1 Overview

This document describes one way of building a high quality mesh in a round pipe. By high quality we mean a mesh that has cells that are close to being square and therefore do not have acute angles between adjacent sides. Figure 1 shows a radial mesh and an “O” mesh. The O-mesh resembles the letter “O” when viewed along the axis of the pipe. The O-mesh is better than a simple radial mesh because the O-mesh avoids narrow wedge-shaped cells near the centerline of the pipe.

In addition to the O-mesh in the pipe, this cookbook demonstrates how two meshes can be joined together even when their vertices and cell faces do not align at the interface. The joining of the mesh is achieved with arbitrary couples in Star-CD.

The following new PROSTAR features are introduced in this cookbook:

1. Extensive use of the PROSTAR command interface to define vertices, splines, and generate cells.
2. Use of splines to define vertices along a curved surface.
3. Creation of a surface mesh with the PATCH command.
4. Creation of a volume mesh by extruding a surface mesh.
5. Joining of adjacent meshes with arbitrary couples.

Figure 1: A radial mesh (left) and an O-mesh (right).
Physical Problem

The sketch in Figure 2 depicts a junction of a round pipe and a rectangular duct. The geometry and the flow are symmetric about the x-z plane through the centerline of the pipe. We will not exploit the symmetry in our model development. In other words, we will construct a mesh for the full duct depicted in Figure 2. The geometrical parameters used in this cookbook are

\[ d = 10 \text{ cm}, \quad H = 16 \text{ cm}, \quad B = 16 \text{ cm}, \quad W = 50 \text{ cm}, \quad L = 20 \text{ cm}, \quad s = 15 \text{ cm}. \]

The mass flow rate at the outlet is set so that the Reynolds number in the outlet duct is 1200. The hydraulic diameter of the outlet is

\[ D_h = 2HB/(H + B) = 16 \text{ cm}. \]

If \( V_{\text{out}} \) is the average velocity in the outlet duct then \( \dot{m}_{\text{out}} = \rho V_{\text{out}} BH \) and the Reynolds number in the outlet duct is

\[ \text{Re}_{\text{out}} = \frac{\rho V_{\text{out}} D_h}{\mu} = \frac{\rho \dot{m}_{\text{out}}}{\mu \rho HB H + B} = \frac{2\dot{m}_{\text{out}}}{\mu(H + B)} \]

Solving for \( \dot{m}_{\text{out}} \) and \( V_{\text{out}} \) gives

\[ \dot{m}_{\text{out}} = \frac{1}{2} \mu(H + B)\text{Re}_{\text{out}} \quad \text{and} \quad V_{\text{out}} = \frac{\nu(H + B)}{2HB}\text{Re}_{\text{out}} \]

For \( \text{Re}_{\text{out}} = 1200 \) and the dimensions of the duct given above,

\[ \dot{m}_{\text{out}} = \frac{1}{2}(1.81 \times 10^{-5}\text{ kg/m/s})(0.32\text{ m})(1200) = 3.4752 \times 10^{-3}\text{ kg/s} \]

\[ V_{\text{out}} = \frac{0.34812\text{ kg/s}}{(1.205\text{ kg/m}^3)(0.02560\text{ m}^2)} = 0.11266\text{ m/s} \]

For incompressible flow, mass conservation requires

\[ V_1 \frac{\pi}{4} d^2 + V_2 BH = V_{\text{out}} BH \]
2 Preparing and Launching PROSTAR

By now, these standard preparatory steps should be routine

1. Create a directory to hold the files for this session.
2. Tell Star-CD where to find the working directory.
3. Run the Star-CD Launcher

Before starting PROSTAR, we must increase the memory available for creating couples. Do this by running PROSIZE from the Star-CD Launcher.

**Pre/Post → Prosize**

1. Click Continue after reading the message in the PROSIZE Info window.
2. Click Modify in response the query about what to do with your existing param.prp file
3. Click OK when PROSIZE says that the param.prp has been copied to param.prp.bak.
4. Click on the Matches tab of the PROSIZE Data Entry window.
5. Increase the Maximum number of couples MAXNCP to 5000. Increase the Maximum number of cells in each couple NCPDIM to 25. Later, if you encounter an error due to insufficient memory for couples in PROSTAR, launch PROSIZE again and increase MAXNCP and/or NCPDIM.
6. Click Save
7. Click OK when PROSIZE says that the param.prp file has been saved in your working directory.
8. Click Exit.

Now, launch PROSTAR and prepare to build the model.

1. Start PROSTAR
2. In the opening screen, enter a meaningful case name, e.g., pipeJunction
3. Give the model a title: File → Model Title....

3 Create the Mesh

The mesh for this problem is created from two adjacent mesh regions: one in the round pipe, and one in the rectangular duct. The two mesh regions do not have common vertices, so it is not possible to simply merge vertices to create a unified mesh. Instead, a Star-CD feature called arbitrary couples is used to join the two mesh regions at their shared interface.

Before the meshes can be joined with couples, the separate meshes must be created. The mesh in the rectangular duct is easily created as a structured array of three-dimensional brick-shaped cells. The mesh in the pipe could be created as a structured mesh in cylindrical coordinates. This would
Figure 3: Sectors of the O-mesh. On the left, the pipe cross section is divided into five sectors. On the right, a set of vertices is used to specify the outline of the first sector.

have the undesirable effect of creating cells near the centerline with acute angles. The narrowness of the cells would increase as the mesh is refined.

There is more than one way to create a high quality mesh in the cylindrical pipe. In this cookbook we create an O-mesh by subdividing the pipe cross section into five sectors as depicted in the left half of Figure 3. Each sector is a topologically similar to a rectangle: it has four sides and four "corners". For the four outer sectors (labelled ① through ④ in Figure 3) the sides are coincident with the duct wall are curved.

The right half of Figure 3 shows how the sides of the first sector are defined by a series of points. These points are vertices in Star-CD, but the vertices on the pipe wall do not necessarily coincide with the corners of fluid cells in the final mesh. The vertices on the pipe wall are used to define a spline, which is used as a surrogate for the true shape of the pipe wall. A spline is a smoothly varying interpolating function constructed from low degree polynomials (typically cubic polynomials). To use a spline in STAR-CD, you first define a series of vertices, and then group these vertices to define the spline.

In the right half of Figure 3, the points labelled 2, 3, ..., 6 are used to define the spline. A STAR-CD spline can have up to 17 points. Later in this cookbook an automated procedure is presented for specifying a variable number of points on the spline along the the outer sectors of the pipe wall.

**Strategy for Mesh Creation**

The mesh for the geometry in Figure 2 is created with these steps

1. Create a surface O-mesh like that in the right half of Figure 1.

2. Extrude the surface mesh to fill the pipe volume.

3. Create a structured mesh in the rectangular duct volume.

4. Establish couples between the fluid cells in the pipe volume that share a face with fluid cells in the rectangular duct volume.

The bulk of the meshing effort is in creating the volume mesh for the pipe and in creating the couples.

The mesh in the pipe volume is created by assembling surface meshes in the five sectors depicted in the left side of Figure 3. The mesh for each sector is defined with the following steps.
1. Define the vertices around the perimeter of the sector.

2. Create a spline by joining the vertices along the outer wall.

3. Create a surface mesh in the sector by subdividing the sector into a uniform number of radial shells and angular sub-sectors. This step is achieved with the PATCH command.

After the surface mesh in the five sectors created, these surface meshes are joined by merging coincident vertices. The surface mesh in the individual sectors are carefully defined so that all vertices on the shared interfaces are coincident.

**Prepare Cell Types**

Three cell types are defined to aid in manipulating the model. Two cell types are used for the fluid volume: one for the pipe volume and another for the rectangular duct volume. The cells in these volumes have identical fluid properties. The cells must be of different type so that they can be joined during via couples. Another cell type is a shell used to create the surface mesh on the end of the pipe.

1. Open the **Cell Tool** from the menu bar of the PROSTAR Display window:

   **Tools → Cell Tool...**

2. Click on an empty entry in the **Cell Table** (item 4 should be blank)

3. Click **Edit Types...** This opens up the **Cell Table Editor**. Enter the following data

<table>
<thead>
<tr>
<th>Cell Type</th>
<th>Shell</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material Number</td>
<td>1</td>
</tr>
<tr>
<td>Color</td>
<td>6</td>
</tr>
<tr>
<td>Group Number</td>
<td>3</td>
</tr>
<tr>
<td>Name</td>
<td><strong>pipeSurf</strong></td>
</tr>
</tbody>
</table>

4. Click **Apply**

5. Repeat steps 2 through 4 to create two additional cell types using the data from the last two columns of the following table. Remember to click **Apply** after the data for each cell type is entered.

<table>
<thead>
<tr>
<th>Field Label</th>
<th>Shells</th>
<th>Pipe</th>
<th>Rectangular</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cell type</td>
<td>Shell</td>
<td>Fluid</td>
<td>Fluid</td>
</tr>
<tr>
<td>Cell index</td>
<td>4</td>
<td>5</td>
<td>6</td>
</tr>
<tr>
<td>Material number</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Color Table Index</td>
<td>6</td>
<td>2</td>
<td>5</td>
</tr>
<tr>
<td>Group Number</td>
<td>3</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Name</td>
<td><strong>pipeSurf</strong></td>
<td><strong>pipeCells</strong></td>
<td><strong>ductCells</strong></td>
</tr>
</tbody>
</table>

Note that the fluid cells have different colors, but all are in the same group (Group # 1). The shell cells in group 3 are used to construct the surface mesh on the end of the round pipe.

6. Click **Close** to close the **Cell Table Editor**
Create the Surface Mesh

The surface mesh is created by entering commands in the PROSTAR Command Window.

The steps for creating the surface cells in the first sector (sector $\Phi$ in Figure 3) are as follows.

1. Create the vertices in the first sector with these commands

   $\begin{align*}
   V, 1, & \ 1.41421356, \ 1.41421356, \ 0.00000000 \\
   V, 2, & \ 3.53553391, \ 3.53553391, \ 0.00000000 \\
   V, 3, & \ 1.91341716, \ 4.61939766, \ 0.00000000 \\
   V, 4, & \ 0.00000000, \ 5.00000000, \ 0.00000000 \\
   V, 5, & \ -1.91341716, \ 4.61939766, \ 0.00000000 \\
   V, 6, & \ -3.53553391, \ 3.53553391, \ 0.00000000 \\
   V, 7, & \ -1.41421356, \ 1.41421356, \ 0.00000000 \\
   \end{align*}$

2. Use the vertices along the pipe wall to define a spline

   SPL 1, VLIST, -2, 3, 4, 5, -6

3. Use the corner vertices of the sector to create surface cells

   PATCH, 1, 2, 6, 7, 6, 6, , , 5000

4. In preparation for viewing the mesh, select all the newly created cells

   \[
   \text{C-}\rightarrow\text{All}
   \]

5. Click *Cell Plot*.

To avoid typographical errors, and to avoid the tedium of entering the commands one at a time, read the commands from the *omesh_coarse.MAC* file.

*FILE omesh_coarse.MAC*

Plot the newly created surface mesh

1. Select all the cells currently defined

   \[
   \text{C-}\rightarrow\text{All}
   \]

2. Click the *Cell Plot* button at the top of the PROSTAR Display Window. The result should look like Figure 4

Create the Fluid Mesh in the Pipe

The fluid cells in the pipe are created by extruding the surface mesh into the volume. The extrusion uses the surface mesh as a pattern.

1. Set the active cell type to the pre-defined fluid cell type

   \[
   \text{ctyp 5}
   \]

2. Extrude the surface mesh

   \[
   \text{vextrude, NZ, , all, , 1, normal, DZ, 0}
   \]

3. In preparation for viewing the mesh, select all the newly created cells

   \[
   \text{C-}\rightarrow\text{All}
   \]

4. Click *Cell Plot*.

The result of the preceding steps should be the cell plot in Figure 5.
Figure 4: Coarse mesh in the pipe cross section.

Figure 5: Fluid mesh for the pipe.
3 CREATE THE MESH

Create the Mesh in the Rectangular Duct

1. Generate the Mesh

   a. Click on the title bar of the Star GUIde NavCenter window to bring it to the front
   b. Click Create and Import Grids
   c. Click Create Grids
   d. Click Create 3-D Grids using Simple Shapes
   e. Click Select a Cell Type
      i. Select entry 6 Fluid
      ii. Click Apply to make this the active cell type
      iii. Click Close
   f. In the Extent of Domain region at the right side of the Star GUIde NavCenter window, enter the following values in the boxes

<table>
<thead>
<tr>
<th></th>
<th>Minimum</th>
<th>Maximum</th>
<th>Number of Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>−15</td>
<td>35</td>
<td>50</td>
</tr>
<tr>
<td>Y</td>
<td>−8</td>
<td>8</td>
<td>20</td>
</tr>
<tr>
<td>Z</td>
<td>−16</td>
<td>0</td>
<td>20</td>
</tr>
</tbody>
</table>

   Note that the dimensions are in centimeters. The scale factor for the geometry will be set when the .geom file is written.
   g. Click Generate Mesh at the bottom of the right side of the Star GUIde NavCenter window.

2. Inspect the mesh

   a. Change the model view in the PROSTAR Display Window
      
      View $\rightarrow$ Isometric $\rightarrow$ 1,1,1
   b. Click Show edges icon along the left the PROSTAR Display window.
   c. Select the Quick Hidden Line plot from the drop down menu at the top of the PROSTAR Display Window. The result should look like the image in Figure 6. Notice that the two blocks of cells are separated by an edge.
   d. Select the Hidden Surface plot from the drop down menu at the top of the PROSTAR Display Window.
   e. Zoom in near the junction between the pipe and the rectangular duct. The result should look like the image in Figure 6. The two distinct meshes will be combined with arbitrary couples.
Figure 6: Edges of the mesh for the pipe and the rectangular duct.

Figure 7: Cells near the junction of the pipe and the duct.
4 Create Couples to Join the Mesh Regions

Refer to Chapter Four in the STAR-CD User Guide for information on creating mesh couples. When couples are used to join meshes, any coincident vertices are not merged.

There are two types of couples in STAR-CD: integral couples and arbitrary couples. We use arbitrary couples in this cookbook. An arbitrary couple creates a connection between a master cell and one or more slave cells.

Create a New Couple Type

1. Open the Couple Tool in the PROSTAR Display Window

2. Click Edit Types... (the yellow button immediately under the Couple Table list). This opens the Couple Table Editor window.

3. In the Couple Table Editor
   a. Select the first empty line in the Couple List on the left half of the window.
   b. Set the following values in the boxes on the right half of the window:

<table>
<thead>
<tr>
<th>Field Label</th>
<th>You Enter/Select</th>
</tr>
</thead>
<tbody>
<tr>
<td>Couple Type</td>
<td>Arbitrary</td>
</tr>
<tr>
<td>Group Number</td>
<td>1</td>
</tr>
<tr>
<td>Master color</td>
<td>8</td>
</tr>
<tr>
<td>Slave color</td>
<td>14</td>
</tr>
<tr>
<td>Overlap Color</td>
<td>5</td>
</tr>
<tr>
<td>Partial Boundaries</td>
<td>On</td>
</tr>
<tr>
<td>Tolerance</td>
<td>0.2</td>
</tr>
<tr>
<td>Name</td>
<td>pipeToDuct</td>
</tr>
</tbody>
</table>

   c. Click Apply

4. Close the Couple Table Editor window. The new couple type should appear in the Couple Table list.

Couple the Pipe to the Rectangular Duct

Create the couple between the pipe and the rectangular duct. The cell size in both zones is comparable, so on that basis one could choose either the pipe cells or the rectangular duct cells to be the master cells. However, at the outer diameter of the pipe, the pipe cells will only partially cover cells on the wall of the rectangular duct. Since the rectangular cells will only be partially covered, choose the rectangular cells as the master cells and the pipe cells as the slave cells.

1. Prepare for the cell selection by zooming in on the nozzle cells to get a view similar to that in Figure 7.

2. In the Couple Tool window, select the first couple type from the list, and click Set Active Type (second yellow button).

3. Click Create Couples...
4 CREATE COUPLES TO JOIN THE MESH REGIONS

<table>
<thead>
<tr>
<th>Field Label</th>
<th>You Enter/Select</th>
</tr>
</thead>
<tbody>
<tr>
<td>Choose Cells</td>
<td>Cell Set</td>
</tr>
<tr>
<td>Master/slave Option</td>
<td>Choose Cell Types</td>
</tr>
<tr>
<td>Couple Type</td>
<td>3</td>
</tr>
<tr>
<td>Master Cell Type</td>
<td>5</td>
</tr>
<tr>
<td>Slave Cell Type</td>
<td>6</td>
</tr>
</tbody>
</table>

Leave the default *Vertex Face Tolerance, Centroid Plane Tolerance, and Normal Angle Tolerance.*

4. Click *Apply*. This creates the couples.

5. Click *Close* to close the *Create Couples* panel.

6. Graphically verify that the couple was created.
   a. In the *Couple Tool Window*:
      
      ![Couple Display](image)
      
      → Masters Last

      This will create a display with the master cells drawn on top of the slave cells.

   b. Select the *Hidden Surface* display from the top of the PROSTAR Display window.

   c. Choose only those cells involved in the couple

   ![C-►](image)

   → New → Couple → Both

   d. Click *Cell Plot*. The result should be a display similar to that in Figure 8

   e. Open the *Couple List* from the menu bar in the PROSTAR Display Window

      Lists → Couples...

   f. Click either the *Show Couples with Cells in Cset only* button or the *Show All Couples* button. The scrolling list in the left half of the window shows the master cells for the couple. Selecting any master cell causes its slave cells to be displayed in the list in the right half of the window.

   g. Close the *Couple List* window

7. Return the display to normal
   a. Turn off the display of the couple. In the *Couple Tool window*:

      ![Couple Display](image)

      → Off

   b. Select and plot all the cells in the model.

      ![C-►](image)

      → All

   c. Click *Cell Plot* in the PROSTAR Display window.
Figure 8: Master and slave cells for the arbitrary couples between the pipe volume and the rectangular duct volume.
5 MATLAB Code to Generate Surface O-Mesh

Let $n_q$ be the number of vertices around the perimeter of any outer sector in left half of Figure 3. For the sample quadrant in the right half of Figure 3, $n_q = 5 (= 6 - 2 + 1)$. Let $i_s$ be the starting index for the vertices on the perimeter of any sector. Each of the outer sectors has indices

$$i_s, i_s + 1, \ldots, i_s + n_q - 1$$

The indices

$$i_s + 1, \ldots, i_s + n_q - 2$$

are on the outer radius of the pipe.

In STAR-CD version 3.15, each spline can have no more than 17 points, i.e. $\max(n_q) = 17$. To avoid reusing a vertex number (which would cause a mesh error), and to make it easy to debug the sector vertex indices, the starting points of the segments differ by 50 as summarized in the following table.

<table>
<thead>
<tr>
<th>Sector</th>
<th>$i_s$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>51</td>
</tr>
<tr>
<td>3</td>
<td>101</td>
</tr>
<tr>
<td>4</td>
<td>151</td>
</tr>
</tbody>
</table>

Sector five is created from four unique inner vertices from the outer sectors.

Refer to the source code of `makeCircleOmesh.m` for details.