convergence, and to activate the Graphical Display Monitor when running EARTH. Therefore, click on 'Menu' then on 'Numerics' Change the total number of iterations to 150. Click on ‘Output’, and toggle Monitor display mode from 'ASCII' to 'Graphics'. Click 'Top Men' then 'OK' to leave the main menu.
As a benchmark, the case can now be run without particles. Run the EARTH solver by clicking on 'Run', 'Earth' and 'OK'. Save these results by clicking on 'File', 'Save as a case'. Enter the case name GENTRA-0.

## 2) Activating GENTRA

### Loading the GENTRA Menu:

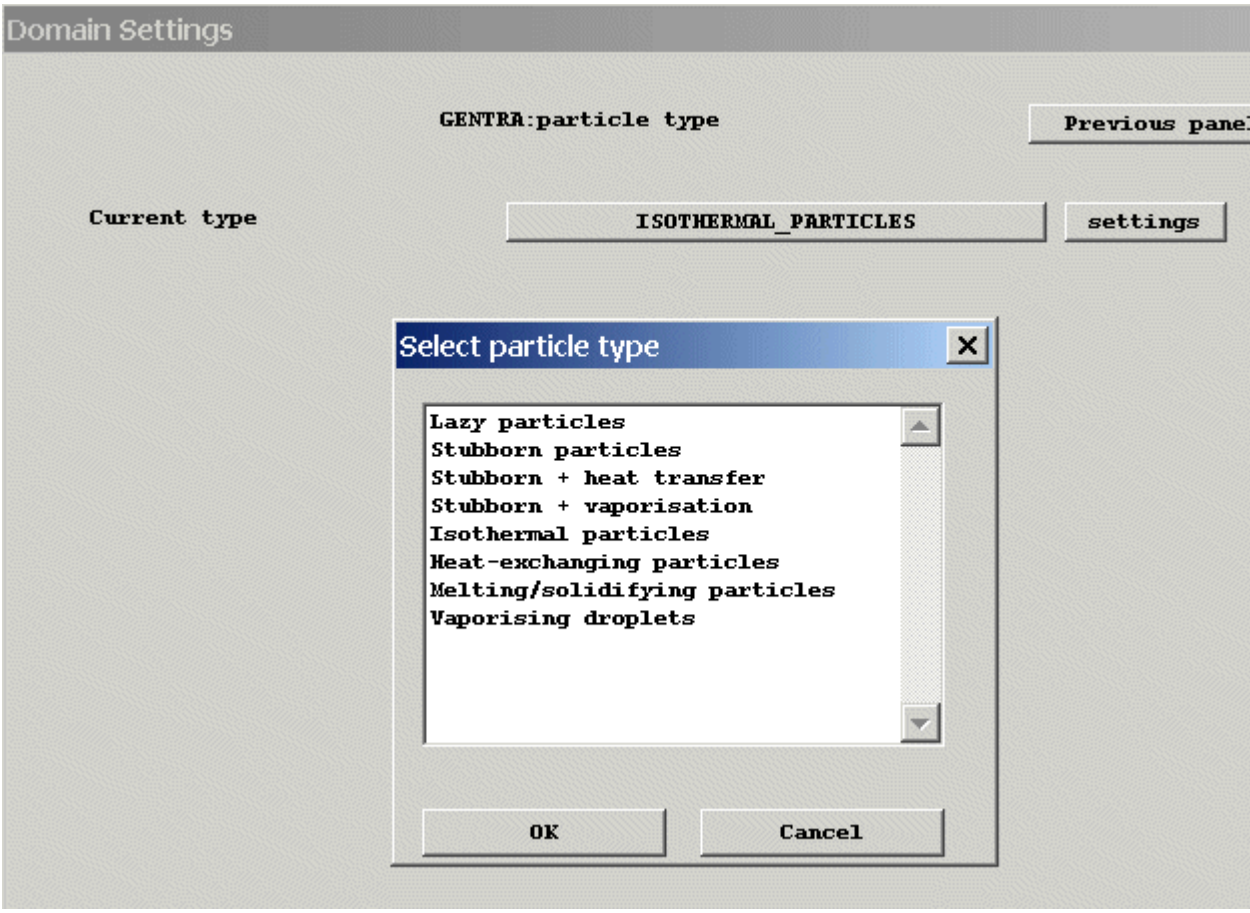
Click on 'Menu', then on 'Models'. Toggle Lagrangian Particle Tracker (GENTRA) to 'ON', then click on 'Settings'. The following menu panel will appear:




**Select the particle type**:

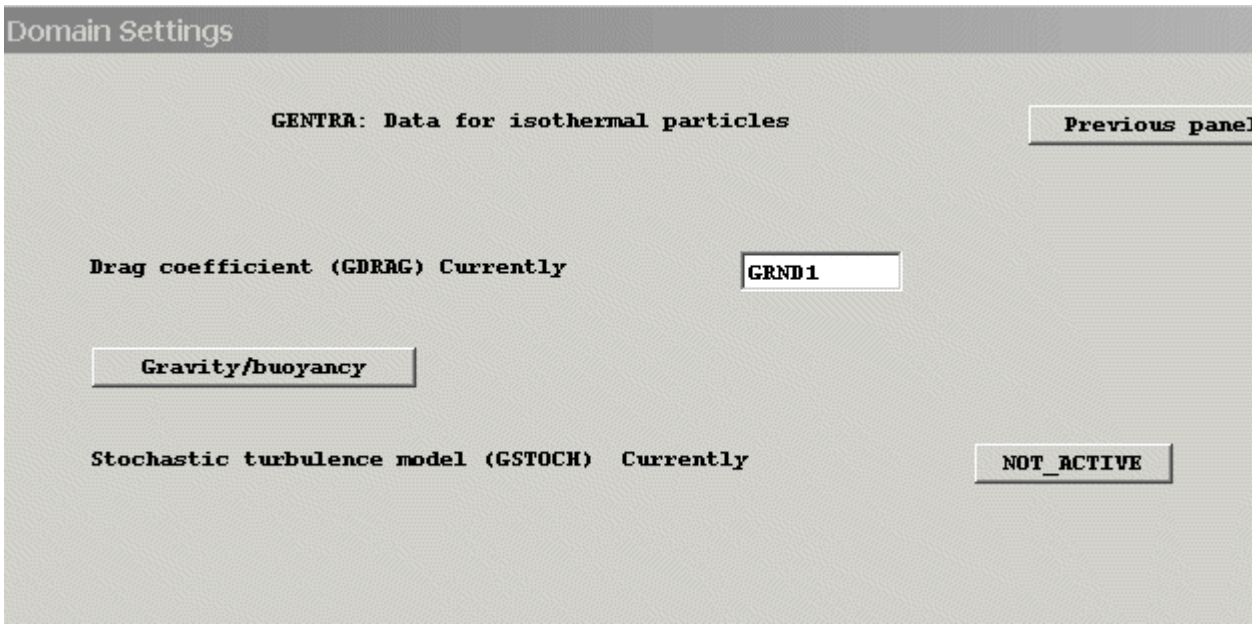
Click on 'Particle Type' so as to display the Particle type menu. The default particle type is 'Isothermal Particles'. This type is appropriate when only momentum transfer is to be considered between the continuous and disperse phases.

Other particles types can be selected by clicking on 'Isothermal_particles'. For this example we do not need to change the particle type.




**Set the physics of the particle:**

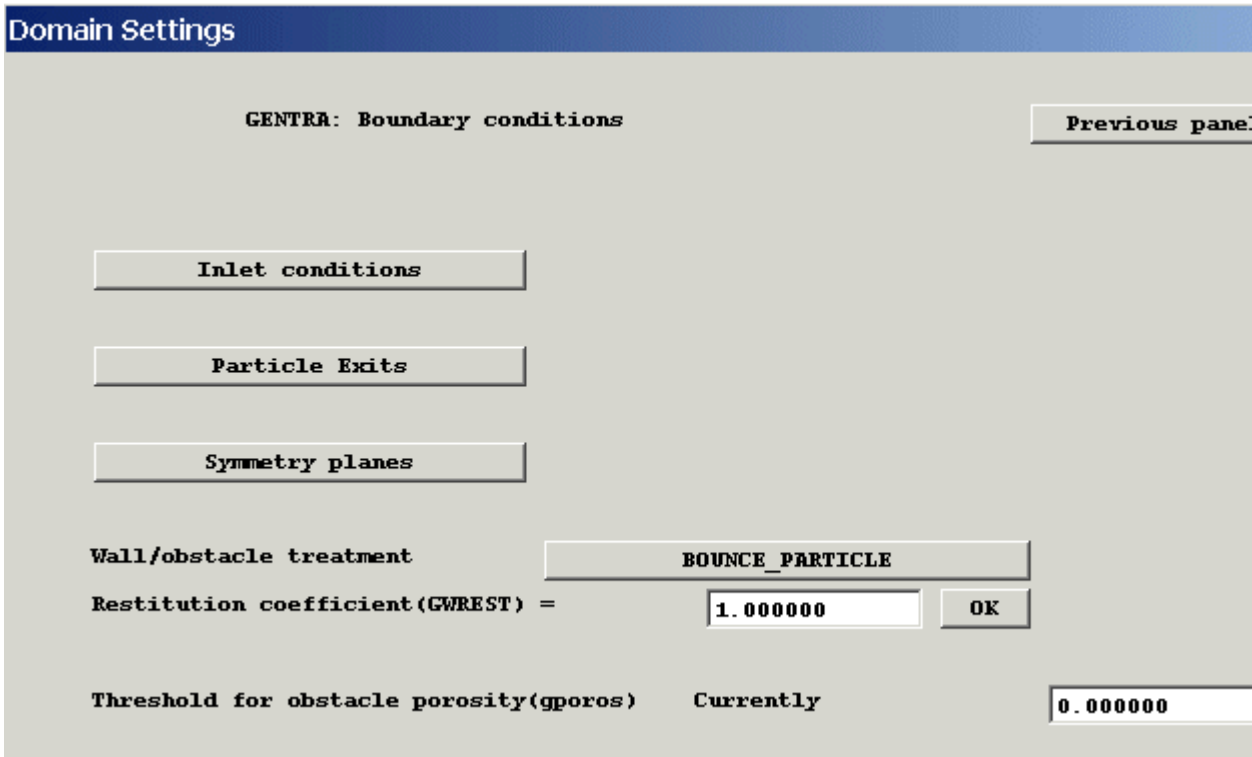
Click 'Settings' for the Menu option 'Physics of current particle type'. As can be seen below, this Menu allows the user to specify the drag coefficient, introduce gravitational or buoyancy forces on the particle, and also activate the stochastic turbulence model for particle tracking.




The built-in drag law GRND1 for solid spheres will suffice. Then return to the previous Menu panel by clicking on 'Previous panel' twice.

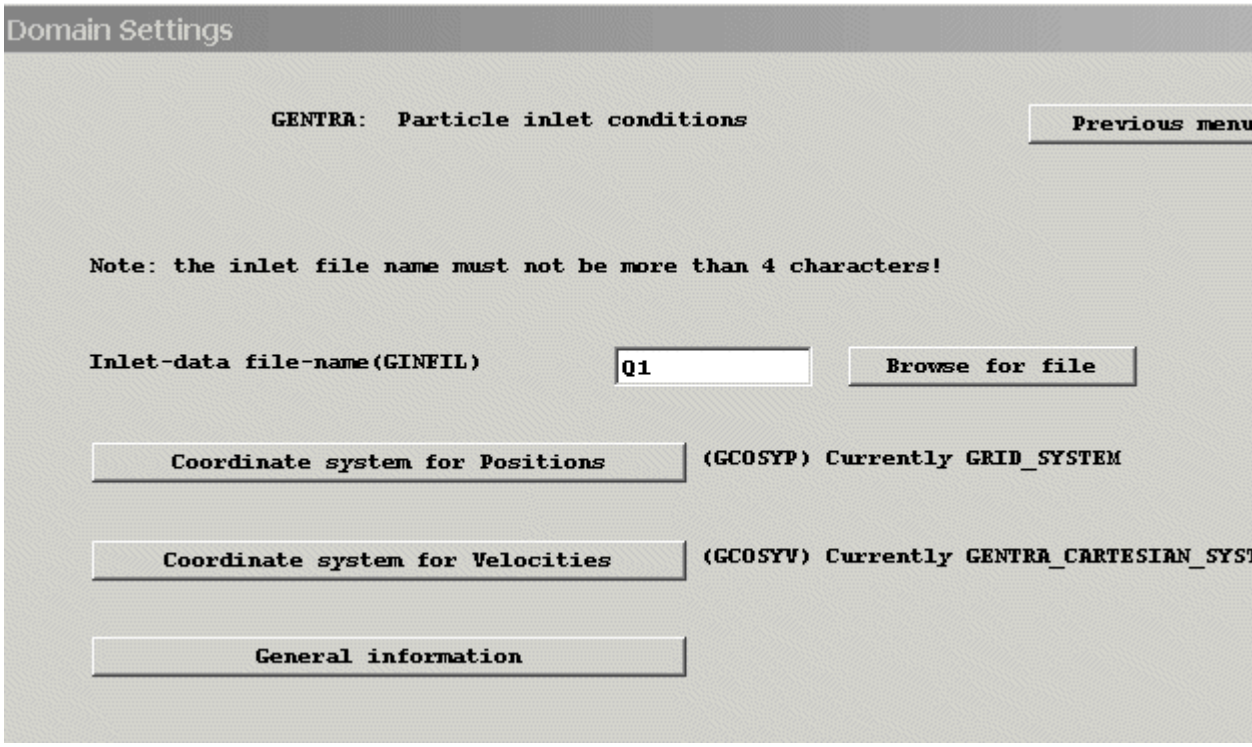
**Setting the boundary conditions for the particles**:

Click 'Boundary conditions for particles'. The following panel will appear:



### Inlet conditions

Click on 'Inlet conditions'.




The particle inlet conditions must be specified by means of a inlet-data file or directly in the Q1 file after the GENTRA Menu session.

For this example, the inlet data for the particles are to be supplied in the Q1 file, which is the default setting. Click on 'Previous panel'

### Particle Exits

Particle exit conditions are set individually for all Inlet, Outlet and Fan objects. By default, particles are allowed to leave through all Inlet, Outlet and Fan objects. The entry here provides help information.

### Symmetry Planes

These are set as Objects. The entry here provides help information. This case does not require any Symmetry planes.

**Wall/Obstacle treatment**

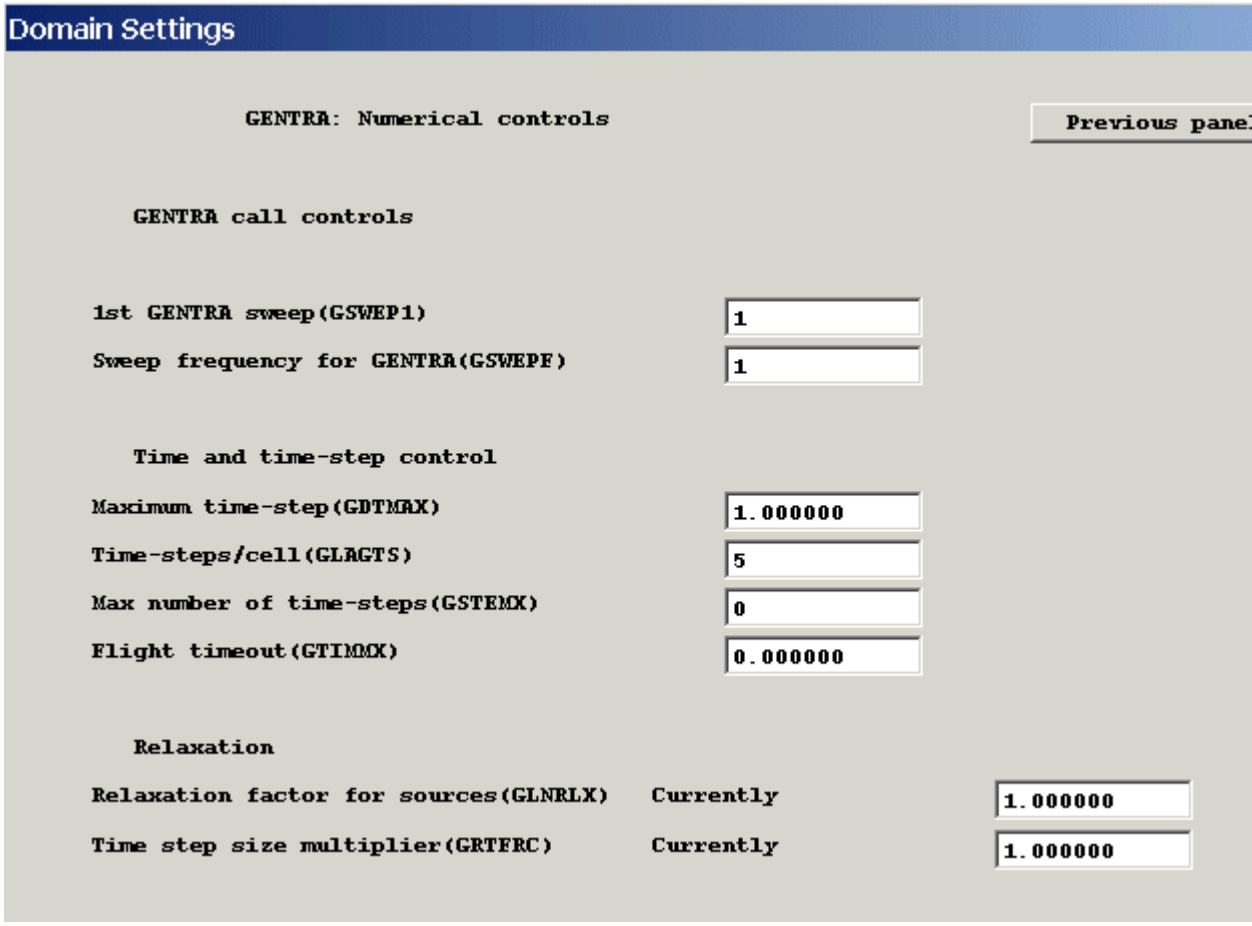
The task here is to specify how the particles interact with walls and obstacles, e.g. bouncing, sticking, removal.

The default is for the particles to bounce. Enter a restitution coefficient of 0.75.

Click on 'Previous panel' to return to the GENTRA Main Menu.

**Setting the Numerical Controls:**

Click on the 'Numerical controls' to select the 'Numerical controls' Menu.



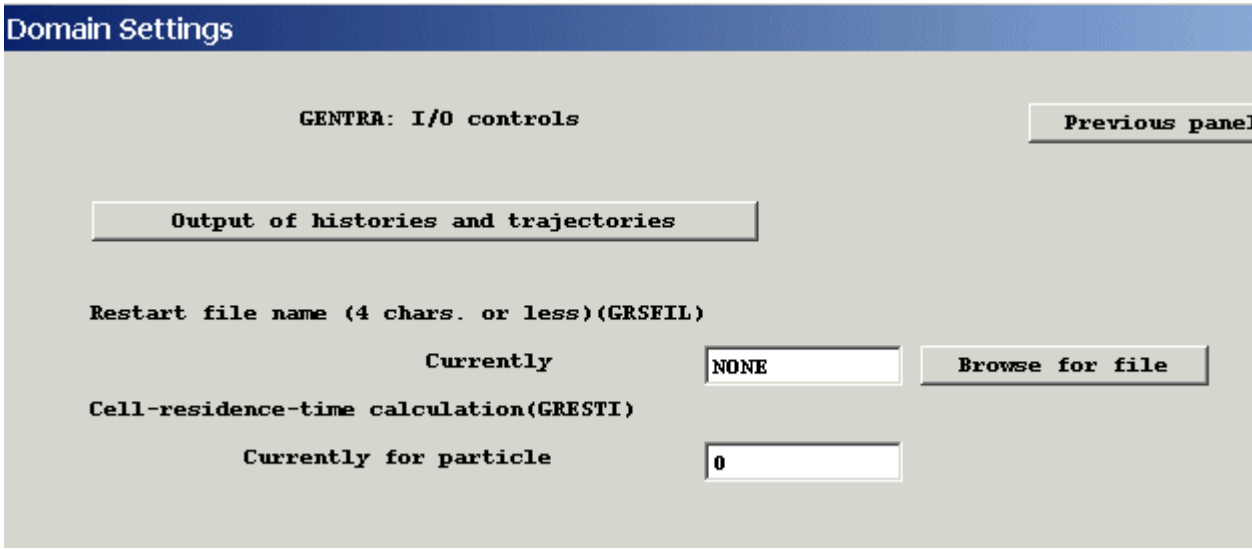
Enter '20' for the 1st GENTRA sweep.

Enter '10000' as the maximum number of time steps.

Enter '3.0' as the flight time out for a particle.

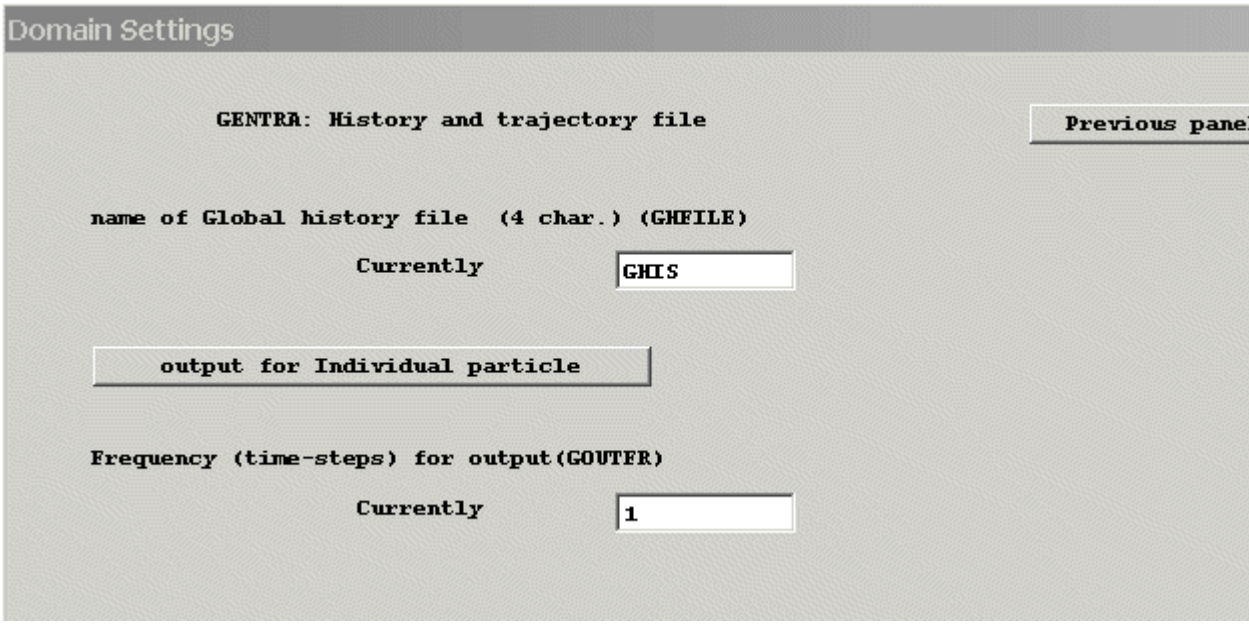
Click on 'Previous panel' to return to the 'GENTRA Main Menu'.

**Setting the Input/Output Controls:**
From the 'GENTRA Main Menu', click on 'I/O Controls '. The following Menu panel will appear on the VDU.

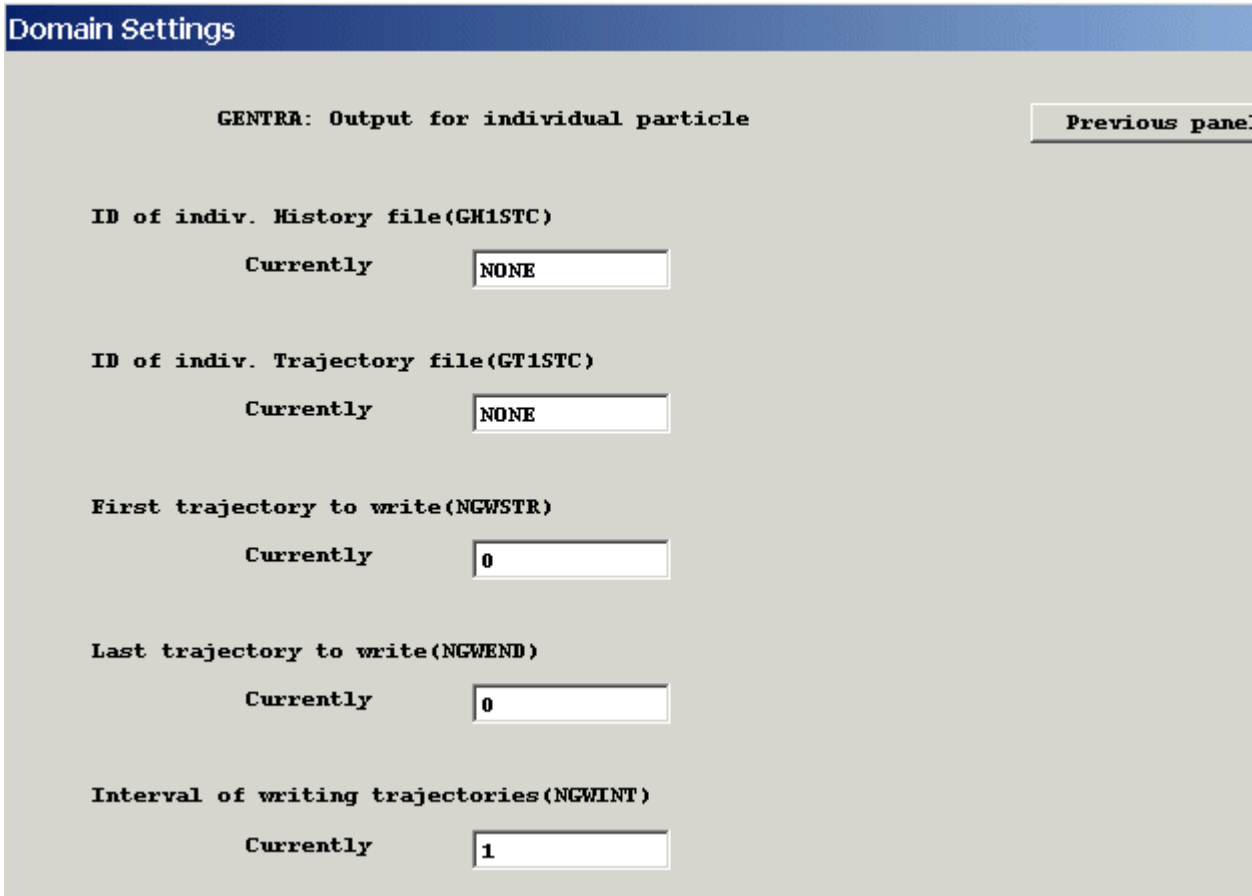



**Output of histories and trajectories:**

Click 'Output of histories and trajectories' to display the following Menu panel.




Since we require individual history and trajectory files, click on 'Output for Individual particle', and then the following menu panel appears.



Enter 'H' as the first character for the name of the individual history file.

Enter 'T' as the first character for the name of the individual trajectory file.

Enter '1' as the first track for which an individual history-trajectory file is required.

Enter '5' as the last track for which an individual history-trajectory file is required.

Click on 'Previous panel' twice to return to the 'I/O controls Menu'.

Click 'Previous panel' twice more to return to the Main Menu.

Click 'Top Menu' and 'OK' to leave the Main menu and return to the VR-Editor.

## 3) Defining the Inlet Conditions for the Particles.

The task is now to specify the particle inlet conditions in the Q1 file.

For more detailed information than is given here, the user is referred to the instructions given in Section 2.7.1 of the GENTRA User Guide TR211.

Click on 'File', then on 'Open file for editing', and then on 'Q1'. Click on 'Yes' when prompted to save files.

Now insert the lines of inlet data between the two special marks provided by the GENTRA Menu Session, i.e.
**<GENTRA-INLET-DATA>** and
**<END-GENTRA-INLET>**.

Please note that the inlet-data lines must not start in the 1st or 2nd column of the Q1 file, and the length of the data should not exceed column 68 in the Q1 file.

The final result is shown in the following extract from the Q1 file:

```
 *-------------------------------------------------------

 * GENTRA GROUP 2: Boundary conditions for particles

 *-------------------------------------------------------

 * Inlet-data file-name

GINFIL   ='Q1'

 <GENTRA-INLET-DATA>

  * X            Y                U          V       DIA
RHO     MASSFLO

   0.0E+00   9.0E-02     11.25 -6.5    1.E-05    1500.
2.00E-03

   0.0E+00   7.0E-02     11.25 -6.5    1.E-05    1500.
2.00E-03

   0.0E+00   5.0E-02     5.63     -0.4    1.E-05    1500.
2.00E-03
```

2.0E-01 0.5E-02 1.80 -0.6 1.E-05 1500.

2.00E-03

2.0E-01 1.0E-02 8.54 -0.3 1.E-05 1500.

2.00E-03

<END-GENTRA-INLET>

In addition, increase LSWEEP from 150 to 500 in the Q1 file, as more sweeps will be needed for convergence of the GENTRA run.

Now save the modified Q1 by clicking on ‘File’, 'Exit' and 'Yes'. Allow VR to re-load themodified Q1.

## 4) Running the GENTRA CFD Solver.

Run EARTH by clicking on'Run', 'Solver' then 'Local solver(Earth)' and 'OK'.

## 5) The GENTRA Output Files.

In addition to the usual **RESULT** and **PHI** file, the EARTH run will produce the following output files:

| | | | | |
|---|---|---|---|---|
| **h00001** | **h00002** | **h00003** | **h00004** | **h00005** |
| **t00001** | **t00002** | **t00003** | **t00004** | **t00005** |
| **genuse** | | | | |

The files **h0001, h0002,** etc are the history files, one for each track.

The files **t0001, t0002,** etc are the trajectory files, one for each track.

The file **genuse** is used to plot the trajectories of all tracks simultaneously in PHOTON.

The history files provide a tabular output of the particle history of each requested track, and these files may be read into AUTOPLOT to produce x-y line plots, such as for example, plots of the particle axial velocity against time.

At each requested Lagrangian time step, the following data are written to each history file:

| PR.No. | TIME | XPAR | YPAR | ZPAR | UCPAR | VCPAR | WCPAR | DIAM | TPAR | TGAS | SFRC |
|---|---|---|---|---|---|---|---|---|---|---|---|

where

PR.No is an integer identifying the number of the track, e.g. 1 for track 1.

TIME is the time in s.

XPAR, YPAR and ZPAR are the Cartesian coordinates of the particle position vector in m.

UCPAR, VCPAR and WCPAR are the Cartesian components of the particle velocity vector in m/s.

DIAM is the particle diameter in m.

TPAR is the particle temperature in K.

TGAS is surrounding fluid temperature in K.

SFRC is the solid mass fraction.

TPAR, TGAS and SFRC are not relevant in the present example.

## 6) Using VR-Viewer & AUTOPLOT to display the Results.

### VR-Viewer

Click on 'Run', 'Post processor',then GUI Post processor(VR Viewer)'.

When the file dialog appears, click 'OK' to use the default files.

The fluid velocity vectors can be displayed by clicking on the Vector toggle on the handset. To colour the vectors by Velocity, click the 'Select velocity button' .

The particle trajectories are displayed by using the macro file written by EARTH. Click on 'Macro' and then enter 'GENUSE' as the name of the macro file to run. Click 'OK' to close the dialog and plot the particle paths.

Now enter 'm' to return back to Menu mode.

Next, Contour plots of the Cartesian components of the interphase momentum force vector can be made, as follows:

Click on the 'Contour toggle' to turn on the contour display. Click on the 'Select variable' button and select 'MOMX' from the list for the forces in the Z direction, and then click on 'OK'.

Repeat this procedure so as to plot contours of MOMY, i.e. the forces in the Y direction.

### AUTOPLOT

In the PHOENICS-Commander environment, click on 'Run, Post processor,X-Y Graph plotter (AUTOPLOT)' and after AUTOPLOT has loaded enter 'FILE' <CR>, and then the following commands:

H00001 2 <CR>
H00002 <CR>

so as to load the history files for tracks 1 and 2.

Temporal plots of the axial component of the particle velocity vectors of tracks 1 and 2 can now be displayed by entering the commands:

| | |
|---|---|
| D 1 TIME UCPAR <CR> | (creates data element 1 from data file 1, i.e. H00001) |
| | |
| D 2 TIME UCPAR <CR> | (creates data element 2 from data file 2, i.e. H00002) |
| | |
| COL3 1;COL9 2 <CR> | (makes an x-y plot of UCPAR against TIME for tracks 1 and 2) |
| | |

Enter the command END <CR> to exit AUTOPLOT and return to the PHOENICS Commander.

## 7) Further Suggestions.

You may now like to create a more demanding problem by using the GENTRA Menu to activate the Stochastic turbulence model.

First, save the current model by clicking on 'File', 'Save as a case'. Enter GENTRA-1 as the case name.

**Stochastic tracking**

The stochastic model allows for the turbulent dispersion of particles as they cross eddies during their trajectories.

Click on 'Run - Pre processor - GUI Pre processor (VR-Editor)' to return to the Editor.

Click on 'Menu', 'Models' and then on 'Settings' to enter the GENTRA Main Menu.

From the 'GENTRA Main Menu', click on 'Particle Type' so as to display the Particle-type Menu.

Next, click 'settings' for the Menu option 'Data for Isothermal Particles'.

Finally, activate the 'Stochastic turbulence model'.

Click 'Previous panel' three times to return to the Main Menu, then 'Top Menu', 'OK' to exit the main menu'.

Run EARTH by clicking on'Run', 'Solver' then 'Local solver(Earth)', and you will find that convergence is more problematic with the stochastic model.

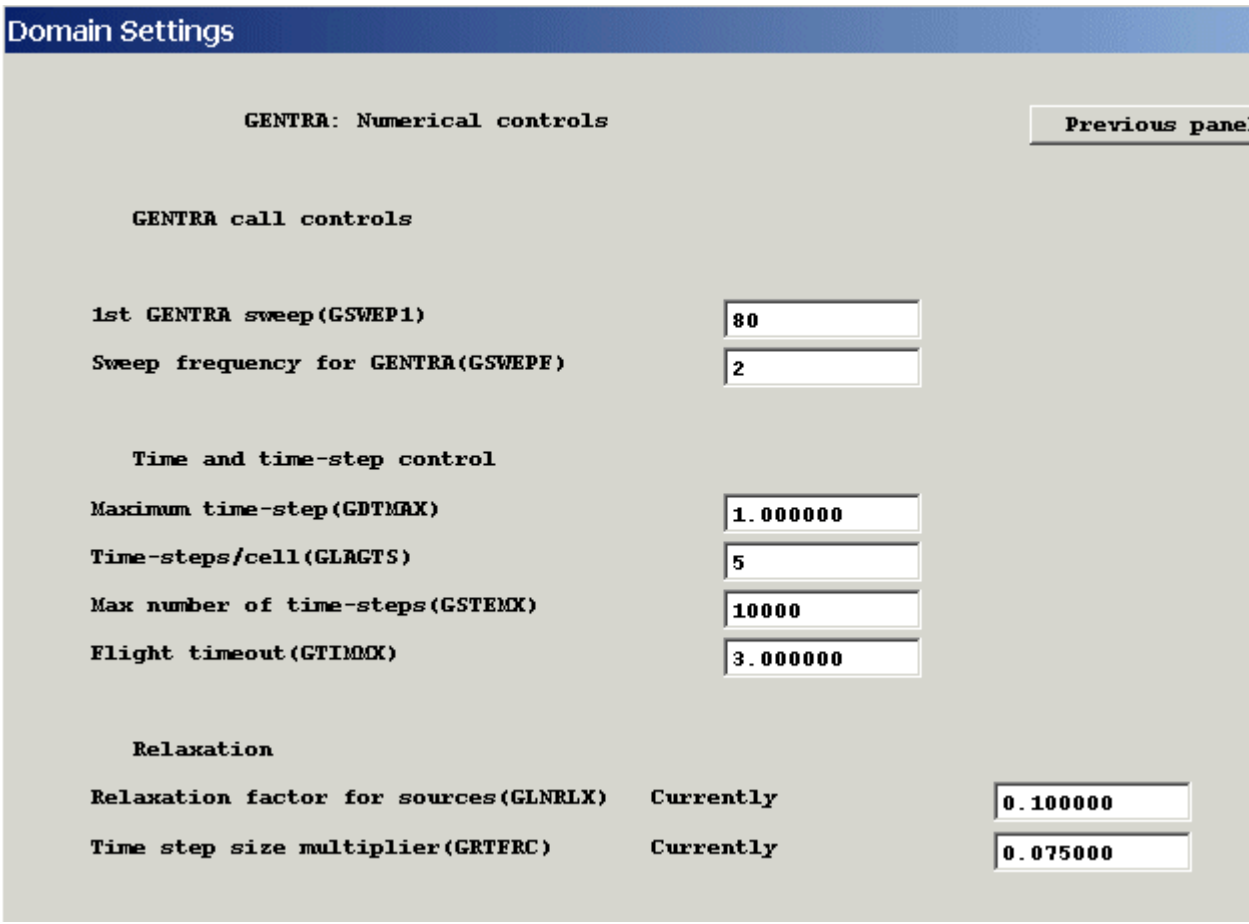
**Convergence control**

You can try to improve the convergence behaviour by using more sweeps (say 500), more inertial relaxation on the fluid momentum equations (say 1.E-3 s), setting KELIN=3 and using linear relaxation on KE and EP (say 0.3). These changes to the fluid-phase numerical parameters should be

made from the 'Numerics', 'Relaxation control' panel of the Main Menu.

Other things to try are the numerical controls for GENTRA particle tracking algorithm.

From the 'GENTRA Main Menu', click on 'Numerical controls' so as to display the Numerical controls Menu.




The foregoing particle-phase settings together are suggested to secure improved convergence behaviour.

Exit the Menu.

Run EARTH by clicking on 'Run' and then 'Earth'.

Home Page

# Setting-up the attributes of moving objects via BVH data file.

Contents


1. Introduction
2. Prerequisites
3. Basics of BVH dataset format
4. BVH data examples
    - Elementary motion: linear movement
    - Simple trajectory: 45 degree climbing
    - Basic spinning: rotating block
    - Composite movement: rolling block
    - Independent motions: crossing paths
    - Connected objects: falling of a cracked wall
5. Engineering applications
    - Rotating-blade impeller
    - Crank-shaft-piston mechanism
6. Summary


---

1. **Introduction**

   In this tutorial, the pre- and postprocessing capabilities of BVH format are demonstrated for assigning the geometrical and kinematical attributes of moving bodies, interaction of which with the flow is simulated by MOFOR techniques.

   In this tutorial, you will read the case and data files ( with and/or without doing the calculation) and perform a number of exercises learning how the BVH format is used for the representation of moving bodies and their assemblies.

2. **Prerequisites**

   This tutorial assumes that you are familiar with the menu structure in **PHOENICS**, and that you have solved Tutorial 1 of MOFOR.

3. **Basics of BVH dataset format**

BVH fomat has two-part structure designed for two classes of the attributes: HIERARCHY and MOTION, time dependent, data.

The hierarchy defines how coordinate systems of the CFD domain and different moving objects are related. The time dependent part defines how, in terms of appropriate distances and time intervals, each coordinate system moves relative to its parent system.

The word **HIERARCHY** at the top of the file is mandatory. It opens the hierarchy part of the attribute settings. UNITS is the word used to specify the attribute units - the default units are inches. METRES must be uppercase.

ROOT, JOINT and End Site are all key words of declaring the frames of reference of object-related coordinate systems. Each frame has a name; ROOT and JOINT must be uppercases and for internal storage reasons each joint's name is restricted to 16 characters.

ROOT is usually used to declare the origin of CFD-grid space. It is often followed by the number of subservient ROOTs or/and JOINTs within brackets.

Each JOINT is followed by a definition section consisting of an open bracket to a matching close bracket. Any joints defined inside the upper, superior joint are the children of this joint.The definition is usually of 1 or 2 lines - OFFSET and CHANNELs - plus any child joints.

The OFFSET is the position of the origin of the rotation axis relative to its parent.

The CHANNELS are the degrees of freedom. They define what type(s) of time dependent data are defined for this joint; the joint may translate or rotate or both, about any axis.

The End Site is the key words used to identify the end of the moving joint.

**MOTION** is a mandatory key word. It identifies the begining of the time dependent part of the BVH file. It is followed by the line specifying the number of Frames. They are the numbers of time moments at which data are given. The number of Frames may

be bigger or smaller than the number of time steps of PHOENICS, but they are all equal in size.

The next line, Frame Time, is the time separation of each set of data. For obvious reasons the total time of the particular set of data is equal to (Frames-1)*Frame Time.

There follows the numbers of frames sets of data; the number of values per line are defined by CHANNELS for each ROOT and JOINT objects as hiearchy specifies.

In what follows the number of BVH files will be exemplified and explained in details.

4. **BVH data examples**
   - ***Elementary motion: linear movement***

     The task is to provide the attributes for a simple motion of a 2D rectangular object called "BLOCK" with uniform velocity of 3m/s in Z-direction of computational domain starting from a given initial, stationary position of the object.

     The hierarchy part of BVH file is as follows:

```
HIERARCHY

UNITS METRES

ROOT Cham

{

        JOINT BLOCK

        {

                CHANNELS 1 Zposition

                End Site
```

```
        }
}
```

Only one of the CHANNELS is defined - the movement is along Z-axis. The initial position and the sizes of the BLOCK are specified in Q1 file.

The time dependent part of the data reads:

```
MOTION
Frames:     2
Frame Time: 1.
0.
3.
```

The Motion section defines the time dependent positions. The file contains 2 sets of positions. At time = 0, Z-position is 0 (plus the offset automatically calculated from initial position of the BLOCK in CFD domain). At time = 1 sec, Z-position is 3 m, which corresponds the velocity of 3 m/s.

Click here to view the animated motion of the object through computational domain (velocity vectors are not shown).

The complete BVH file can be downloaded from here.

To run the simulation:

- Place linear.bvh into your working directory;

- Download Q1 file into the same directory;
- Run VR-editor, Earth and
- VR-viewer to see the movements and resulting velocity fileds.

o ***Simple trajectory: 45 degree climbing***

The task is to provide the attributes for a motion of a 2D, Y-Z, circular object called "CYLINDER" following a prescribed linear trajectory of 45 degree climbing. The velocity of the object is, therefore, uniform and the components in Y- and Z-direction are given as 4 m/s. The object starts from a stationary position at the low-south corner of computational domain.

The hierarchy part of BVH file is as follows:

```
HIERARCHY

UNITS METRES

ROOT Cham

{

        JOINT CYLINDER

        {

                CHANNELS 2 Yposition Zposition

                End Site

        }

}
```

Two CHANNELS are defined - the movement is along Y- and Z-axis. The initial position and the sizes of the CYLINDER are specified in Q1 file.

The time dependent part of the data reads:

**MOTION**

**Frames: 2**

**Frame Time: 1.**

**0. 0.**

**4. 4.**

The Motion section defines the time dependent positions. The file contains 2 sets of positions arranged in two columns: first column is for Y- and second one is for Z-position. At time = 0, Y- and Z-positions are 0 (plus the offset automatically calculated from initial position of CYLINDER in CFD domain). At time = 1 sec, Y- and Z-positions are 4 m, which correspond the velocity components of 4 m/s.

Click here to view the animated motion of the object through computational domain (velocity vectors are not shown).

The complete BVH file can be downloaded from here.

To run the simulation:

- Place diagonal.bvh into your working directory;
- Download Q1 file into the same directory;
- Run VR-editor, Earth and

- VR-viewer to see the movements and resulting velocity fileds.

- ***Basic spinning: rotating block***

The task is to provide the attributes for a rotation of a 2D, Y-Z, rectangular object called "BLOCK" with angular velocity of 30 rpm (revolutions per minute). The rotation is about X-axis of object centre of mass and starts from its stationary position at the middle of computational domain.

The hierarchy part of BVH file is as follows:

```
HIERARCHY

UNITS METRES

ROOT Cham

{

        JOINT BLOCK

        {

            OFFSET   0.0 1.5 2.5

                CHANNELS 1 Xrotation

                End Site

        }

}
```

The OFFSET defines the position of the rotation axis relative to the ROOT frame: it places the axis in the middle of a given computational domain of 1x3x5 m. Only one of the CHANNELS is defined - the rotation about X-axis.

The time dependent part of the data reads:

MOTION

Frames: 2

Frame Time: 1.

0.

180.

The Motion section defines the time dependent positions. The file contains 2 sets of positions. At time = 0, angular position of rotation is 0. At time = 1 sec, angular position is 180 degrees, that gives the angular velocity of 0.5 revolution per second or 30 rpm as required.

Click here to view the animated motion of the object through computational domain (velocity vectors are not shown).

The complete BVH file can be downloaded from here.

To run the simulation:

- Place rotblock.bvh into your working directory;
- Download Q1 file into the same directory;
- Run VR-editor, Earth and
- VR-viewer to see the movements and resulting velocity fileds.

o ***Composite movement: rolling block***

The case exemplifies the specification of the attributes for composite movement: the translation of rotating block. It results in the effect of the rolling 2D rectangular object with the longitudinal (Z-direction) velocity component of 3 m/s and the angular rotation about block centre line (X-axis) of 30 rpm. The movement starts from the initially steady position defined in Q1 file.

The hierarchy part of BVH file is as follows:

```
HIERARCHY

UNITS METRES

ROOT Cham

{

        JOINT BLOCK

        {

            OFFSET    0.0 1.5 0.7

                CHANNELS 2 Zposition Xrotation

                End Site

        }

}
```

The OFFSET defines the position of the rotation axis relative to the ROOT frame: it places the X-axis in the middle of BLOCK initial position. Two CHANNELS are defined - the movement along Z-axis and the rotation about X-axis.

The time dependent part of the data reads:

```
MOTION

Frames:        2

Frame Time: 1.

0.             0.
```

3.0 180.

The Motion section defines the time dependent positions. The file contains 2 sets of positions. At time = 0, the longitudinal and angular positions are both 0. At time = 1 sec, the Z-position is 3m and angular position is 180 degrees, which give the translation velocity of 3m/s and angular velocity of 0.5 revolution per second or 30 rpm as required.

Click here to view the animated motion of the object through computational domain (velocity vectors are not shown).

The complete BVH file can be downloaded from here.

To run the simulation:

- Place rolblock.bvh into your working directory;
- Download Q1 file into the same directory;
- Run VR-editor, Earth and
- VR-viewer to see the movements and resulting velocity fileds.

o ***Independent motions: crossing paths***

The case exemplifies the setting-up the attributes for the independent movements of two objects following their own linear trajectories in 2D, Y-Z, computational space.

The movement of the first object, called CYL1, starts from its stationary position at the low-south corner. The velocity of CYL1 is uniform and the components in Y- and Z-direction are both given as 4 m/s.

The second object, CYL2, starts at high-south corner of the domain and follows the linear trajectory crossing the one of CYL2. It has the constant velocity component of -3 m/s in Z-direction and 3 m/s in Y-direction.

The hierarchy part of BVH file is as follows:

HIERARCHY

UNITS METRES

ROOT Cham

{

JOINT CYL1

{

CHANNELS 2 Yposition Zposition

End Site

}

JOINT CYL2

{

CHANNELS 2 Yposition Zposition

End Site

}

}

Two CHANNELS are defined for each object - the movement is along Y- and Z-axis. The initial position and the sizes of the objects are taken from Q1 file.

The time dependent part of the data reads:

MOTION

Frames: 2

**Frame Time: 1.**

**0. 0. 0. 0.**

**4. 4. 3. -3.**

The Motion section defines the time dependent positions. The file contains 2 sets of positions arranged in four columns.

The first two columns are for CYL1: first one defines Y- and second one defines Z-position. At time = 0, Y- and Z-positions are 0 (plus the offset automatically calculated from initial position of CYL1 in CFD domain). At time = 1 sec, Y- and Z-positions are 4 m, which correspond the velocity components of 4 m/s.

The next two columns are for CYL2: third one defines Y- and the fourth one defines Z-position. At time = 0, Y- and Z-positions are 0 (plus the offset automatically calculated from initial position of CYL2). At time = 1 sec, Y-position is 3 m, which corresponds the velocity components of 3 m/s. In contrast, Z-position is set as -3 m to provide -3 m/s for the velocity component in that direction as required.

Click here to view the animated motions of the objects through computational domain (velocity vectors are not shown).

The complete BVH file can be downloaded from here.

To run the simulation:

- Place twocylin.bvh into your working directory;
- Download Q1 file into the same directory;
- Run VR-editor, Earth and
- VR-viewer to see the movements and resulting velocity fileds.

- ***Connected objects: falling of a cracked wall***

  The case exemplifies the setting-up the attributes for the connected movements of two objects following their relative rotation in 2D, Y-Z, computational space.

  The movement of the first object, called BLOCK, starts from its stationary position as a vertical wall with its base placed next to the middle of the bottom domain boundary. BLOCK is allowed to fall. It does so by rotating clockwise about X-axis of its south-high corner. The angular velocity of the fall is 75 degrees per second.

  Initially, the second smaller object, TIP, sits stationary on the top of BLOCK and can be regarded as a part of the wall. Once the whole wall starts to fall, it cracks and TIP begins to move in opposite direction by rotating counterclockwise about X- axis of the north-low corner of the BLOCK. The angular velocity of the TIP relative to the BLOCK is 180 degree per second.

  The hierarchy part of BVH file is as follows:

```
HIERARCHY

UNITS METRES

ROOT Cham

{

  JOINT BLOCK

  {     OFFSET   0.0 0.0 2.1

        CHANNELS 1 Xrotation

    JOINT TIP

      {  OFFSET   0.0 2.5 -0.5

         CHANNELS 1 Xrotation
```

```
        End Site
      }
    }
}
```

Two JOINTS are defined: BLOCK is a parent joint, and TIP is the BLOCK's child.

The OFFSET of BLOCK defines the position of the rotation axis relative to the ROOT frame: it places the origin of parent related coordinate frame at the south-high corner of BLOCK initial position. Only one of the CHANNELS is defined for parent JOINT- the rotation about X-axis.

The OFFSET of TIP defines the position of the rotation axis relative to the BLOCK frame: it places the origin of its coordinate frame at the north-low corner of BLOCK initial position. Only one of the CHANNELS is defined for child JOINT- the rotation about X-axis.

The time dependent part of the data reads:

```
MOTION
Frames:     2
Frame Time: 1.
0.      0.
75.     -180.
```

The Motion section defines the time dependent positions. The file contains 2 sets of positions arranged in two columns.

The first column is for parent, BLOCK, to define its angular position in degrees of rotation. At time = 0, there is no rotation, so that the angular position is 0. At time = 1 sec, the angular position is 75 degrees. It gives the required fall velocity. The angle is positive as needed for clockwise direction of rotation.

The next column is for child, TIP to define its angular position in degrees of rotation. At time = 0, there is no rotation, so that the angular position is 0. At time = 1 sec, the angular position is -180 degrees. It gives the required velocity of TIP rotation. The angle is negative to provide counterclockwise direction of rotation.

Click here to view the animated motions of the objects through computational domain (velocity vectors are not shown).

The complete BVH file can be downloaded from here.

To run the simulation:

- Place brokntip.bvh into your working directory;
- Download Q1 file into the same directory;
- Run VR-editor, Earth and
- VR-viewer to see the movements and resulting velocity fileds.

5. **Engineering applications**
   - ***Rotating-blade impeller***

Impellers are the units one most commonly associates with stirred reactors. The particular design considered here consists of the ROD, mounted on rotating vertical SHAFT. The rod carries two PADDLEs, which agitate the flow. The paddles are allowed to rotate in different directions about the axis of the rod. As the impeller

rotates it forces the surrounding fluid to rotate with it. The design with rotating paddles is used when "in-depth" mixing is a must.

The task of the exercise is to provide the attributes for connected movements of SHAFT-ROD-PADDLEs assembly following their relative rotation in 3D computational space. The angular velocity of the shaft is 15rpm. The blades rotate with the same velocity clokwise and counterclockwise, respectively.

The hierarchy part of BVH file is as follows:

```
HIERARCHY

UNITS METRES

ROOT Cham

{

  JOINT SHAFT

   {   OFFSET    0.15 0.25 0.25

       CHANNELS 1 Xrotation

   JOINT ROD

    JOINT PADDLE1

     {   OFFSET    0.0 0.0 0.0

         CHANNELS 1 Yrotation

         End Site

     }

     JOINT PADDLE2

      {   OFFSET    0.0 0.0 0.0
```

CHANNELS 1 Yrotation

End Site

}

}

}

Four JOINTS are defined: SHAFT is a parent joint, and ROD, PADDLE1 and PADDLE2 are the SHAFT's childs.

The OFFSET of SHAFT defines the position of the rotation axis relative to the ROOT frame: it places the origin of parent related coordinate frame at the middle of the SHAFT bottom face of its initial position. Only one of the CHANNELS is defined for parent JOINT- the rotation about X-axis.

The ROD rotates about the same axis and has no freedom to move relatively to SHAFT. That is why the RODs definition section is completely ommited here. The OFFSETs of both PADDLEs are also zero: both rotates about the same origin, but their CHANNELs should indicate that it is Y-axis they rotate about.

The time dependent part of the data reads:

MOTION

Frames: 2

Frame Time: 1.

0. 0. 0.

90. 90. -90.

The Motion section defines the time dependent positions. The file contains 3 sets of positions arranged in two columns.

The first column is for parent, SHAFT, to define its angular position in degrees of rotation. At time = 0, there is no rotation, so that the angular position is 0. At time = 1 sec, the angular position is 90 degrees. It gives the required 15 rpm. The angle is positive as needed for clockwise direction of rotation. There is no need for the ROD time data because it is a child which has no associated CHANNELS.

The next column is for second child, PADDLE1, to define its angular position in degrees of rotation. At time = 0, there is no rotation, so that the angular position is 0. At time = 1 sec, the angular position is 90 degrees. It gives the required velocity of PADDLE1 rotation about Y-axis of the SHAFT. The angle is positive as needed for clockwise direction of rotation.

The third column is for third child, PADDLE1, to define its angular position in degrees of rotation. At time = 0, there is no rotation, so that the angular position is 0. At time = 1 sec, the angular position is 90 degrees. It gives the required velocity of PADDLE1 rotation about Y-axis of the SHAFT. The angle is negative to provide counterclockwise direction of rotation.

Click here to view the animated motions of the objects through computational domain (velocity vectors are not shown).

The complete BVH file can be downloaded from here.

To run the simulation:

- Place rotpaddl.bvh into your working directory;
- Download Q1 file into the same directory;
- Run VR-editor, Earth and
- VR-viewer to see the movements and resulting velocity fileds.

- ***Crank-shaft-piston mechanism***

The model of the mechanism schematics of which is shown here consists of the followings:

- The crankshaft (at the centre of the engine), rotates at constant speed
- The big end - where the connecting rod connects to the crankshaft
- The little end - where the connecting rod attaches to the piston.
- There is one small joint used to offset the connecting rod from the other parts of the linkage so that the positions of the component parts do not interfere with each other.

The crankshaft is the lowest long rod; the cranks are the pair of grey rods; the connecting rod is the long rod up to the large vertical piston. The long narrow rod at the end of the cranks shows the position of the extra joint (the Big End) preventing the connecting rod interfering with the crank joints. The top of the connecting rod is the Little End joint which links onto the piston.

The rotation rate of the engine (the crankshaft) is assumed constant; so the motion of the crankshaft is a simple rotation about the Z axis. Its angular position is:

Angb = omega*time

The position of the end of the crankshaft of radius r (relative to the crankshaft) is

(xb,yb) = (r.sin(Angb), r.cos(Angb))

The top end of the connecting rod moves along a line of constant x & y (x,y=0,0). Its z position is a function of the crankshaft rotation. The top end of the connecting rod is at a constant distance from the end of the crankshaft; its position is:

zc = yb+sqrt(lc^2-xb^2).

The value lc is the length of the connecting rod, xb the position of the end of the crankshaft (given above). The angle of the connecting rod (relative to the world) is derived from the crankshaft x coordinate and the difference between the top of the connecting rod and the y coordinate of the crankshaft.

Angc = atan(zc-yb, xb)

The angle relative to the crankshaft (this is the angle required by the hierarchy) is:

AngCC = Angc-Angb

The piston has zero rotation relative to the world, this requires its orientation relative to the connecting rod to be (-Angc), and relative to the connecting rod angle

Angp=(-Angc) = -AngCC+Angb.

Thus just 3 angles specify the full description of the piston & its linkage.

The hierarchy part of BVH file is as follows:

```
HIERARCHY

UNITS METRES

ROOT BLOCK

{

    OFFSET      0.000     0.00      0.00

    CHANNELS 0

    ROOT CRANKS

    {
```

```
OFFSET  0.2000     0.1500     0.00

CHANNELS 1 Zrotation

JOINT CRANKSHA

{   # at radius = 0.1m

        OFFSET  0.00        0.1         0.00

        CHANNELS 0

        JOINT BIGEND

        {   # offset to the centre of piston

                OFFSET  0.00        0           0

                CHANNELS 1 Zrotation

                JOINT CONROD

                {   # crank length = 0.25m

                        OFFSET  0.00        0.25        0.00

                        CHANNELS 1 Zrotation

                        End Site

                        {   # piston itself = c. 4 inches, =0.09m

                                OFFSET  0.0         0.09
0.000000

                        }

                }

        }

}
```

```
    }
}
```

The OFFSET is the position of the origin of the coordinate system relative to its parent. The example ROOT joint ('crankshaft') is offset by 0.2, 0.15, 0.1 metres from the world (0,0,0) - this places it in the centre of the CFD domain (defined in the q1 file).

The ROOT joint rotates about the Z axis -this is the angle of the crankshaft, and hence of the cranks.

The CHANNELS define what type(s) of time dependent data are defined for this joint; the joint may translate or rotate or both, about any axis (eg Zrotation means that rotation about the Z axis is defined in the time section).

The Joint 'Cranks' has a length of 0.1 m - that is the distance of the 'Big End' joint from the crank shaft. When there is no rotation the Cranks start at (0.2, 0.15, 0.18) and end at (0.2, 0.25, 0.18). The Cranks joint has no time.

The 'Big End' joint is offset by 0.08 metres perpendicular to the system stopping the connecting rod from interfering with the Big End joint. The Big End has one time dependent data - a rotation about the Z axis (relative to the cranks) data.

The connecting rod is 0.25 m long - it has to be longer than the Big End so that the piston can remain on the axis of the engine- and has rotation about the Z axis (relative to the 'Big End', and hence relative to the crankshaft). The 'connecting rod' rotation specifies the rotation of the Piston relative to the 'connecting rod'.

The piston (or 'End Site') is of length 0.09 metres.

The time dependent part of the data reads:

MOTION

Frames: 48

Frame Time: 0.001

0 0 0

9 -12.58755762 3.587557617

18 -25.10030723 7.100307227

(….. total of 48 lines with data defined in the hierarchy)

Frames specify that there will be 48 times at which data is given. Frame Time - the time separation of each set of data.

There follows the number of frames (48) sets of data; the hierarchy specified that only 3 angles were required, so there are 3 values per line of data.

The rotation angles at time = 0 are (0,0,0). At time 0.00005 sec (9, -12.58755762, 3.587557617).

Click here to view the animated motions of the objects through computational domain (velocity vectors are not shown).

The complete BVH file can be downloaded from here.

To run the simulation:

- Place engine.bvh into your working directory;
- Download Q1 file into the same directory;
- Download the geometry data files for block, crank,

connecting rod and piston into the same directory;

- Run VR-editor, Earth and
- VR-viewer to see the movements and resulting velocity fields.

6. **Summary**

This tutorial has demonstrated the use of many of the features of BVH data file available in **PHOENICS**. For more information on these and related features, see the lecture "MOFOR: Moving Frames of Reference".

---

# Flow in an Agitated Tank using MOFOR

---

## Introduction

This tutorial illustrates the setup and simulation of a three-dimensional agitated vessel by using MOFOR, Moving Frames of Reference, capability of **PHOENICS 3.5** to predict the time-dependent flow, mixing and chemical reaction. The time-varying impeller-fluid interaction is modelled by allowing the objects associated with the moving impeller to rotate relative to the stationary mesh associated with the vessel.

Chemical industry makes much use of large reactor vessels, in which contact between the reactants is effected by mechanical stirring. The productivity of the reactor depends greatly on the effectiveness of the mixing; it is often important to predict the flow and concentration fields in the the mixing region and in its neighbourhood in order to properly design the rotating impellers.

In this tutorial, you will learn how to

- Set material properties, initial and boundary conditions for a transient calculations;
- Read an existing body movement, BVH, file into **PHOENICS**;
- Set material properties and attributes of moving bodies;
- Activate MOFOR capability;
- Postprocess transient data sets;
- Calculate flow and mixing as a pre-cursor of reactive flow predictions;
- Examine the flow and concentration fields using graphical means;
- Enable the chemical-reaction model and define its inputs;
- Enable the higher-order discretization scheme for improved prediction of concentration fields;
- Enable re-start capability for multi-cycle simulations.

---

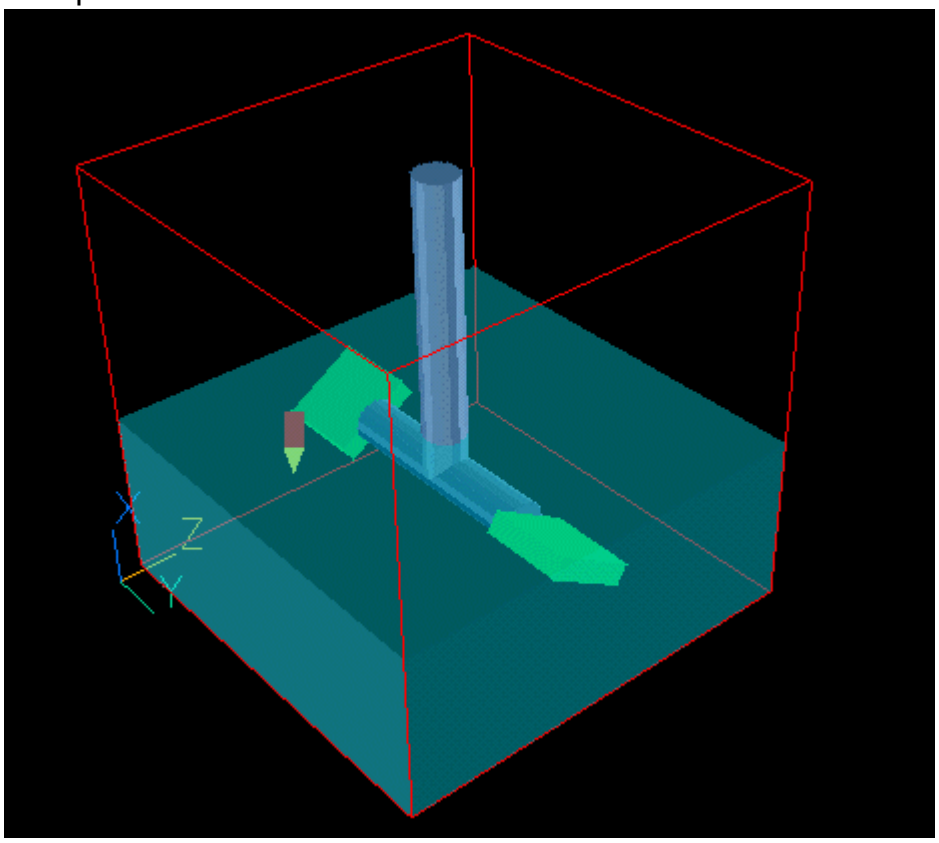
**Pre-requisites**

This tutorial assumes that you have little experience with **PHOENICS**. However, it is supposed that you are, at least, generally familiar with its user interface, VR-Editor/Viewer. If you are not, please review TR324 "Starting with PHOENICS-VR".

---

**Problem description**

The problem to be considered is shown here.



The rectangular vessel, 0.5x0.5x0.5 m, houses two layers of initially unmixed, chemically reactive fluids: one of the 0.2 m in height is underneath of another. They are both at rest, and are separated by a horizontal interface. The impeller, suddenly set in motion, forces the surrounding fluids to rotate with it, mix with each other and react to produce the product.

The impeller consists of the rod, mounted on rotating vertical shaft. The rod carries two inclined paddles, which agitate the flow. The angular velocity of the shaft is 60 rpm.

The physical properties of the fluids are assumed as the same and constant.

Reaction is treated as a single-step irreversible diffusion-controlled with an infinitely fast rate between reactants:
$Fluid_1$(Fuel) + $Fluid_2$(Oxidant) = 2.Product
The fluid composition is related to the mixture fraction according to the Simple Chemical Reaction Scheme, SCRS, concept.
The computational task is to predict unsteady mixing, represented by the subsequent distributions of velocity and concentrations, and the productivity history of the reactor, ie the product concentration at any point as a function of operational time.

---

## Preparation

1. Copy the file agitated.bvh to your working directory.
2. Start the VR-Editor of **PHOENICS**.

---

## Step 1: Domain geometry and grids

1. Click on 'Menu' and then on 'Geometry'.
2. Change the X-,Y-, and Z-Domain Size to 0.5 m.
3. Change the Number of cells to 20 cells in X, Y and Z.
4. Click on 'Apply' to implement settings.
5. Click on Time dependence 'Steady' to toggle to 'Transient'.
6. Click on 'Time step settings'.
7. Set:
    - Starting time: 0.0
    - Duration: 1.0
    - Number of steps: 20
8. Click on 'Apply' to implement settings.
9. Click on 'OK' to close the 'Time step settings' window.
10. Click on 'OK' to close the 'Geometry' window.
11. Click on 'OK'.

---

## Step 2: Modelling settings

1. Click on 'Models'.
2. Set the 'Solution for velocities and pressure' ON.
3. Click on 'Laminar' of 'Turbulence models'.
4. Locate 'CONSTANT-EFFECTIVE' to select the model of effective turbulent viscosity.
5. Click on 'OK' to implement the choice.

6. To activate MOFOR and read-in BVH file for object movement, click on 'Sources', then on 'OFF' next to MOFOR. A file browser will appear. Select the file AGITATED.BVH and click 'Open'.
7. Click on 'Settings' of 'Solution control/Extra variables'.
8. To activate solution for concentration field:
    - Type C1 in the field of 'SOLVE'.
    - Click on 'Apply' to implement the setting.
    - Click on 'Previous panel'

---

## Step 3: Domain properties

1. Click on 'Properties'
2. Set 'Use property tables' ON.
3. Make sure that 'The current domain material is' air at 20 deg c, 1 atm.

---

1. Click on 'Initialisation'.
2. Set 'FIINIT Value' to 0.0 for all variables.
3. Click on 'Apply' to implement the settings.

---

## Step 5: Numerics ( solution controls)

1. Click on 'Numerics'.
2. Change 'Total number of iterations' to 20.
3. Click on 'Relaxation control'.
4. Change 'DTFALS' for U1,V1,W1 and C1 to 1.
5. Click on 'Previous panel'.

---

## Step 6: Output settings

1. Click on 'Output'.
2. Click on 'Settings' of 'Field dumping'.
3. Change 'Step frequency' to 1.
4. Set 'CSG1' of 'Start letter for PHI' to M.
5. Click on 'Previous Panel'.
6. Click on 'Top Menu'.
7. Click on 'OK'

---

## Step 7: Monitoring locations and view

1. Change X-, Y- and Z- of 'Position' to 0.1
2. Click on 'Reset'.
3. Change X, Y and Z of 'View direction' to 0.32, 0.78 and -0.54, respectively.
4. Change X, Y and Z of 'Up direction' to 0.95, -0.24 and 0.22, respectively.
5. Change X, Y and Z of 'View center' to 0.25
6. Change 'View angle' to 3.
7. Change 'View size' to 0.39
8. Click on 'OK'.
9. Click here to see the screen image.

---

## Step 8: Pressure relief

1. Click on 'Obj'.
2. Change 'Name' to RELIEF.
3. Click on 'BLOCKAGE' of 'Type'.
4. Locate 'PRESSURE_RELIEF'.
5. Click on 'OK'.
6. Click on 'OK'.
7. Click here to see the screen image.

---

## Step 9: Initial concentration

1. Click on 'Obj'.
2. Change 'Name' to FLU1IN.
3. Set:
    - XPos = 0.0 ; XSize = 0.2
    - YPos = 0.0 ; YSize = 0.5
    - ZPos = 0.0 ; ZSize = 0.5
4. Click on 'Apply'.
5. Click on 'Default' of 'Geometry'
6. Go into 'Shapes'.
7. Locate 'cubet.dat'.
8. Click on 'Open'.
9. Click on 'Colour'.
10. Click on 'Default' of 'Colour mode'.
11. Click on 'Based on default' to toggle to 'User'.
12. Click on 'Solid' of 'Transparency' to toggle to 'Transparent'.
13. Change 'Colour value' to 5.
14. Click on 'OK'.

15. Click on 'BLOCKAGE' of 'Type'.
16. Locate 'USER_DEFINED'.
17. Click on 'OK'.
18. Click on 'Attributes'.
19. Change 'New' to INIF1.
20. Click on 'Page Dn'.
21. Click on 'Apply'.
22. Click on 'Page Up'.
23. Change 'Type' to INIVAL.
24. Change 'Coefficient' of C1 to 0.0.
25. Change 'Value' of C1 to 1.0.
26. Click on 'Page Dn'.
27. Click on 'Apply'.
28. Click on 'Page Up'.
29. Click on 'OK'.
30. Click on 'OK'.
31. Click here to see the screen image.

---

## Step 10: Initial position of SHAFT object

1. Click on 'Obj'.
2. Change 'Name' to SHAFT.
3. Set:
    - XPos = 0.15 ; XSize = 0.35
    - YPos = 0.225 ; YSize = 0.05
    - ZPos = 0.225; ZSize = 0.05
4. Click on 'Apply'.
5. Click on 'Default' of 'Geometry'
6. Go into 'Shapes'.
7. Locate 'cylinder.dat'.
8. Click on 'Open'.
9. Click on 'Rotation' of 'Geometry'
10. Change 'Clip-art' to 9/24.
11. Click on 'OK'.
12. Click on 'Attributes' of 'Type'.
13. Click on 'Other materilas' of 'Types'.
14. Locate 'Domain material'.
15. Click on 'OK'.
16. Click on 'OK'.
17. Click on 'Yes' of 'Objects affects the grid' to toggle to No.
18. Click on 'OK'.
19. Click here to see the screen image.

---

## Step 11: Initial position of ROD object

1. Click on 'Obj'.
2. Change 'Name' to ROD.
3. Set:
    - XPos = 0.125 ; XSize = 0.05
    - YPos = 0.125 ; YSize = 0.25
    - ZPos = 0.225; ZSize = 0.05
4. Click on 'Apply'.
5. Click on 'Default' of 'Geometry'
6. Go into 'Shapes'.
7. Locate 'cylinder.dat'.
8. Click on 'Open'.
9. Click on 'Rotation' of 'Geometry'
10. Change 'Clip-art' to 5/24.
11. Click on 'OK'.
12. Click on 'Attributes' of 'Type'.
13. Click on 'Other materilas' of 'Types'.
14. Locate 'Domain material'.
15. Click on 'OK'.
16. Click on 'OK'.
17. Click on 'Yes' of 'Objects affects the grid' to toggle to No.
18. Click on 'OK'.
19. Click here to see the screen image.

---

## Step 12: Initial position of PADDLE1 object

1. Click on 'Obj'.
2. Change 'Name' to PADDLE1.
3. Set:
    - XPos = 0.111 ; XSize = 0.1
    - YPos = 0.375 ; YSize = 0.1
    - ZPos = 0.287; ZSize = 0.025
4. Click on 'Apply'.
5. Click on 'Default' of 'Geometry'
6. Go into 'Shapes'.
7. Locate 'cubet.dat'.
8. Click on 'Open'.
9. Click on 'Colour'.
10. Click on 'Default' of 'Colour mode'.
11. Click on 'Based on default' to toggle to 'User'.
12. Change 'Colour value' to 7.
13. Click on 'OK'.
14. Click on 'Rotation' of 'Geometry'

15. Change 'Beta' to 45 degree.
16. Click on 'OK'.
17. Click on 'Attributes' of 'Type'.
18. Click on 'Other materilas' of 'Types'.
19. Locate 'Domain material'.
20. Click on 'OK'.
21. Click on 'OK'.
22. Click on 'Yes' of 'Objects affects the grid' to toggle to No.
23. Click on 'OK'.
24. Click on 'Edit'.
25. Click on 'Object attributes'.
26. Locate "FLU1IN".
27. Click on 'OK'
28. Click on 'Hide'
29. Click here to see the screen image.

---

## Step 13: Initial position of PADDLE2 object

1. Click on 'Obj'.
2. Change 'Name' to PADDLE2.
3. Set:
    - XPos = 0.132 ; XSize = 0.1
    - YPos = 0.025 ; YSize = 0.1
    - ZPos = 0.204; ZSize = 0.025
4. Click on 'Apply'.
5. Click on 'Default' of 'Geometry'
6. Go into 'Shapes'.
7. Locate 'cubet.dat'.
8. Click on 'Open'.
9. Click on 'Colour'.
10. Click on 'Default' of 'Colour mode'.
11. Click on 'Based on default' to toggle to 'User'.
12. Change 'Colour value' to 7.
13. Click on 'OK'.
14. Click on 'Rotation' of 'Geometry'
15. Change 'Beta' to -45 degree.
16. Click on 'OK'.
17. Click on 'Attributes' of 'Type'.
18. Click on 'Other materilas' of 'Types'.
19. Locate 'Domain material'.
20. Click on 'OK'.
21. Click on 'OK'.
22. Click on 'Yes' of 'Objects affects the grid' to toggle to No.
23. Click on 'OK'.

24. Click here to see the screen image.

---

## Step 14: Running the simulation

1. Click on 'Run'.
2. Click on 'Earth'.
3. Click on 'OK'.

---

## Step 15: Displaying flow and mixing

1. *Postprocessing **at t=0.5** seconds*
    1. Click on 'Run'.
    2. Click on 'VR-Viewer'.
    3. Click 'No' of 'User intermediate step files'.
    4. Change 'Plot step number' to 10.
    5. Click on 'OK'.
    6. Change 'Probe position' X to 0.15
    7. Click on 'V'.
    8. Click on 'C'.
    9. Change 'Velocity vector scale' to 0.04'
    10. Click on 'Vector toggle' to see the flow vectors.
    11. Click on 'Y'.
    12. Click on 'C'.
    13. Locate 'C1'.
    14. Click on 'OK'.
    15. Click on 'Contour toggle' to see concentration contours.
2. *Moving to **t=1.0** seconds*
    1. Click on 'F6'.
    2. Change 'Plot step number' to 20.
    3. Click on 'OK'.
3. *Moving to the **previous time** moment*
    1. Click on 'F7'.
    2. Click on 'OK'.

---

## Step 16: Enabling and Solving for Chemical Reaction

1. Click on 'Run'.
2. Click on 'VR Editor'.
3. Click on 'Menu'.
4. Click on 'Models'.

5. Click on 'OFF' of 'Energy Equation'
6. Locate 'Ethalpy'.
7. Click on 'OK'.
8. Click on 'StATIC' to toggle to 'TOTAL'
9. Click on 'OFF' of 'Combustion/Chemical Reaction'.
10. Locate '3 gases mixing'.
11. Click on 'OK'.
12. Click on 'Settings' of '3_GASES_MIXING'.
13. Change 'Stoichiometric ratio' to 1.
14. Change 'Heat of combustion' to 0.
15. Change 'Specific Heats' to 1005.
16. Change 'Molecular Weights' to 29.
17. Click on 'Previous panel'.
18. Click on 'Properties'.
19. Click on 'The current domain material is'.
20. Locate 'Gases'.
21. Click on 'OK'.
22. Locate 'Air at 20 deg C'.
23. Click on 'OK'.
24. Change 'Reference pres (Pa)' to 100000.
25. Change 'Reference temp (K)' to 273.
26. Click on 'Initialisation'.
27. Change 'FIINIT Value' of H1 to 301500.
28. Change 'FIINIT Value' of MIXF to 0.
29. Click on 'Apply'.
30. Click on 'Numerics'.
31. Click on 'Relaxation control'.
32. Change 'DTFALS' of H1 to 1.
33. Change 'DTFALS' of MIXF to 1.
34. Click on 'Previous Panel'.
35. Click on 'Top Menu'.
36. Click on 'OK'.
37. Click on 'Edit'.
38. Click on 'Object attributes'.
39. Locate 'FLU1IN'.
40. Click on 'OK'.
41. Click on 'Attributes'.
42. Change 'Coefficient' of MIXF to 0.
43. Change 'Value' of MIXF to 1.
44. Click on 'OK'.
45. Click on 'OK'.
46. Click on 'Run'.
47. Click on 'Earth'.
48. Click on 'OK'.

## Step 17: Enabling Higher-Order Descritization for C1

1. Click on 'Menu'.
2. Click on 'Numerics'.
3. Click on 'Differencing Schemes'.
4. Click on 'Set schemes individually'.
5. Locate 'Variable' of C1.
6. Click on 'UPWIND' of 'SCHEME'.
7. Locate, say, 'MINMOD'.
8. Click on 'OK'.
9. Click on 'Previous Panel'.
10. Click on 'Top menu'.
11. Click on 'OK'.
12. Click on 'Run'.
13. Click on 'Earth'.
14. Click on 'OK'.

## Step 18: Enabling re-start

1. Click on 'Menu'.
2. Click on 'Initialisation'.
3. Click on 'Restart for all variables'.
4. Click on 'Top menu'.
5. Click on 'OK'.
6. Click on 'Edit'
7. Click on 'Object attributes'
8. Locate 'FLU1IN'.
9. Click on 'OK'.
10. Click on 'Delete'.
11. Click on 'Yes'.
12. Click on 'Run'.
13. Click on 'Earth'.
14. Click on 'OK'.

## Summary

In this tutorial, you have modelled the time-dependent flow involving moving-impeller-fluid interaction. You have learned how to initialise the positions, set the attributes of the moving bodies and activate MOFOR to handle their effects on the flow field. Similar procedures

can be used for any further applications involving moving bodies.

You have used **PHOENICS**'s time-dependent flow prediction capability, and you have learned how to set solution parameters. These time-dependent flow prediction procedures can also be applied to other, non-moving-body, analyses.

You have modelled the transport, mixing, and reaction of chemical species. The reaction system was defined by using and modifying a SCRS entry in the **PHOENICS** model database. The procedures used here can be applied to other reacting flow sys ems.

You have also learned how to use the graphics of VR-viewer to analyse the results of transient simulations. This exercise has illustrated how to use the higher-order discretization schemes and re-start capability for longer, multi-stage simulations using the cyclic nature of the flow-inducing body movements.

---

Home Page

# WORKSHOP - PLANT

## Introduction

There follow two workshop sessions which create PLANT settings for:-

1. Non-linear properties
2. The use of PLANT to set a linear inlet velocity profile

# Workshop 1 - Non-linear Properties

This case analyses the flow in a cavity, the top wall of which moves with a constant velocity. The top wall is at one temperature, the moving wall is at a different temperature and two other walls are adiabatic. The calculation is made for the Reynolds number Re = 100 , for Prandtl number Pr = 1 and for the exponent in the viscosity formula EMU = EMU0*EXP(-beta.(T - T0)).
In this case, EMU0 = 1E-3, beta=1.6 and T0 = 0.0.
( See M. Morzynski and Cz. O. Popiel in Numerical Heat Transfer, v.13, pp. 265-273, 1988 for further details and comparison).

**Accessing PHOENICS-VR.**

*From the system level:*

To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.

*In PHOENICS-VR environment,*

Start with an ‘empty’ case - click on 'File' then on 'Start New Case', then on 'Core’, then click on 'OK'; to confirm the resetting.

*To enter VR Editor:*

This is the default mode of operation.

## Within VR Editor.

**Set the domain size and activate solution of the required variables**:

Click on 'Main Menu' and then on 'Geometry'. Set the domain size to 0.1 * 0.1 * 0.1. Click 'OK' to close the Grid mesh dialog.

Click on 'Models'. Set the 'solution for velocities and pressure' ON.

Set the Energy Equation to TEMPERATURE.

Click on 'Properties', then 'Storage'. Turn the storage of Viscosity ON, so that it can be plotted after solution has been obtained. Click on 'Edit properties of current material'. Click on 'Conductivity', and select 'Prandtl-Number' from the list of formulae. Set the Prandtl Number to 1.0.

Click on 'Previous panel', 'Top menu', and on 'OK' to exit the Main Menu.

**Create upper, moving wall**:

Click on 'New Object'.

Change name to MOVING.

Set position and size of object as:

Xpos: 0.0 Xsize: 0.1

Ypos: 0.0 Ysize: 0.1

Zpos: 0.1 Zsize: 0.0

Define Type as Plate.

Click on 'Attributes'. Set the Slide Velocity to 1.0 in the X direction. Click on 'Energy Source - Adiabatic'. Select Surface temperature. Set the surface temperature to 1.0. Click on 'OK' to close the attributes dialog box.

Click on 'OK' to close the Object Dialogue Box.

**Create the lower stationary wall** by duplicating MOVING:

Click on the ‘Duplicate object or group’ button on the handset.

Click on the image of the copy. This opens the Object Dialogue Box of the object copy. Change name to FIXED.

Click on 'Attributes'. Set the Slide Velocity to 0.0 in the X direction. Set the surface temperature to 0.0. Click on 'OK' to close the attributes dialog box.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the right adiabatic wall**:

Click on 'New Object'.

Change name to RIGHT.

Set position and size of object as:

Xpos: 0.1 Xsize: 0.0

Ypos: 0.0 Ysize: 0.1

Zpos: 0.0 Zsize: 0.1

Define Type as Plate.

Click on 'OK' to exit from the Object Dialogue Box.

**Create the left adiabatic wall** by duplicating RIGHT:

Click on the ‘Duplicate object or group’ button.

Click on the image of the copy. Change name to LEFT.

Click on 'OK' to exit from the Object Dialogue Box.

**Create a fixed-pressure point in the middle:**

Click on 'New Object'.

Change name to FIXEDP.

Set position and size of object as:

Xpos: 0.05 Xsize: 0.005

Ypos: 0.0 Ysize: 0.1

Zpos: 0.05 Zsize: 0.005

Define Type as Pressure_Relief.

Click on 'OK' to exit from the Object Dialogue Box.

**Set the grid**:

Click on the 'Mesh toggle' button. The default mesh (shown as orange lines) will appear on the screen. Click anywhere on the image, and the ‘Grid mesh settings’ dialog box will appear. Change the Number of cells to 15 in X and 15 in Z. Set Power/ratio for both X and Z to 2.0, and Symmetric to Yes. Click on 'OK'; to close the dialog box. The mesh will now be shown as blue lines. Click on ‘Mesh toggle’ again to turn off the mesh display.

**Make the PLANT settings:**

Click on 'Run', 'Pre processor' then 'Fortran creator (Plant menu)'. Click on 'Properties of the medium', then click 'Add'. On the next panel, scroll the list of available properties down until 'laminar kinematic viscosity' appears. Click on it to select it.

In the 'PLANT statements' box, enter the formula:

VISL=1.E-3*EXP(-1.6*TEM1)

Click OK to set the expression, then Exit to quit the PLANT menu.

**Make the solution control settings:**

Click on 'Main menu', then on 'Numerics'. Set the total number of iterations to 100. Click on 'Relaxation'. Set the Self-Adjusting Relaxation algoritHm (SARAH) parameter to 0.005, and the relaxation value for TEM1 to 1000.

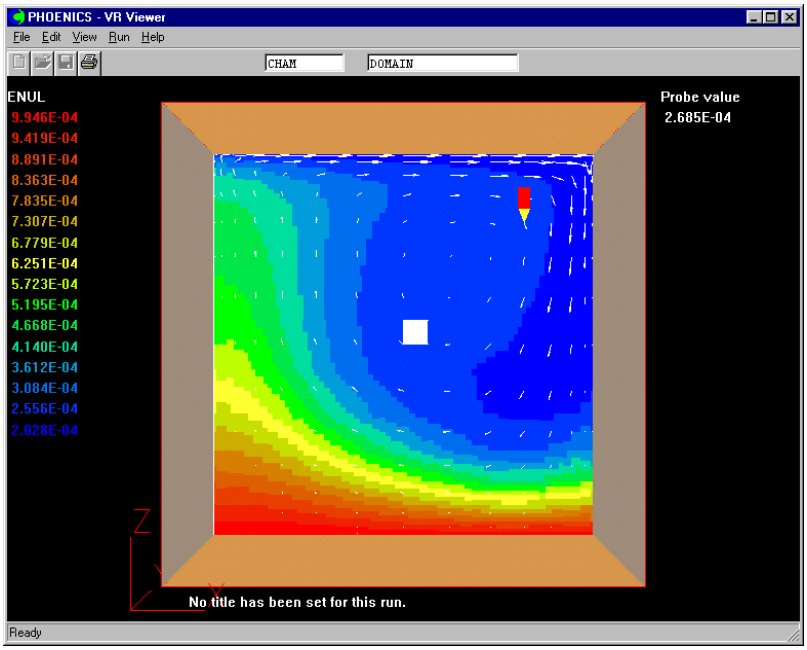
Click 'Previous panel', 'Top menu' and 'OK' to exit the Main menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Options', then 'Run version'. For Earth, select 'Prompt'. Each time the Earth solver is run, you will be asked whether to run the 'Public' or 'Private' version. The Public Earth is the standard delivery version. The Private Earth is the local modified copy.
Click on 'Run', 'Solver' then 'local solver (Earth)'. Select 'Private'. Allow the compilation and re-linking to proceed.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)' and view the velocity vectors, contours and isosurfaces. Click on C to select the viscosity variable, ENUL. (Hint – change the slice direction to Y)




# Workshop 2 - linear inlet profile

This workshop shows the user how to set a linear inlet velocity profile using PLANT. It is based in library case 240. The library case is 2 dimensional in the Y-Z plane, with Z as the main flow direction. The inlet velocity is uniform at 5.0, and the inlet enthalpy is 9.0.
The inlet mass-flow will be calculated from:

Min = 5.0*RHO1 *(YVLAST-YG2D))

and the inlet velocity from:

Win = 5.0*(YVLAST-YG2D)

The inlet enthalpy will be constant at 9.0.
Here, YVLAST is the domain height, and YG2D is the index of the store containing cell-centre locations in the Y direction.

**Accessing PHOENICS-VR.**

*From the system level:*

To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.
In PHOENICS-VR environment,

Load library case 240 - click on 'File' then on 'Load from Libraries'. Enter 240 in the case number box, then click on 'OK'; to confirm the loading.
*To enter VR Editor:*
This is the default mode of operation.

## Within VR Editor.

**De-activate the INLET object** - it will be replaced by PLANT coding.

> Click on 'Edit', then 'Find object'. Select IN from the list and click 'OK'. This makes IN the currently-selected object. Now click on 'Edit' then 'Object attributes'. Click on 'Type', and select 'NULL'. Click 'OK' to close the Object dialog box. The inlet object will be replaced by a null object. The idea is to preserve the grid, but eliminate any onther effect of the original object.

### Refine the grid.

> Click on 'Grid mesh', then anywhere on the domain. Change the number of cells in Y and Z from 5 to 15. Click 'Ok' to close the mesh dialog, and 'Grid mesh' again to hide the grid display.

### Modify the numerical controls

> The library case does not do enough iterations to procure convergence on a finer grid. Click on 'Main menu', then 'Numerics'. Reset the total number of iterations to 200. Click on 'Relaxation', and set the Self-Adjusting Relaxation algoritHm (SARAH) parameter to 0.005.

### Activate graphical monitoring of convergence

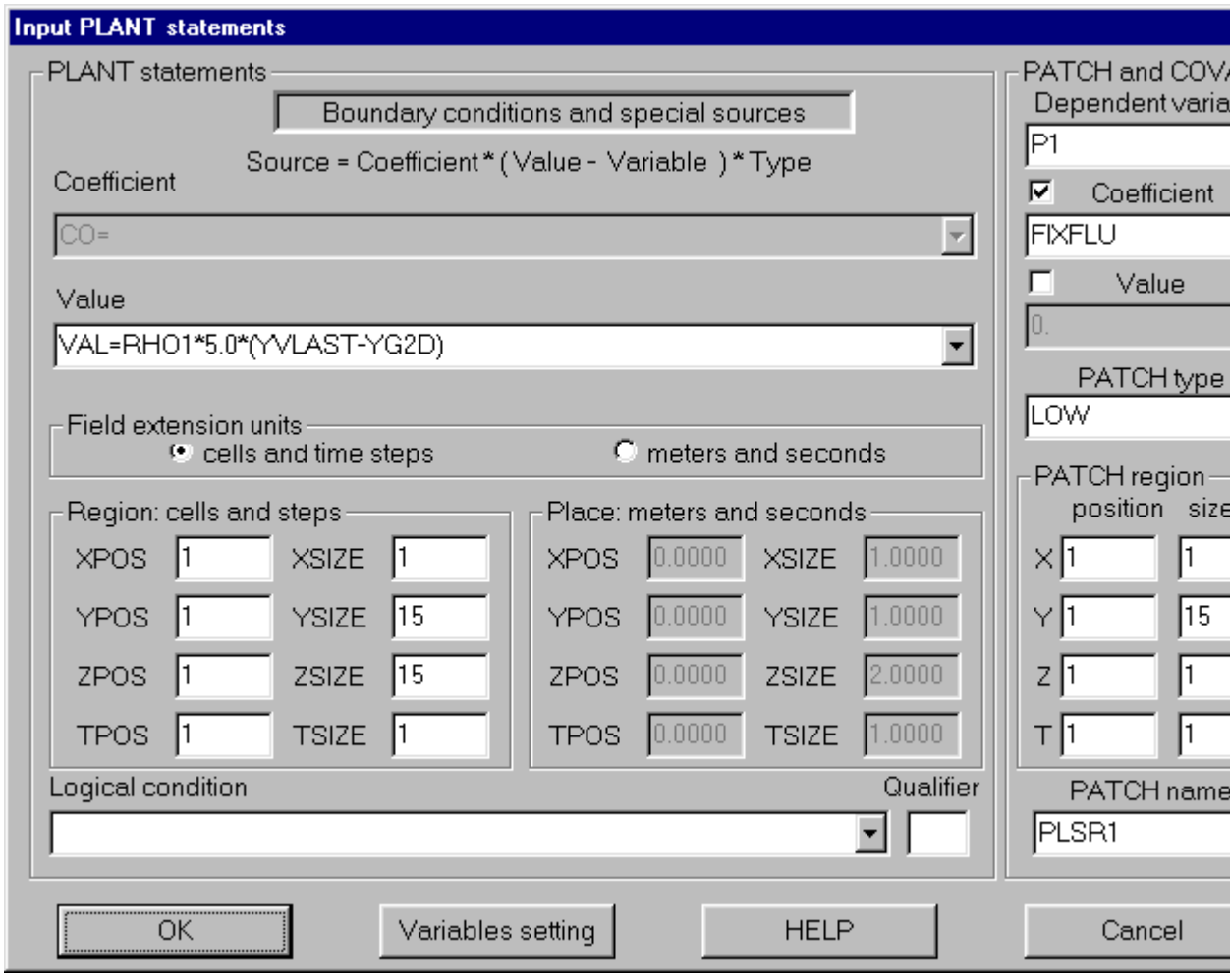
> Click on 'Previous panel', then 'Output'. Toggle the Monitor display mode from 'ASCII' to 'GRAPHICS'.

**Reset the Outlet** Coefficient

Click twice on the OUTLET object, then on 'Attributes'. Change the Coefficient from 1.0 to 1000. This will keep the exit pressure closer to zero.

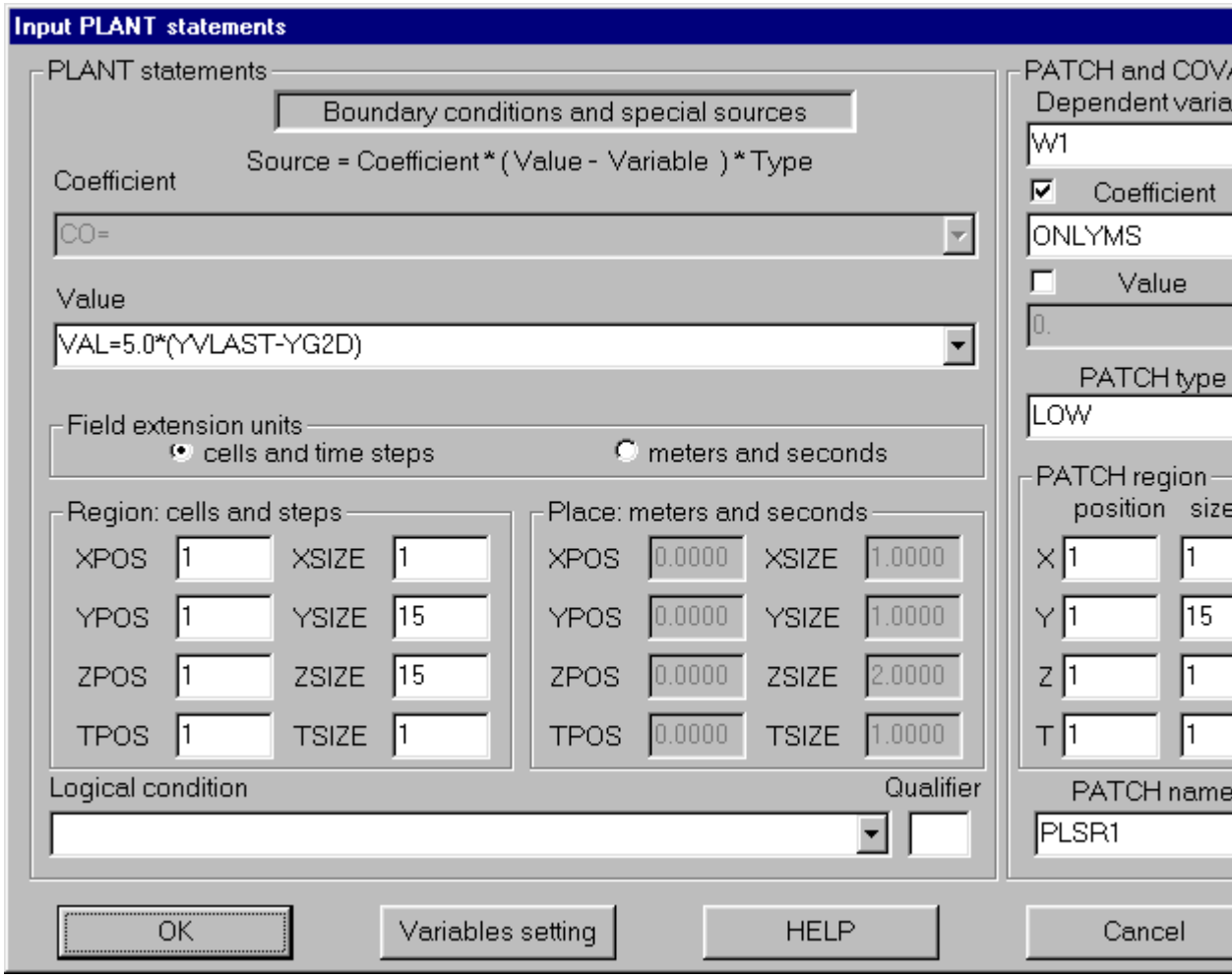
**Make the PLANT settings:**

Click on 'Run', 'Preprocessor' then 'Plant menu'. Click on 'Boundary conditions and special sources', then on 'Add'. Make settings for P1 as shown in the image below. First select P1 from the list of dependent variables - the other input boxes will not become 'live' until a variable has been chosen.

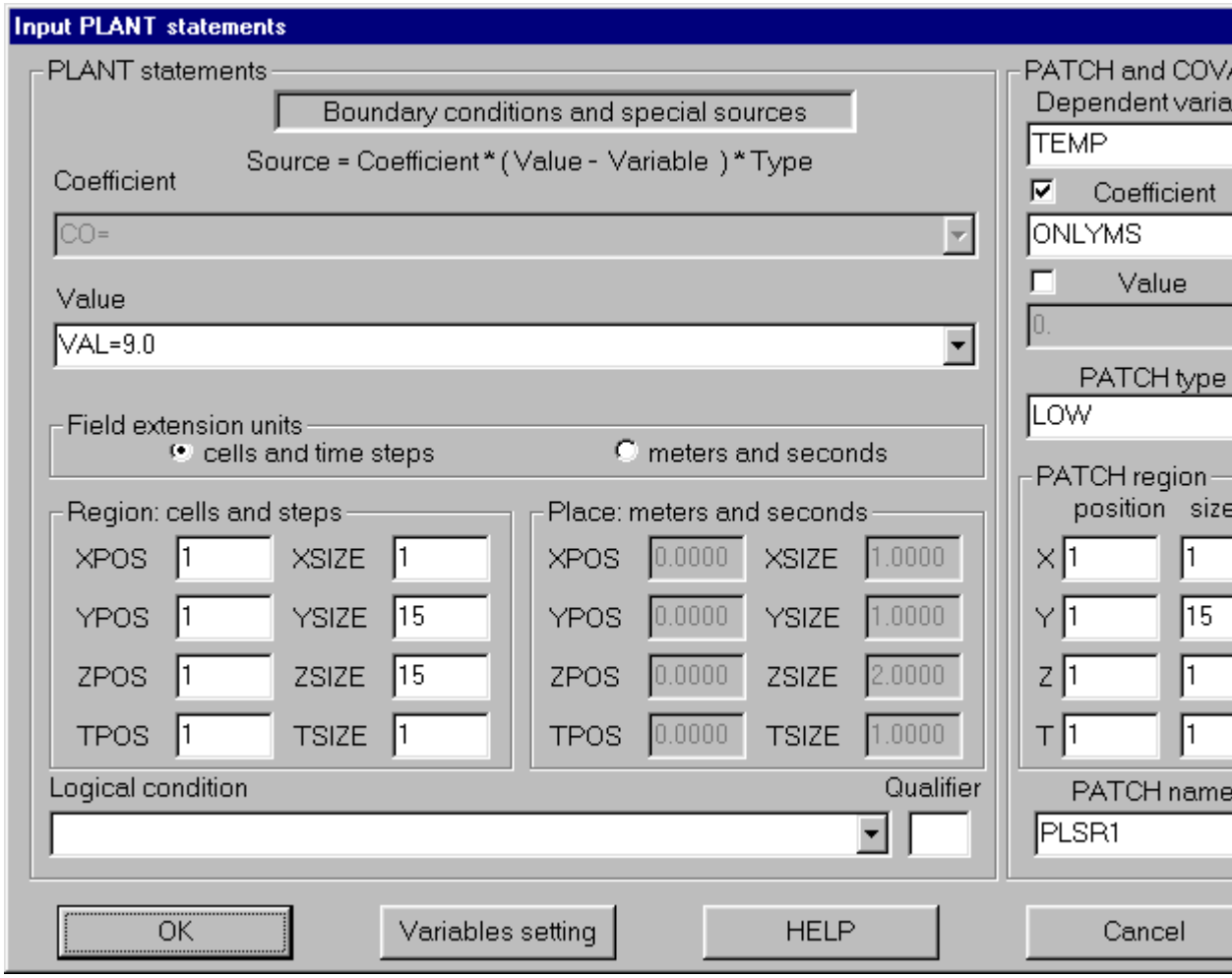



Click 'OK' to close the dialog. Click 'Add' again, and make settings for W1 as shown below. Normally, each boundary condition is

applied as a separate PATCH. In this case, we want the mass flow (P1), velocity and enthalpy conditions to be set by the same PATCH. Change the PATCH name in the bottom-right corner to PLSR1 - the name used for the P1 PATCH.




Click 'OK' to close the dialog. Click 'Add' again, and make settings for TEMP as shown below. Remember to change the PATCH name to PLSR1.



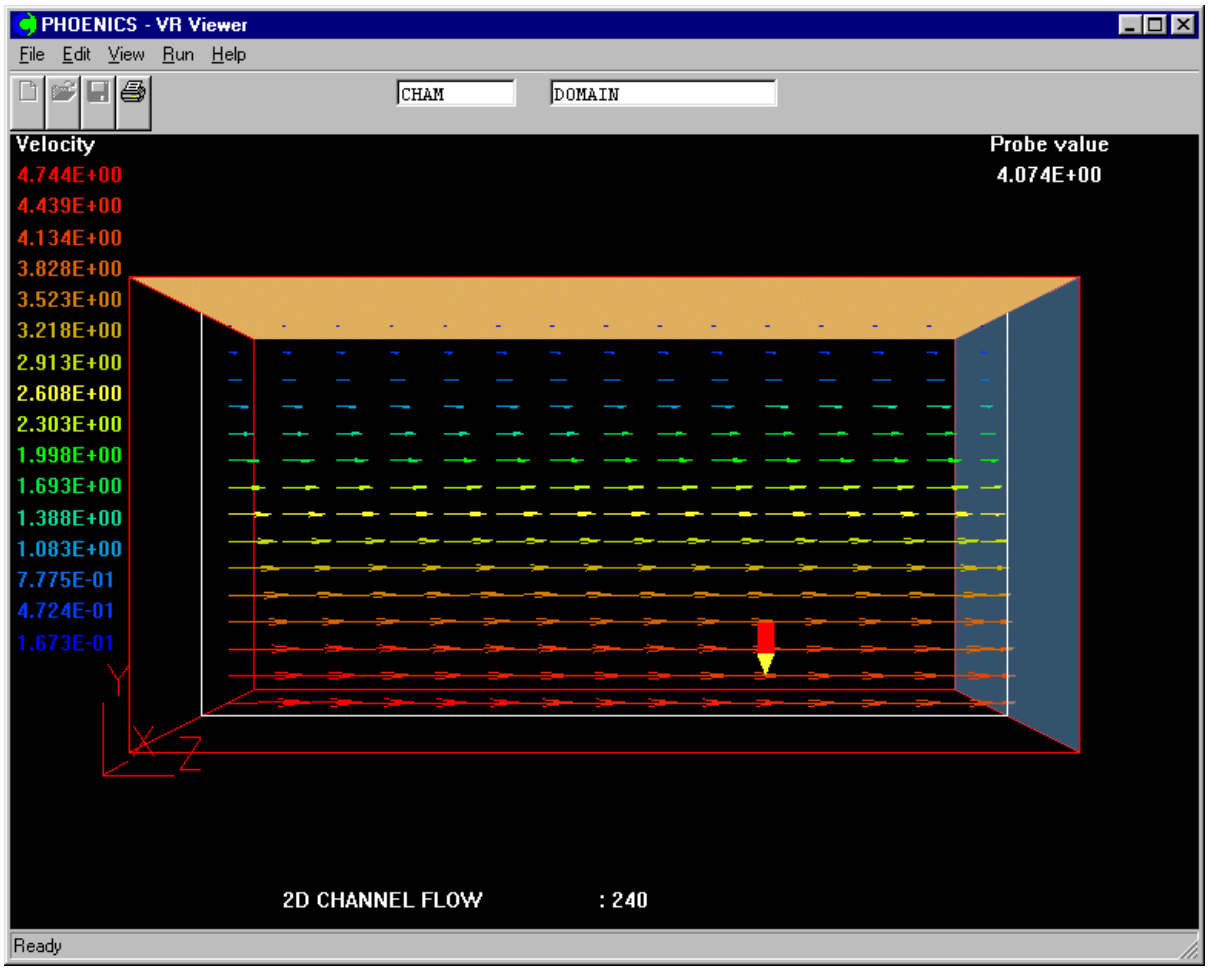

Click 'OK', then 'Exit' to leave the Plant menu.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Options', then 'Run version'. For Earth, select 'Prompt'. Each time the Earth solver is run, you will be asked whether to run the 'Public' or 'Private' version. The Public Earth is the standard delivery version. The Private Earth is the local modified copy.
Click on 'Run', 'Solver' then 'Local solver (Earth)'. Select 'Private'. Allow the compilation and re-linking to proceed.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)' and view the velocity vectors, contours and isosurfaces. (Hint - set the slice direction to X).



Home Page

# WORKSHOP - Modifying GROUND

This session is intended to provide hands-on experience of the expansion of PHOENICS capabilities through the insertion of FORTRAN coding. The structure of GROUND is described here, and the use of GROUND is explained here.
During this session, the Ideal Gas law for density will be coded into GROUND. This is, in fact, a built-in option in GREX, but it is representative of the kind of modification typically effected in GROUND. Furthermore, its availability in GREX will allow an easy check of the correctness of the new coding, by simple comparison of results.

## Introduction

The Ideal Gas law for density is:

$$\rho = P / (R * T)$$

where: R - is the gas constant for the particular gas. It will have to be passed from Q1, using, say, RG(1). The value for air is 286.4.
P - is the pressure. Note that in PHOENICS the pressure is obtained by adding the constant PRESS0 to the values held in the P1 store.
T - is the temperature. The integer index used to reference cell-by-cell values of temperature is TEMP1.
In the SI unit system, R is in J/kg/K, P is in N/m**2 and T is in K.

## Insertion of GROUND Coding

An empty, or model, GROUND is provided for the user's private use by clicking on the EDIT button of the PHOENICS-VR Environment and then clicking on Ground.

A model GROUND will appear on the screen in Notepad, and can now be edited. On saving, this GROUND will normally reside in the D_PRIV1 directory.
Insert GROUND coding for the Ideal Gas law by use of two different techniques:

1. The FN function subroutines, which operate directly on the F array. See the FUNCTion library entry in POLIS for a complete list of FN subroutines; and

2. Direct access to the F array, using the L0F function to obtain starting locations.

The entry to GROUND for density will be activated in Q1 by setting RHO1=GRND. (In VR, this means selecting USER for the density formula.)

## Compiling and Linking

The new GROUND has to be compiled before a new EARTH can be built. Compilation is performed via the COMPILE / Ground menu of VR-Environment, and building through BUILD / Earth.
If a 'private' Earth is selected to be run (see below), and the GROUND file is newer than the Earth executable, the option of recomiling and relinking will be offered.

## Testing the Coding

The coding will be tested by comparing the results generated by the GREX option and the new coding. Library case B529 (which is concerned with supersonic flow around a rocket) happens to use the Ideal Gas Law, and hence can be used for testing. The following steps should be followed:

1. VR must be turned off to enable the multi-run to work. Click on **Run**, then **Commander**. From this environment, any of the PHOENICS modules can be run.

2. Click on **Run**, **Pre processor**, **Text mode (Satellite) - T** and load library case B529. This can be done from command mode (F2) by typing LOAD(B529), or from the Top Menu, by going to Library (F4), selecting '2 Load specified case' and entering B529. End (F7), saving the input file.
3. Edit the Q1- click on File, Open file for Editing, select Q1. Modify the first line to read RUN(1,3) instead of RUN(1,1). This indicates that three EARTH runs will be made in one submission. This is called a multi-run. See the Encyclopaedia entry on **RUN** for more information.
4. Change LSWEEP from 100 to 2 in order to reduce run time. We are not interested in converged solutions, just identical solutions!
5. Before the STOP at the bottom of the file, insert **NOWIPE**=T. This will prevent the input data being reset to default values for subsequent runs. Each run then becomes a parameter variation on the first.
6. By default, EARTH writes out the flow field into a file called PHI (or PHIDA). In this case, each run would overwrite the PHI file from the previous run. This can be avoided by changing the name of PHI for each run. This is done by setting **NSAVE**=RUN1 (say) for the first run.
7. This case happens to set **USEGRD** =F. Delete this line, otherwise the new GROUND coding will never be executed!!

The settings before the STOP define the default case, against which we will test the new coding.

8. After the STOP, insert RHO1=GRND, which will switch off the built-in coding, and activate the new coding. Set RG(1) to the required gas constant.
9. Set IG(1)=1 to elect the first method of calculation, and set NSAVE=RUN2 (say) to prevent the PHI file being overwritten. The name of the PHI file will be changed to RUN2. You may also wish to change the TEXT

command to identify the particular case. Finally, add another STOP to indicate the end of the data settings for run 2.

10. Repeat for the other coding style option, setting IG(1) and NSAVE appropriately.

The bottom of the Q1 should look like this:

```
 GROUP 23. Field print-out and plot control

NPRINT=LSWEEP;NYPRIN=1;NZPRIN=2

PATCH(YZ,CONTUR,1,1,1,NY,1,NZ,1,1)

PLOT(YZ,P1,0.0,20.0)

LIBREF=529



 #### new lines added for GROUND Workshop

USEGRD=T

NOWIPE=T

LSWEEP=2

NSAVE=RUN1

STOP



 #### Data for Run 2

RHO1=GRND

RG(1)=286.4

IG(1)=1

NSAVE=RUN2

STOP
```

```
#### Data for Run 3

IG(1)=2

NSAVE=RUN3

STOP
```

## Running the Test Cases

Run the Satellite again to process the modified input file. We now need to run the local 'private' Earth, not the default 'public' one used up to now. To do this:

- click on the Options button of the PHOENICS-VR Environment
- click on run Version
- click on Earth
- Click on Private

Run the solver by clicking on the Run - Solver - Local solver (Earth) button of the PHOENICS Commander. Note the residuals for each case. They should be the same, regardless of the coding style.
The RESULT file produced will contain the output from all the runs, one after the other. Inspection of the density fields will show whether or not they are identical.
Also, the PHI files produced can be accessed through AUTOPLOT, and the results compared graphically. Again they should produce identical plots.

## Solution - required GROUND Coding

You should try to avoid looking at **this page** unless you are **completely** stuck.

Home Page

# WORKSHOP - Boundary Conditions and Sources

## Objectives of this workshop session

This workshop is intended to provide extensive practice with boundary conditions and sources. The following topics are covered:

- The specification of **inlet conditions**, and particularly the relationship between inlet conditions for mass (mass flux) and velocity.
- The use of **FIXVAL** and **FIXFLU**.
- The specification of **outlet conditions** (use **FIXP**).
- The use of boundary conditions and sources of the "wall" type.

For some of the steps involved, either the solution or a helpful hint is given at the end of this document. (Such steps are marked **see note** in the main text). However, participants are advised to try for themselves before turning to the solution.
Library cases are loaded by clicking on File, Load from libraries, and entering the required case number in the 'Case number' data entry box.
Results are saved by clicking on File, Save as a case.
If you prefer to do the tutorial in 'Command mode' using PIL, click on 'File', 'Start New Case', then on 'Core' to get an 'empty' Q1. Now, instead of loading the library cases as above, click on Run, Satellite to run in command mode.
The Q1 file can be edited by clicking on File, Open file for editing, Q1.
The 'Hints and Solutions' section gives answers for both VR and command modes.

## FIXVAL and FIXFLU

Load core library case 240 (2D channel flow) and learn about the case through VR Editor and the main menu, or by editing the Q1 file. Change the surface enthalpy of the plate from 0 J/kg to 5 J/kg. Activate the graphical convergence monitor. (see note 1).
Run the case, and inspect the results (particularly, the Z-direction velocity). See how the VR settings in the Q1 have been translated into PIL settings in the RESULT file. You might want, at this stage, to save the Q1, PHI and RESULT files as a new case, for future reference.
Set a heat flux along the wall, with a value of 5 $W/m^2$ (see note 2). Run the case, and look in the RESULT file. The net source of enthalpy (re**NAME**d to **TEMP**) reported in the RESULT file (lines starting NET SOURCE OF) for this **PATCH** is 10.0 why? (see note 3). Save the Q1, PHI and RESULT files.
In the original Q1 file for this case, fix the enthalpy in the near-wall cells to 5 J/Kg. (see note 4). How is the solution changed, and why? Check the net source printout in the RESULT file. Why has the H1 source for this object vanished, and can anything be done about it? (see note 5.
What is the difference between the original case, and the two versions above? (note 6).

## Wall-type boundary-conditions

Load core library case 921 using the VR Editor and the 2D menu. Learn about the case through VR Editor and the main menu, or by inspecting the Q1 file. Activate the graphical convergence monitor, then run the case and save the results as before. (See note 7)
In the RESULT file, find the net momentum source for the moving wall. Why is it negative? (see note 8)
Change the direction / and / or speed of the moving wall. Make some or all of the other walls move. See what happens to the signs of the wall momentum sources when the flow direction is reversed.

### Inlet boundary-conditions

Load library case 240 again. Change, in the inlet boundary-condition, the **VALUE** command for **W1** to a **COVAL** command with **FIXVAL** as coefficient (see note 9). Then run the case. What is the difference with the standard case (see note 10). Why is this?
In a new Q1 file for case 240, set the density to 5.0 (see note 11.) Then run the case and check, in the RESULT file, the new velocity and pressure distribution. What has happened and why? (see note 12). What corrections should be made in VR-Editor or the Q1 file to avoid this? (See note 13)
In a fresh copy of the Q1 file for this case, change the input data where needed so that the inlet velocity is 10.0 (see note 14). Run the case and check the RESULT file.

### FIXP (outlet) boundary-conditions

Locate, in the RESULT file for case 240, the outlet boundary-condition. Which kind of boundary condition is it (see note 15)
Run the standard case and examine the RESULT file: is the exit pressure close enough to 0.0? If not, why not and how can you tell? (see note 16). Modify the case so as to bring the outlet pressure closer to 0.0, and check the results. (see note 17)
Increase the number of grid cells to 10 x 10. Then modify the outlet boundary condition, so that it only covers the upper half of the duct. (see note 18). Run the case and examine the results with VR-Viewer. What happens and why? (see note 19)

## Hints and Solutions

### Note 1.

### VR Editor mode:

Click on the wall object to give an Object Dialogue Box.
Click on Attributes. Enter 5.0 in the 'Value' data entry box.
Click on Main menu / Output / ASCII (this will toggle to GRAPHICS)

## Command mode:

Group 22

Set TSTSWP=-1
The surface enthalpy would be set by a PATCH/COVAL combination such as:

Group 13

PATCH (NORTH, NWALL, 1, 1,NY, NY, 1, NZ, 1, LSTEP)
COVAL (NORTH, TEMP, 1.0, 5.0)
Note also that the enthalpy equation has had its NAME changed to TEMP. From that line on, any reference to enthalpy must be by the name TEMP.

## Note 2.

## VR Editor Mode:

A heat source can be attached to an existing feature (the wall) by the following sequence:
Click on the wall object to give an Object Dialogue Box.
Click on Attributes, then on 'Surface Enthalpy' next to Energy source. From the list of heat sources select 'Surface Heat Flux', and click on OK. Enter 5.0 in the 'Value' data entry box. Click on 'Total Heat Flux' to toggle to 'per unit area'.

## Command Mode:

The flux of enthalpy would be set by a PATCH/COVAL combination such as:

Group 13

PATCH (HOTNORTH, NORTH, 1, 1,NY, NY, 1, NZ, 1, LSTEP)
COVAL (HOTNORTH, TEMP, FIXFLU, 5.0)

## Note 3.

The total source of enthalpy (TEMP) for the wall is the product of the flux per unit area times the wall area. The first is 5.0, and the second is 2.0 x 1; the product is therefore 10.0.

## Note 4.

## VR Editor Mode:

A constant temperature can be attached to an existing feature (the wall) by the following sequence:
Click on the wall object to give an Object Dialogue Box. Click on Attributes, then on 'Surface Heat Flux' next to Energy source. From the list of heat sources select 'Linear Heat Source'. Enter 1.0E10 in the 'Coefficient' data entry box.

## Command Mode:

The values of enthalpy in the near wall cells would be fixed by a PATCH/COVAL combination such as:

Group 13

PATCH (FIXED, NORTH, 1,1, NY,NY, 1,NZ, 1, LSTEP)
COVAL (FIXED, TEMP, FIXVAL, 5.0)

## Note 5.

The source for a PATCH/COVAL pair is S = C (V - $\phi$ ). With a large coefficient, the effect of the source is to fix the value, V = $\phi$ (allowing for rounding errors). In such a case, S = 0. Earth is programmed not to print the source whenever 1E10 or more appears as the coefficient, to avoid rounding error. FIXVAL is a huge number - 1E20. If you use a smaller large number, say 1E6, for the coefficient, the value will still be fixed, but not so closely that the resulting source is lost in rounding error.

## Note 6.

The original case 240 treats the north boundary as a wall - the surface enthalpy is fixed to 5, but the enthalpy in the near-wall cell is calculated. The heat flux is calculated from the local enthalpy differences. In the second example, the heat flux in each cell is set to 5, and the local enthalpy calculated accordingly. In the last case, the near-wall cell enthalpies themselves are fixed to 5.

## Note 7.

The case concerns a closed cavity with one moving and three stationary walls. It contains a centrally-located heated steel block.

## Note 8.

Velocity, and hence momentum, is a vector quantity. The wall velocity is set to -0.1, so it is imparting negative momentum to the fluid (accelerating in the minus x direction). The NORTHW patch also has a negative momentum source, as it is slowing down (removing momentum from) fluid moving the positive x direction.

## Note 9.

## VR Editor mode:

This part of the exercise can only be done with the menu by changing the type of the inlet object from INLET to USER_DEFINED.
Click on the inlet object, then on Type. from the list of types, select USER_DEFINED and click OK. Click on Attributes to bring up the PATCH/COVAL dialog box. In the 'New' data entry box type IN. In the 'Type' entry box type LOW and click 'Apply'. Now set Coefficient and Value for the variables as below:

| Variable | P1 | V1 | W1 | TEMP |
|---|---|---|---|---|
| Coefficient | FIXFLU | 0.0 | FIXVAL | 0.0 |
| Value | 5.0 | 0.0 | 5.0 | 9.0 |

Click 'Apply' to check that the values have been entered correctly, then Click OK.

## Command Mode:

In group 13 use COVAL (IN, W1, FIXVAL, 5.0)

## Note 10.

By using FIXVAL, the velocity is 5.0 throughout the inlet section, whereas it was not constant with the other command. This is so because FIXVAL fixes the value of velocity at the faces of the cells involved, whereas the other commands (COVAL / ONLYMS or VALUE) only set the momentum convected in, but allow the cell velocities to change in response, for instance, to wall friction.

## Note 11.

## VR Editor mode:

Main menu/ Properties / RHO = 5.0

## Command Mode:

Insert RHO1 = 5 in Group 9

## Note 12.

The resulting inlet velocity is now smaller, and the pressure in the first row of cells is also low. The W1 velocity at IZ=1 is therefore seeing a pressure rise - (P(iz=2) - (P(iz=1)). This is because the inlet mass-flux is still set to 5.0 for an inlet velocity of 5.0. This causes an imbalance between momentum and velocity.
The FIXFLU COVAL for P1 (mass) specifies the total inlet mass-flux, which is
$m"_{in}$ = 5.0*inlet_velocity

The outflow from the first row of cells is
$m"_{out}$ = RHO1*W1(iz=1)
Since RHO1 has been increased, the W1(iz=1) velocity has to be smaller if continuity is to be satisfied. The pressure rise is there to decelerate the flow in line with continuity. The continuity equation for IZ=1 states:
$m"_{in}$ = $m"_{out}$
Hence W1(iz=1) = $m"_{in}$ / RHO1 = 5 / 5 = 1.
The (simplified) momentum equation is
m"in.Win - rho. W1. W1 = -$\Delta$ P
Hence $\Delta$ P = rho.W1.W1 - m"in. Win = 5 * 1 * 1 - 5 * 5 = -20

## Note 13.

## VR Editor Mode:

Click on inlet object twice, then on Attributes. Click on Inlet density is 'User-set' to toggle it to 'Domain fluid'. The mass flux at the inlet will now be calculated from the set velocity, and the density set for the domain fluid in the properties panel of the main menu.

## Command Mode:

In Group 13, multiply the value in the FIXFLU COVAL for P1 by RHO1. i.e.,
COVAL(IN, P1, FIXFLU, RHO1 * 5).
RHO1 is set to 1.0 by default, so its presence is implicit in the original set-up.

## Note 14.

## VR Editor Mode

Click on inlet object twice, then on Attributes. Change the Z component of the inlet velocity from 5.0 to 10.0.

## Command Mode:

Changes will be needed in the COVAL commands for P1 (total mass-flux) and W1 (inlet velocity). This is best done by parameterisation:

```
      Group 13

REAL(WIN)
WIN=10
PATCH (IN, LOW, ...
COVAL (IN, P1, FIXFLU, RHO1 * WIN)
COVAL(IN, W1, ONLMS, WIN)
```

## Note 15.

The pressure is held close to 0.0 at the outlet. Fluid can pass in either direction, depending on the local pressure difference.

## Note 16.

Inspection of the RESULT file shows that the pressures at IZ=5 are of the order 5. There are also positive V1 velocities - VR-Viewer shows that the vector field near the exit is peculiar. In a fully developed flow, there should be no pressure gradient normal to the wall, but the internal pressure has been allowed to drift too far away from the constant external pressure.

## Note 17.

The internal pressure at the exit can be made closer to the fixed external pressure by increasing the coefficients in the COVAL command (e.g., multiply it by 1000).

## VR Editor Mode;

Click on outlet object twice/Attributes/enter 1000.0 under coefficient for P1

## Command Mode:

Group 13

COVAL (OUTLET, P1, 1E3, 0.0)

## Note 18.

## VR Editor Mode;

Click on the Mesh Toggle button on the hand-set. Click on the image of the solution domain to bring up the grid mesh Settings dialog box. Change the Number of cells in the y-direction to 10 and the z-direction to 10.
Click on OK, then on the mesh Toggle button again to remove the display of the mesh.
Click on outlet object twice/change Ysize to 0.5/change Ypos to 0.5/OK

## Command Mode;

Make the following changes:

Group 4

GRDPWR(Y, 10, 1, 1)

Group 5

GRDPWR(Z, 10, 2, 1)

Group 13

PATCH (OUTLET, HIGH, 1, NX, NY/2+1, NY, NZ, NZ, 1, 1)

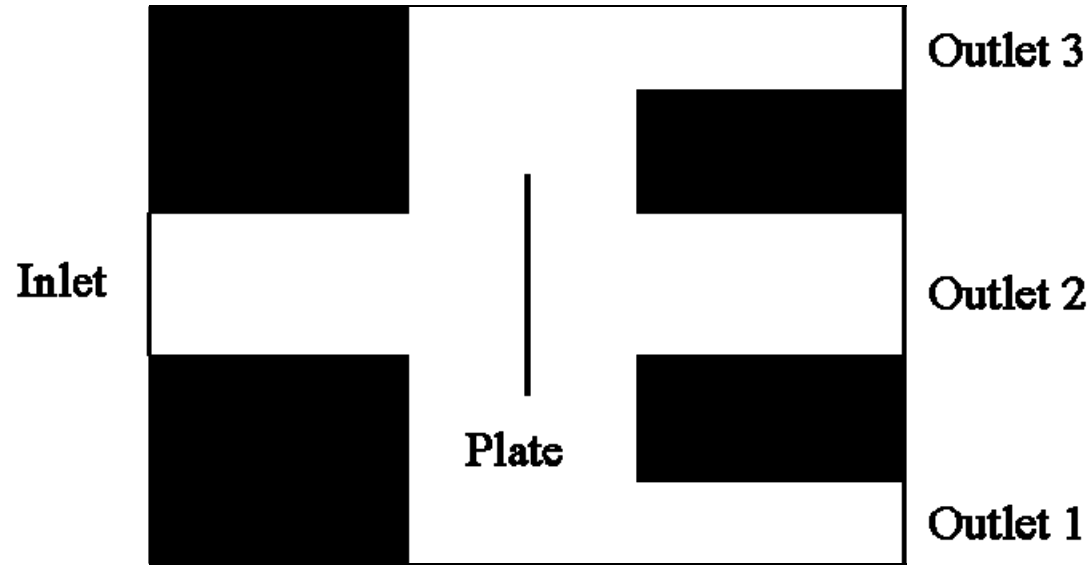
## Note 19.

The outlet has been particularly blocked or restricted, with the fluid leaving only through those cells in the PATCH. This is because, for the domain boundary cells not covered by FIXP PATCH the default boundary condition applies; no flux of matter, no flux of properties.

# WORKSHOP - Using PHOTON

## Flow Splitter with Multiple Outlets

The geometry for this case is shown below and is a 2D case to be set-up using PHOENICS-VR.




Flow enters from the left, and is diverted by the plate. The location and size of the plate determines the flow split between the three outlets.
The dimensions and flow rates are at the participants discretion.
Start with the plate located symmetrically, then move it up or down to see the effect on the flow split.
Try changing the outlet conditions to allow different exit pressures at each outlet. This can also greatly affect the flow patterns.

## Viewing Results using PHOTON

Run the case created by PHOENICS-VR.

To view the results run PHOTON from the PHOENICS-VR Environment by clicking on 'Run', 'Post processor' then 'Text mode (PHOTON)'.
Photon can be run in Menu mode or command mode. Which mode is used is entirely up to the participant.

## MENU Mode Operation

Type M <rtn> to enter menu mode, Click File, pHi-file, Accept to load the PHI file.

- To draw geometry: Click **File**, **photoN-use**, type in **PATGEO** <rtn>.
- To draw vectors: Click **Draw**, **vEctors**, **Do.**
- Filled contours: Click **Draw**, **cOntours**, Select the contour variable by clicking **P1**, then the required variable. Click **Do** to draw the contours.
- Isoline contours: As above, but click **Fill** before **Do.**
- To change view: Click **View**, then **Rotate** then click left mouse button on image and hold down to rotate view with the mouse.
- Menu tutorials: Click **setuP**, then **Tutorial**. Select a case - Radial Impeller is BFC, moving van is Cartesian. Then select the required tutorial. After a tutorial session, you will have to reload the original PHI file.

## COMMAND Mode Operation

Type **P** (to load PHI file)
Press **RETURN** (shown as <rtn> below) several times until Command? appears.

- To draw geometry: **USE PATGEO** <rtn>.
- To plot vectors: **VEC x,y or z plane number SH** <rtn> (e.g. VEC Z 1 SH will plot vectors on z 1 and will shade them)

- Filled contours: **CON variable name x,y or z plane number FI** <rtn>, **fill size** <rtn> (e.g. CON P1 Z 1 FI;0.01 will plot pressure contours on z 1, The fill size was 0.01)

- Isoline contours: **CON variable name x,y or z plane number SH** <rtn>, **range and interval** <rtn> (e.g. CON P1 Z 1 SH;INT 14 will draw isobars and shade them. Here the interval has been set to 14, the range will be the default whole range)

- To change view: **VIEW view vector or x,y or z** (e.g. VIEW Z will view from the z-direction VIEW 1 1 1 provides an isometric view)

N.B. A semicolon can be used to separate more than one command on a single line.

Home Page

# WORKSHOP - Combustion

## Overview

The case below exemplifies how the Simple Chemical Reaction Scheme can be applied to a combustion chamber. The flow is turbulent and the reaction-rate is physically controlled by means of the Eddy-Breakup Model. A complete step-by-step guide, showing how to set the case from the default mode of operation, is provided.

## Accessing PHOENICS-VR.

*From the system level:*

To enter the PHOENICS-VR environment, click on the PHOENICS icon on the desktop, or click on Start, programs, PHOENICS, PHOENICS.

*In PHOENICS-VR environment,*

Start with an 'empty' case - click on 'File' then on 'Start New Case', then on 'Core'', then click on 'OK'; to confirm the resetting.

*To enter VR Editor:*

This is the default mode of operation.

## Within VR Editor.

**Set the domain size and grid numbers**:

Click on 'Menu' (Main Menu) on the VE-Editor panel; and then on 'Geometry'.

Click on 'Cartesian' under co-ordinate system; and select 'Cylindrical-polar' and then on 'OK'.

Leave 'Inner radius' to 0.0 and 'tolerance' to 1.e-3.

Change the x-domain size to 0.628.

Change the y-domain size to 0.065.

Change the z-domain size to 0.245.

Set 'number of cells' to 8 in x-direction, 10 in y-direction and 25 in z-direction.

Click on 'OK' to close the Grid mesh settings dialog.

Click on 'OK' to exit from 'Menu'.

Click on 'Reset' button on the movement panel.

Click on 'Fit to Window'

Use Arrow buttons on the movement panel to position the Y-direction of the coordinate upright and z-direction pointing to the left.

## Activate solution of the required variables and models

Click on 'Menu'; and the on 'Models'.

Turn 'Solution for velocities and pressure' to ON.

Click on ENERGY EQUATION; and then choose ENTHALPY and toggle STATIC to TOTAL.

Click on LAMINAR (Turbulence models) and choose the KEMODL.

Click on COMBUSTION and choose 3_GASES_KINETIC.

## Specify properties

Clikc on 'Property'.

Turn 'Use property table' to OFF. The density will be automatically set to 3_GASES.

Turn (Density) 'Storage' to ON

Check that 'Temperature' should now be 3_GASES-KINETIC

and 'Specific Heat' is 3_GASES.

Click 'Page down' and continue the settings.

Change 'Reference Pres (Pa) to 8E5 and Reference Temp(K) to

0.0.

Click 'Page up'.

## Set Initial values

Click on 'Initialisation'.

Set the following initial values

- 10 to W1
- 11.25 to KE
- 353 to EP
- 11595 to H1
- 3.6 to DEN1
- 0.0548 to FUEL
- 0.0548 to MIXF

## Enter numerical settings

Click on 'Numerics'.

Set 'Total number of iterations' to 150.

Click 'Relaxation control' and set the following values:

- -0.6 to P1
- 0.01 to U1
- 0.01 to V1
- 0.01 to W1
- 10 to H1
- 10 to FUEL
- 10 to MIXF

Click 'Previous panel' and then click on 'Limits on variables'

Change VARMAX of P1 to 1E5 and VARMAX of DEN1 to 4.0

Change VARMIN of TMP1 to 273 and VARMIN of H1 to 1.0

Clikc 'Previous panel'

Click on 'Top menu' and then on 'OK' to exit from 'Menu'.

## Create Objects and specify boundary conditions

### Create Blockages for combustion chamber

Click on 'Obj' Button on the VR-Editor control panel.

Change name to CMP1.

Set Position and Size of the object as:

Xpos: 0.0 Xsize: 0.628

Ypos: 0.0 Ysize: 0.013

Zpos: 0.0 Zsize: 0.025

Define Type: Blockage.

Clikc 'OK' to exit from the Object Dialogue Box.

Click on 'Obj'.

Change name to CMP2.

Click 'Geometry'

Find POLWE2 in a subdirectory, shapes; and then select and click on 'Open'

Set Position and Size of the object as:

Xpos: 0 Xsize: 0.628

Ypos: 0.013 Ysize: 0.052

Zpos: 0 Zsize: 0.025

Define Type: Blockage

Clikc 'OK' to exit from the Object Dialogue Box.

### Create the Inlet

Click on 'Obj'.

Change name to INLET.

Set Position and Size of the object as:

Xpos: 0 Xsize: 0.628

Ypos: 0.013 Ysize: 0

Zpos: 0.009 Zsize: 0.004

Define Type: INLET.

Click on 'Attributes'

Toggle Inlet density to User-set and then set density to 3.606 and temperature 773K

Click on 'Gas composition' and set FUEL to 0.0548; Oxidant to 0.9452

Click on 'Apply' and 'OK'

Set velocity in Y-direction at 100 m/s.

Turn 'Object side' to High

Click on 'OK' twice to exit from the Object Dialogue Box.

### Create the secondary air inlet

Click on 'Obj'.

Change name to SIN1.

Set Position and Size as:

Xpos: 0 Xsize: 0.10

Ypos: 0.065 Ysize: 0

Zpos: 0.0735 Zsize: 0.0245

Define Type: INLET

Click on 'Attributes'.

Set Temperature at 773K.

Set velocity in Y-direction at -40. m/s

Click on 'OK' and check the position and size again.

Click on 'OK' to exit from the Object Dialogue Box.

Click on 'Obj'.

Change name to SIN2.

Set Position and Size as:

Xpos: 0.314 Xsize: 0.1

Ypos: 0.065 Ysize: 0

Zpos: 0.0735 Zsize: 0.0245

Define Type: INLET

Click on 'Attributes'.

Set Temperature at 773K and click 'OK'

Set velocity in Y-direction at -40. m/s

Click on 'OK' and check the position and size again

Click on 'OK' to exit from the Object Dialogue Box.

### Create the dilute air inlet

Click on 'Obj'.

Change name to DIN.

Set Position and Size as:

Xpos: 0.0 Xsize: 0.1

Ypos: 0.065 Ysize: 0

Zpos: 0.15 Zsize: 0.0245

Define Type: INLET

Click on 'Attributes'.

Set Temperature at 773K and click 'OK'

Set velocity in Y-direction at -40. m/s

Click on 'OK' and check the position and size again.

Click on 'OK' to exit from the Object Dialogue Box.

### Create the outlet

Click on 'Obj'.

Change name to Outlet.

Set Position and Size as:

Xpos: 0.0 Xsize: 0.628

Ypos: 0.0 Ysize: 0.065

Zpos: 0.245 Zsize: 0.0

Define Type: OUTLET

Click on 'Attributes' and then change coefficient to 1.0.

Click on 'OK' twice to exit from the Object Dialogue Box.

### Adjust grid distribution in Z-direction

Click on 'Mesh toggle' button, and the grid will appears on the screen.
Click on the first three regions, 1, 2 and 3 in Z-direction and set 3 grids in each region and click on 'OK'.

## Running the Solver.

In the PHOENICS-VR environment, click on 'Run', 'Solver' then 'Local solver(Earth)', and click on 'OK'; to confirm running Earth.

## Using the VR Viewer.

In the PHOENICS-VR environment, click on 'Run', 'Post processor',then GUI Post processor (VR Viewer) Viewer)'. Click 'OK' on the file names dialog to accept the default files.
To view:

- Vectors - click on the 'Vector toggle'
- Contours - click on the 'Contour toggle'
- Streamlines - click on the 'Create Streamline' button
- Iso-surfaces - click on the 'Iso-Surface toggle'

To select the plotting variable:

- To select Pressure - click on the 'Select Pressure button' .
- To select Velocity - click on the 'Select Velocity button'
- To select Temperature - click on the 'Select Temperature button'
- To select any other variable - click on the 'Select a Variable button' . Select H1 for enthalpy, or KE for Turbulent kinetic energy.

To change the direction of the plotting plane, set the slice direction to X, Y or Z

To change the position of the plotting plane, move the probe using the probe position buttons

.

## Saving the results.

In the PHOENICS-VR environment, click on 'Save as a case', make a new folder called 'COMBUST', select the new folder, and save as 'SCRS'.