Overview

This document gives step-by-step instructions for simulating turbulent flow from an array of jets that impinge on a flat surface. It is an extension of an earlier cookbook¹. In this version of the cookbook, the mesh in the impingement zone is created with a different procedure. The new procedure, described below, allows moving the outflow boundary farther away from the jet while minimizing the use of computational cells near the outflow.

Physical Problem

The physical problem is identical to that solved in the preceding cookbook. Figure 1 is a schematic of the jet impingement apparatus, and Figure 2 shows the relative positions of the four jets in the four-jet array. Figure 3 shows a section through two of the nozzles in the four-jet array. The geometrical parameters used in this cookbook are

\[ S = 12 \text{ mm}, \quad d = 3.18 \text{ mm}, \quad W = 40 \text{ mm}, \quad H = 12 \text{ mm} \]

Create the Mesh

As before, the mesh for this problem consists of three mesh zones joined together by arbitrary couples. Taking advantage of the symmetry lines indicated in Figure 2, the solution is obtained in the upper right quadrant of the physical domain.

The new twist in this cookbook is the definition of the mesh in the impingement zone. The impingement zone mesh will be created by assembling a two-dimensional mesh in the \((x, y)\) plane. This mesh is then extruded vertically \((z\) direction) to define the mesh in the impingement zone volume. The procedure for defining the mesh in the impingement zone is:

1. Define vertices of four cell blocks.

2. Manually create the blocks from the vertices. These blocks exist in the \((x, y)\) plane, and will be used to establish a two-dimensional mesh in that plane.

3. Define a mesh in each block.

4. Merge vertices common to the mesh blocks
5. Extrude the two-dimensional mesh into the three-dimensional volume.

**Prepare Cell Types**

Create four cell types with the *Cell Tool*. Three cell types are used for the fluid volume. One cell type is for the solid heater material that is used in the heat transfer analysis. Another cell type is a shell which is used to create the mesh in the impingement zone.

Open the *Cell Tool* from the menu bar of the Prostar Display window:

**Tools → Cell Tool...**

The following table summarizes the cell parameters. Note that the fluid cells have different colors, but all are in the same group (Group # 1). The heater belongs to group number 2. The shell cells in group 3 are used to construct the impingement zone mesh.

<table>
<thead>
<tr>
<th>Field Label</th>
<th>Impingement</th>
<th>Nozzle</th>
<th>Plenum</th>
<th>Heater</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Zone</td>
<td>Shells</td>
<td>Zone</td>
<td>Zone</td>
</tr>
<tr>
<td>Cell type</td>
<td>Fluid</td>
<td>Shell</td>
<td>Fluid</td>
<td>Fluid</td>
</tr>
<tr>
<td>Material number</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Color Table Index</td>
<td>2</td>
<td>15</td>
<td>7</td>
<td>5</td>
</tr>
<tr>
<td>Group Number</td>
<td>1</td>
<td>3</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Name</td>
<td>ImpingeCells</td>
<td>BaseShells</td>
<td>NozzleCells</td>
<td>PlenumCells</td>
</tr>
</tbody>
</table>
Figure 4. Top view of blocks defined in the \((x, y)\) plane. The location of the nozzle is shown for reference. It is not part of the block definition.

**Impingement Zone Mesh Generation**

Figure 4 shows the block definitions in the \((x, y)\) plane.

1. Generate vertices for the blocks in \((x, y)\) plane.
   a. Open the *Create Geometric Entities* panel in the Star GUIde NavCenter:
      
      *Create and Import Geometry → Create Geometric Entities*
   
   b. Select *Vertex* in the Select Items box at the top of the *Create Geometric Entities* panel.
   
   c. Select the *Vertices* tab in the bottom half of the *Create Geometric Entities* panel.
   
   d. Enter the coordinates of the nine vertices in Figure 4
      
      | vertex | \(x\) | \(y\) | \(z\) |
      |-------|------|------|------|
      | 1     | 0    | 0    | 0    |
      | 2     | 20   | 0    | 0    |
      | 3     | 50   | 0    | 0    |
      | 4     | 0    | 20   | 0    |
      | 5     | 20   | 20   | 0    |
      | 6     | 50   | 20   | 0    |
      | 7     | 0    | 50   | 0    |
      | 8     | 20   | 50   | 0    |
      | 9     | 50   | 50   | 0    |

   i. Click the *Vertices* button inside the yellow vertices tab
   
   ii. Enter 0, 0, 0 in the \(x\), \(y\), and \(z\) boxes, and Click done.
   
   iii. Repeat steps (i) and (ii) for vertices 2 through 9
   
   e. Verify that the vertices were entered correctly
      
      i. Open the vertex list tool from the Prostar Display Window
      
      *Lists → Vertices...*

      Verify that the nine vertices exist, and that they have the correct coordinates. Correct any errors with the buttons and input fields at the bottom of the *Vertex List* window.
ii. Click [Cell Plot] and [Vertex] in the top of the Prostar Display Window

iii. Turn on the vertex numbering in the display window

   **Plot → Number → Number**

   If the numbers don’t show up click [Replot]. Verify that the vertex plot resembles Figure 4.

2. Generate patches on the \((x, y)\) plane

   a. Open the **Cell Tool** from the menu bar of the Prostar Display window:

      **Tools → Cell Tool...**

   b. Select the shell type cell from the cell table at the top of the tool, and click the [Set Active Type] button.

   c. Form patches from the nine vertices in model. This must be done in the Prostar Command Window. Bring the window to the foreground.

      i. Create the first patch by entering the vertices in the right-handed sense

         \[
         \text{patc } 1 2 5 4 20 20
         \]

         The first four arguments of the \texttt{PATC} command are the four vertices progressing around the patch in a right-handed sense. The last two arguments are the number of cells in the \(x\) and \(y\) directions respectively. By changing these last two arguments, you can increase or decrease the mesh spacing as desired. The first four numeric arguments cannot be changed.

      ii. Create the remaining patches

         \[
         \begin{align*}
         \text{patc } 2 & \ 3 \ 6 \ 5 \ 10 \ 20 \\
         \text{patc } 4 & \ 5 \ 8 \ 7 \ 20 \ 10 \\
         \text{patc } 5 & \ 6 \ 9 \ 8 \ 10 \ 10
         \end{align*}
         \]
d. Display the cells in the \((x,y)\) plane.
   i. Create a cell set from all cells (including those just created)
      \[ C\rightarrow \text{All} \]
   ii. Click [Cell Plot]
   iii. Turn off numbering
      \[ \text{Plot} \rightarrow \text{Number} \rightarrow \text{Number} \]
      This toggles the number display off.
   iv. Replot.

   The result of the preceding steps should be a plot similar to that in Figure 5.

3. Extrude the two-dimensional shell into the \(z\) direction to create a volume mesh.
   a. Set the active cell type to cell type 4: \textit{impingementCells}
      i. Open the Cell tool
         \[ \text{Tools} \rightarrow \text{Cell} \]
      ii. Select entry number four in the list
      iii. Click \textit{Set Active}.
   b. Create a cell set from all shells in the model
      \[ C\rightarrow \text{New} \rightarrow \text{Shell} \]
   c. Extrude the shell by entering the \texttt{VEXTRUDE} command in the command window.
      \[
      \texttt{vcextrude 12 2000 cset all 1 normal 1 0}
      \]
      The first two arguments specifies the number of cell layers to create (12), and the increment (2000) in starting cells numbers for each layer. Since there are fewer than 2000 cells in each layer, incrementing the starting cell number by 2000 prevents the creation of duplicate cell number. The \texttt{cset all 1} arguments tell the \texttt{VEXTRUDE} command to use all cells in the current cell set (cf. step 3b, above). The \texttt{normal 1 0} arguments cause the cells to be generated normal to the plan of the base shell cells, in increments of 1 mm, and using the right-handed sense of the shell cells to determine which of the two normal directions for the extrusion.
   d. View the new three dimensional mesh
      i. Select all cells in the model
         \[ C\rightarrow \text{All} \]
      ii. [Cell Plot]
      iii. View as and edge plot: Click the edge button on the left of the Prostar Display window. The results should look like Figure 6.
   e. Merge the vertices: open the \textit{Assemble Grids} panel in the Star GUIde Navcenter
      \[ \textit{Create and Import Grids} \rightarrow \textit{Assemble Grids} \]
      i. Select the \textit{Vertex Merge} tab
      ii. Click \textit{All Vertices}
      iii. Keep \textit{Low}
      iv. Set the tolerance to 0.05
      v. Click Merge
      vi. Verify the merge by clicking [Replot]. All edges except the bounding edges should disappear.

This completes the construction of the impingement zone mesh. The final mesh is shown in Figure 7.
Figure 6: Mesh blocks resulting from the VCExtrude command. The internal edges indicate that the redundant vertices of cells have not yet been merged.

Figure 7: Final mesh for the impingement zone.
Completion of Model
The remaining steps in model creation identical to that used in the preceding cookbook. Continue by creating the mesh in the nozzle plate, and then the mesh in the plenum zone. Refer to the preceding cookbook for detailed steps.

Unfortunately the extended mesh zone at the outlet does not eliminate the recirculation at the outlet.

References