Gridgen

Tutorials

Version 15

POINTWISE®
Get the Point.

RIGHT TO USE LICENSE

1. DEFINITIONS.

(a) "Licensed Software" means any computer program(s) supplied by Pointwise, Inc. (PWI) to Licensee under a valid Purchase Order or Contract, whether in object code, reconfigurable binary, or any other form; video media and training aids; and any backup or other copies, updates, derivative works, modifications, enhancements, and extensions thereof.

(b) "Documentation" means user manuals, documentation binders, release notes, installation notes, written utility programs and other written or graphic materials related to the Licensed Software and all copies thereof.

(c) "Licensed Product(s)" means the Licensed Software and Documentation.

(d) "Maintenance Period" means the first calendar year of a perpetual license or 12 months for an annual license.

2. LICENSE.

PWI grants to Licensee a non-exclusive, non-transferable license to use the Licensed Products in accordance with the terms and conditions set forth herein. As long as this License is in effect, Licensee may transfer its use of the Licensed Products to a replacement computer system on a temporary or permanent basis, provided that Licensee gives PWI written notice of such transfer.

3. TERM and FEES.

The Term of the Right to Use License is annual or perpetual, as set forth in the Purchase Order or Contract, provided that the License has not been terminated as set forth in Section 10.

All applicable fees are payable to PWI within thirty (30) days from Invoice Date. Failure to pay such fees in a timely manner is a material breach of this License. Licensee agrees to pay any legal or collection fees incurred by PWI in collecting any amounts that may be delinquent.

4. COPIES and MODIFICATIONS.

Licensee may copy the Licensed Products in whole or in part, in written or machine readable form for use in understanding the Licensed Software and for archive or backup purposes. Licensee shall reproduce and include PWI's trade secret or copyright notices on and in any copies, in any form. Licensee shall not reverse assemble, reverse compile or otherwise reverse engineer the Licensed Software in whole or in part.

The license includes the right to reproduce the Gridgen Version 14 User Manual exclusively for the use of Licensee and its employees only. All other reproduction and distribution is strictly prohibited.

5. OWNERSHIP.

All Licensed Products and the information they contain, any updates and all copies of them are PWI's or PWI's licensors' property, and title to them remains in PWI or such licensors. All applicable rights in patents, copyrights or trade secrets in the Licensed Products are and will remain in PWI and PWI's licensors. No title to or ownership of the Licensed Software or the information they contain is transferred to Licensee. Licensee agrees that any terms and conditions imposed by PWI's licensor and communicated by PWI to Licensee by PWI, shall apply to the Licensed Products.

6. CONFIDENTIALITY.

(a) Licensee will take all reasonable precautions to maintain the confidentiality of the Licensed Products, and agrees to take all reasonable and necessary steps to protect the patents, trademarks, copyrights, trade secrets and any other forms of intellectual or industrial property of PWI in the Licensed Products.

(b) Licensee will not provide the Licensed Products to any person, other than employees of Licensee, without PWI's prior written consent, except during the period any such person is on Licensee's premises with Licensee's permission for purposes specifically related to Licensee's use of the Licensed Products.

7. COPYRIGHT AND PATENT INDEMNITY.

PWI assures Licensee that, to the best of PWI's knowledge, the Licensed Products do not infringe any patent, copyright, or trade secret. In the event any legal proceedings are brought against Licensee claiming an infringement of a patent, copyright, or trade secret based on Licensee's use of the Licensed Products, PWI agrees to defend at PWI's own expense any such legal proceeding relating to such claim or claims and to hold Licensee harmless from any damage incurred or awarded as the result of settlement or judgment against Licensee, provided Licensee gives PWI prompt written notice within fifteen (15) days of any such claim or the institution of any such claims against Licensee, and further, Licensee cooperates completely with PWI in providing all necessary authority, information, and reasonable assistance to enable PWI, at PWI's option, to settle or defend such claims.

8. LIMITATION OF WARRANTY AND LIABILITY.

PWI warrants that the Licensed Products will perform substantially in accordance with all written specifications furnished to Licensee by PWI if properly used. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, PWI DISCLAIMS ALL OTHER WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, WITH REGARD TO THE LICENSED PRODUCTS. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, IN NO EVENT SHALL PWI BE LIABLE TO LICENSEE OR ANY PARTY FOR ANY SPECIAL, INCIDENTAL, INDIRECT, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING, WITHOUT LIMITATION, DAMAGES FOR LOSS OF BUSINESS INFORMATION, OR ANY OTHER PECUNIARY LOSS) ARISING OUT OF THE USE OF OR INABILITY TO USE THE LICENSED PRODUCTS EVEN IF PWI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES AND REGARDLESS OF THE FAULT OR NEGLIGENCE OF PWI. PWI's liability to Licensee for damages shall not exceed the amount of the license fee paid by Licensee to PWI.

9. MAINTENANCE AND SUPPORT.

(a) PWI will be responsible for providing corrections for programming errors and periodic software updates only during the Maintenance Period and during any period when the Licensee is covered by a Maintenance Agreement from PWI or an authorized representative of PWI. The specific policy for software updates and enhancement varies on a per product basis. PWI warrants that during the term of this License it will use reasonable efforts to remedy defects in the unaltered Licensed Software made known to it by Licensee. PWI does not warrant that the Licensed Products will meet all requirements of Licensee, or that the operation of the Licensed Software will be uninterrupted or error free, or that all software defects will be corrected.

(b) During the Maintenance Period, PWI agrees to telephone hot-line support available to Licensee. These services may be provided by PWI or a designated third party.

10. TERMINATION.

Licensee may terminate this License upon thirty (30) days written notice to PWI. PWI may terminate this License for any material breach by Licensee of the terms and conditions contained herein upon thirty (30) days written notice to Licensee. Licensee will have thirty (30) days from the date of such notice to cure the breach, and if the breach is cured to the satisfaction of PWI, this License will remain in full force and effect.

Upon termination, Licensee shall promptly:
(i) discontinue use of the Licensed Products;

(ii) remove the Licensed Products from any software in Licensee's possession or control that incorporates or uses the Licensed Products in whole or in part; (iii) erase or destroy any of the Licensed Products contained in the computer memory or data storage apparatus under the control of Licensee. Licensee's obligations under Section 6 shall survive any termination of this License.

11. GENERAL TERMS.

The entire agreement between the parties is contained herein and in a valid Purchase Order or Contract, which supersede all proposals, oral or written, and all other communications between the parties relating to this Agreement and it may be executed in any number of counterparts, each of which shall constitute an original, and all of which taken together shall constitute one and the same Agreement. This Agreement is not assignable by Licensee without prior written permission from PWI. The section headings and subheadings herein are for convenience only and shall not affect the interpretation or construction of this Agreement. PWI shall not be liable for any failure or delay in performance due in whole or in part to any cause beyond PWI's control.

This Agreement and all transactions under it shall be governed by the laws of the State of Texas. All claims arising under or related to this Right to Use License, Purchase Order or Contract shall be settled finally and exclusively by arbitration in accordance with the Commercial Arbitration Rules of the American Arbitration Association (AAA). Arbitration shall take place in Dallas, Texas and be administered by the AAA's Dallas, Texas office.

**Volume 3
Tutorial Workbook
Version 15**

TABLE OF CONTENTS

1. Cubes and Spheres: Layer Manager

1.1 Introduction

This section includes a brief tutorial to help you become familiar with the basic operations of Gridgen's Layer Manager. It will also introduce you to the database pick masking capability available in all database entity browsing operations. There will be no actual entity creation in this tutorial. Only database model manipulation will be covered as an introduction to how the Layer Manager gives you ultimate control over your database model and its display.

1.2 Topics Covered

This sample session is a short introduction to using Gridgen. Basic Gridgen skills you will learn are:

• assigning entities to certain layers using the **Assign Layer** command

• manipulating layers using the layer Browser list

• manipulating the layer Browser list using filters

• changing the display state of certain layers using **Isolate**, **All On** and **All Off**

• creating saved layer sets

• using the database Browser command, **DB Pick Mask**

1.3 Background

Work through this tutorial to familiarize yourself with the basic operations of Gridgen's Layer Manager.

1.4 Geometry

For this introduction to Gridgen's Layer Manager we will work with a geometry model consisting of three nested spheres next to three nested cubes. The model is shown in Figure 1.1 below.
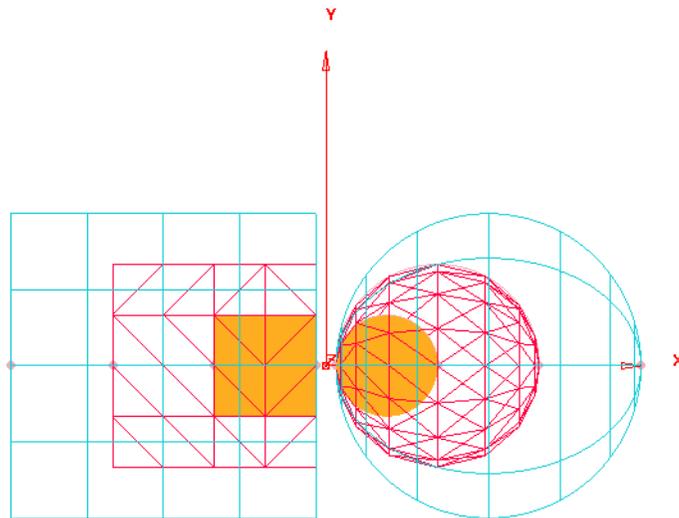
Figure 1.1 Database Geometry Model

This is purely a contrived problem for demonstration purposes; however, the operations to be demonstrated will be common to any models you work with in Gridgen.

1.5   Start Gridgen

Start Gridgen by typing the string `gridgen` on Unix workstations or by simply double-clicking the Gridgen shortcut on your PC's desktop. After a few seconds Gridgen will start, and you will see the Gridgen logo in the Display window. The logo will disappear after a few seconds or as soon as you select your first menu button.

1.6   Database Model Import

To begin the tutorial, import the composite geometry file, `lyrmgr.dba`. This file should be available in your Gridgen installation directory under *gridgen_home_path*`/examples/tutorial/lyrmgr`. You may wish to copy the file to some other working directory before proceeding. From the MAIN MENU:

1.   **Input/Output**

2.   **Database Import**

3.   Use the file Browser to navigate to the directory containing the copy of `lyrmgr.dba` and select it.

4.   **Done**

In your Display window you should now see nothing. That is right, the model has been imported, but nothing appears. You can verify model import by looking at the number of DB entities in the Status window, top left of the Gridgen screen as shown in the next figure.

1.7   Display Full Database Model

No entities are displayed because all of the layers in the database model are currently turned off. This is your first task; turn on all layers to view the model, starting from the MAIN MENU:

1. **Layer Manager**

2. Status: **All On**

Your Layer Manager panel should appear as in the following figure.
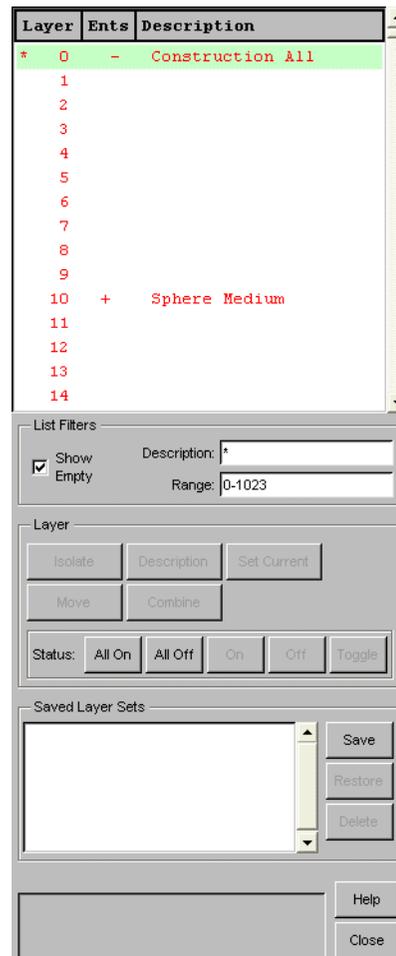
3. Click in the Gridgen GUI window and use the *r* hot-key on the keyboard to have Gridgen recalculate the viewing extent box for the model.

Your model in the Display window should appear as in the next figure.
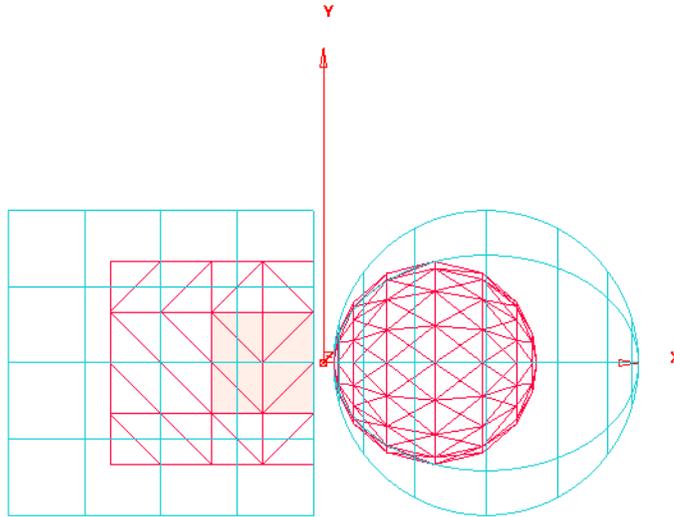
In this version of the model, the small cube has been set to the **Shaded** display attribute, so in this perpendicular view, it appears very light and almost white. As you rotate your model, you will see it more clearly. Also of importance, note that the small sphere is not visible. It would appear to be missing. Actually it is only disabled, but is also somewhat hidden in the layer list. Your next task is to find the layer it is in, isolate that layer, and enable the sphere.

1.8   Find and Enable Small Sphere

Your Layer Manager panel has been pushed to the background on your desktop due to your use of Gridgen's primary GUI. To restore it immediately, use the keyboard hot key, ***Shift+m***.

First some discussion of the layer Browser and its contents is warranted. Refer to the next figure below for the following description. The first column, labeled Layer, shows the layer number. There are a total of 1024 layers, starting with 0 and ending at 1023. An asterisk (*) is shown in this column for layer 0; this means layer 0 is the current working layer. This is also why layer 0 is highlighted in green. The second column labeled Ents indicates the existence of entities with either a positive or negative sign (+ or -). If neither are present, there are no entities in that layer. A positive sign (+) indicates the existence of enabled entities; a negative sign (-) indicates the existence of disabled entities. Finally, the third column labeled Description contains an alpha-numeric description of that particular layer, presumably of its contents.

Note that in our case we only see two layers that contain any entities. Layer 0 contains construction entities, all of which are disabled as indicated by the negative sign (-). Layer 10 contains the medium sphere and all entities are enabled as indicated by the positive sign (+). To compact the layer Browser list, turn off the Show Empty list filter in the List Filters section of the Layer Manager panel:

1. Show Empty

This removes all empty layers from the layer Browser list, giving you a nice condensed list of all layers that contain enabled and/or disabled entities. The reduced layer Browser list is shown in the next figure.

Layer	Ents	Description
* 0	–	Construction All
10	+	Sphere Medium
15	+	Sphere Large
20	+	Cube Small
25	+	Cube Medium
30	+	Cube Large
831	–	

You now have a hint as to where that missing sphere is located. Layer 831, although not labeled, apparently contains some disabled entities. Isolate that layer now and then enable all entities to determine what it contains.

2. Place the mouse cursor over layer 831 in the list and left-click to select it. It will now be highlighted in blue.

3. **Isolate**

This turns off all other layers, indicated by their text in the layer Browser turning from red (on) to black (off). This command also sets layer 831 to the "current working layer", meaning if any new database entities were now created, they would automatically be placed in layer 831. Also note that no entities are shown in the Display window since all other layers are off and apparently everything in layer 831 is disabled. Now, from the MAIN MENU in Gridgen, proceed to the DATABASE COMMANDS menu:

1. **Database**

2. **En/Disable**

You can now see the disabled small sphere in the DB ENABLE STATUS Browser list and rendered in purple in the Display window.

3. Pick **All**

4. **Done**

Now you see the nicely shaded display of the small sphere in the Display window. Your Layer Manager panel has again been pushed to the background on your desktop due to your use of Gridgen's primary GUI. Use *Shift+m* to pop it up. Take the following steps to turn on the entire model again and restore the view to original orientation:

1. Status **All On**

2. Click in the GUI and use the *r* hot-key to restore orientation.

Your model should now appear as in the figure below.
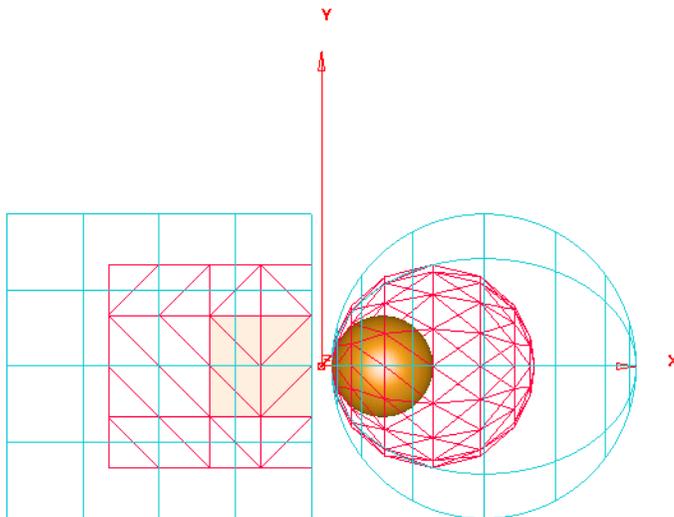
1.9   *Move and Description*

Use *Shift+m* to pop up the Layer Manager panel. Even though we have restored the display of the small sphere, you can see that it is not layered consistently with the other objects in the model. Since the medium sphere is in layer 10 and the large sphere is in layer 15, the small sphere likely really belongs in layer 5. Move the entity now and label the layer accordingly.

1. Select layer 831.

2. **Move**

The Move dialog box pops up:
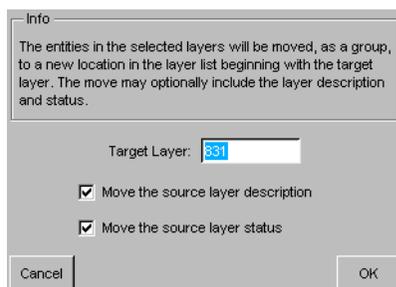
3. Enter 5 in the Target Layer field.

4. **OK**

You were not instructed to alter the toggles for either Move the source layer description or Move the source layer status. The source layer has no description, so that option is irrelevant. The source layer status (on) should be preserved.

Now finish the job with a new description for layer 5.

5. Select layer 5.

6. **Description**

The Description dialog box pops up:

7. Enter `Sphere Small` in the Description field.

8. **OK**

Now all the major objects in the model should be layered and described consistently. Your layer Browser should now show the updated layer list as shown below. Note that layer 5 is now included in the list and layer 831, even though empty, is still included in the list since it is the current working layer; the current working layer is always listed in the layer Browser and set to on.

1.10 Further Filtering

Now try experimenting with the other filter tools. First try the Description text string filter. This filter will try to match the string you enter to the text in each layer description. By default the field contains only an asterisk (*), the wildcard character, matching all layer descriptions, and therefore all layers are shown in the list. The **Show Empty** filter takes precedence over the Description filter, which takes precedence over the Range filter. Since layer 5 is currently selected, make it the current working layer first so that layer 831 will drop from the list.

1. **Set Current**

2. Enter `Sphere*` in the List Filters Description field.

Now the layer Browser lists only the layers whose description begins with "Sphere":
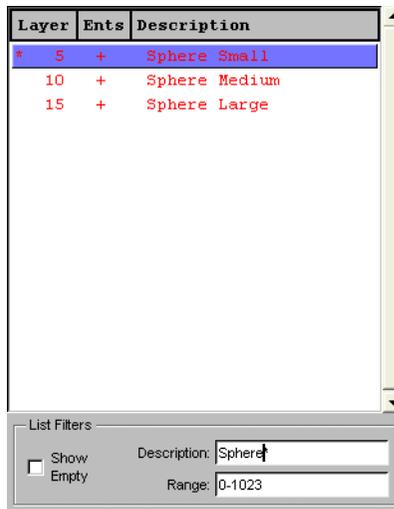
3. Enter `Cube*` in the List Filters Description field.

Now the layer Browser lists all of the layers whose description begins with "Cube", plus layer 5, the "Sphere Small" layer.
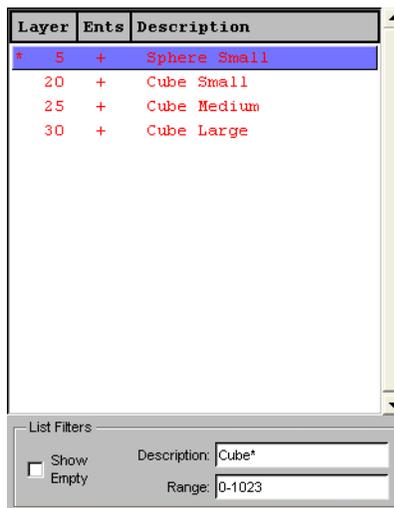
This illustrates the fact that the current working layer will always be shown in the list. The current working layer will also always be on. If you wish to turn off the current working layer so that its entities are no longer displayed, you must select a new current working layer. This is done by simply selecting a single layer in the layer Browser list and using the **Set Current** command. Feel free to experiment with these list filters more when the tutorial is complete. For now, return to the full list of all layers containing entities.

4. Enter `*` in the List Filters Description field.

1.11   More Database Organization

Layer 0 contains all of the entities used to create the spheres and cubes. All of the entities are currently disabled. Reorganize these now into layers 1, 2, and 3 with layer 1 containing all point entities, layer 2 containing all line entities, and layer 3 containing all circular arc entities. To make reassignment of the entities easier, isolate layer 0 and use the **DB Pick Mask** command.

1. Select layer 0 from the layer Browser.

2. **Isolate**

The Gridgen GUI should still be at the DATABASE COMMANDS menu; continue there:

3. **En/Disable**

4. Pick **All**

5. **Done**

All construction entities are now enabled and therefore visible in the Display window. Continue now with the assignments:

1. **Assign Layer**

2. **DB Pick Mask**

3. Surfaces **Off**

4. Curves **Off**

5. Others **Off**

6. Others **point**

This last command toggles the point entity back to active or unmasked, while all other entity types are masked off.

7. **Done**

8. Pick **All**

9. **Done**

10. Select layer 1 from the ASSIGN DB ENTITY LAYER Browser list.

Since layer 0 was isolated, all other layers are off, including layer 1. Therefore since the database points were assigned to layer 1, they are no longer visible in the Display window. Continue on to line and arc assignments.

1. **DB Pick Mask (Active)**

The "Active" addition to the button text indicates that a mask is in effect so that some entity types will not be available for picking.

2. Others **Off**

3. Curves **line**

This last command toggles the line entity back to active or unmasked, while all other entity types are masked off.

4. **Done**

5. Pick **All**

6. **Done**

7. Select layer 2 from the ASSIGN DB ENTITY LAYER Browser list.

Now the line entities disappear from the Display window. Since no other entity types other than circular arcs remain in this layer, restore the mask settings back to default before assignment.

1. **DB Pick Mask (Active)**

2. Surfaces **On**

3. Curves **On**

4. Others **On**

5. **Done**

6. Pick **All**

7. **Done**

8. Select layer 3 from the ASSIGN DB ENTITY LAYER Browser list.

Now the Display window is again empty. Use *Shift+m* to pop the Layer Manager panel back to the top.
Then use Status: **All On** to turn all layers back on. All entities should now be visible, including the points,
lines and arcs, as shown below.
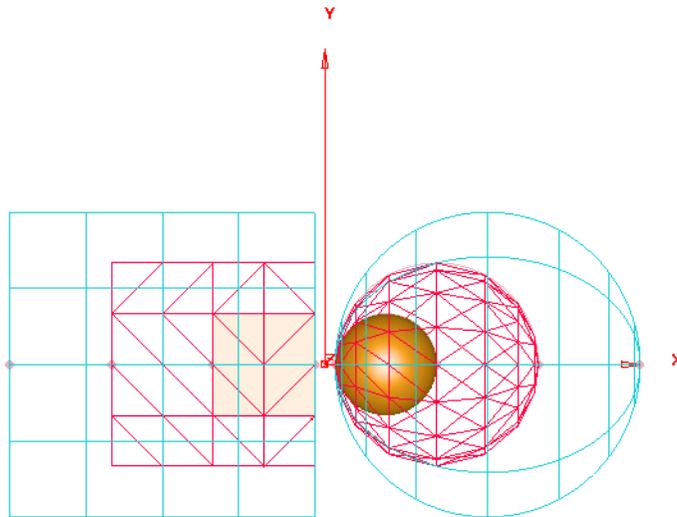
1.12   More Descriptions

Now provide some descriptions for the three new layers in use for the construction entities. Continuing with
the Layer Manager panel:

1. Select layer 1 using the mouse cursor and left mouse button.

2. **Description**

3. `Construction Points`

4. **OK**

5. Select layer 2.

6. **Description**

7. `Construction Lines`

8. **OK**

9. Select layer 3.

10. **Description**

11. `Construction Arcs`

12. **OK**

Your layer Browser list should now look like the one below. Note that since we previously set layer 0 to the current working layer, it is still shown in the list, even though it is now empty. But this is probably a good setup, since any newly created entities will now be placed in this top layer in the list which is clean.

Layer	Ents	Description
* 0		Construction All
1	+	Construction Points
2	+	Construction Lines
3	+	Construction Arcs
5	+	Sphere Small
10	+	Sphere Medium
15	+	Sphere Large
20	+	Cube Small
25	+	Cube Medium
30	+	Cube Large

1.13   Saved Sets

Finally, to get a handle on quickly displaying certain groups of objects, create saved layer sets for the construction entities, the spheres and the cubes.

1. Select layers 5 through 30 (click on 5 and drag the mouse down the list).

2. Status **Off**

3. Saved Layer Sets **Save**

The Saved Layer Set dialog box pops up:

Layer Set Name:	
Cancel	OK

4. Enter `Construction` in the Layer Set Name field.

5. **OK**

You now have a new saved layer set in the Saved Layer Sets Browser:
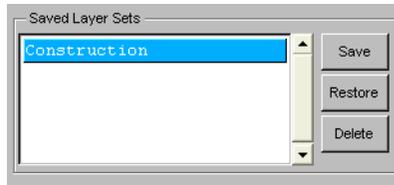
Continue on for the spheres and cubes.

1. Select layers 1 through 15.

2. Status **Toggle**

3. Saved Layer Sets **Save**

4. Enter `Spheres` in the Layer Set Name field.

5. **OK**

6. Select layers 5 through 30.

7. Status **Toggle**

8. Saved Layer Sets **Save**

9. Enter `Cubes` in the Layer Set Name field.

10. **OK**

Now the Saved Layer Sets Browser should be updated as below.
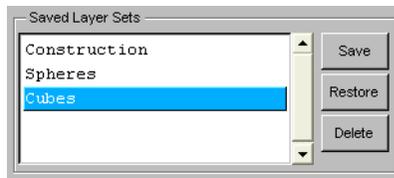
To restore a saved set, select it in the list and use **Restore** or simply double click the entry in the list. You can remove a set by selecting it and choosing the **Delete** command. Note that the save sets you have made also include the current working layer, layer 0. Verify this by setting some other layer as current, and then restoring one of the saved layer sets.

Congratulations! You have completed this tutorial. While you have not used all of the Layer Manager commands and capabilities in this tutorial, you have used nearly all of them and certainly the most frequently used. Feel free to experiment further with this model and the layering changes you have made. Other tutorial database models that would make interesting cases for layer experimentation would be the intersecting pipes model in Section 5, the 747 Nacelle model in Section 7, the LDI combustor model in Section 8, and the mixer model in Section 12.

2. 2D Bump: Basic Skills

2.1   Introduction

This section includes a brief tutorial to help you become familiar with the basic operations of Gridgen. If you have never used Gridgen before, you should work through this and the other tutorials before applying Gridgen to actual problems.

2.2   Topics Covered

This sample session is a short introduction to using Gridgen. Basic Gridgen skills you will learn are:

• creating connectors, the edge curves on which grid points are distributed, using the **Add Segment** command

• copying and translating connectors, using **Copy**

• creating structured domains, which are quadrilateral surface grids, using **Assemble Edges**

• Run Solver **Structured** to improve a domain's grid quality through application of an elliptic PDE method

• creating unstructured domains, which are triangular surface grids

2.3   Background

Work through this tutorial to familiarize yourself with the basic operations of Gridgen.

2.4   Geometry

For this brief introduction to Gridgen we will generate a 2D grid for a rectangular domain with a circular bump on one wall. The table below gives the coordinates of important points in Figure 18.2.

Bump Coordinates

label	x	y	z
A	0.0	-8.0	0.0
B	-20.0	0.0	0.0
C	-10.0	0.0	0.0
D	10.0	0.0	0.0
E	20.0	0.0	0.0
F	20.0	20.0	0.0

Bump Coordinates

label	x	y	z
G	−20.0	20.0	0.0
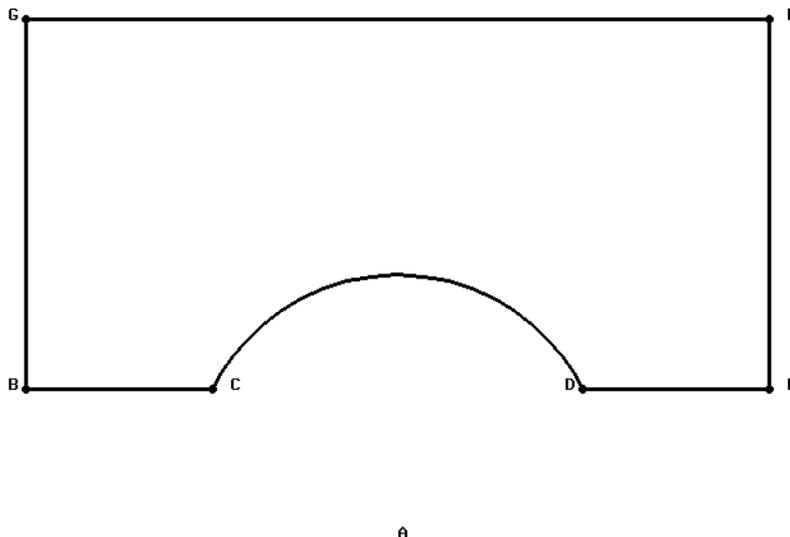
Figure 18.2 Schematic of the Bump Geometry

This is purely a contrived problem for demonstration purposes; however, similar problems arise when studying fairings on aircraft, constricted channel flows, wavy wall heat exchangers, and many other practical engineering problems.

2.5   Start Gridgen

Start Gridgen by typing the string `gridgen` on Unix workstations or by simply double-clicking the Gridgen shortcut on your PC's desktop. After a few seconds Gridgen will start, and you will see the Gridgen logo in the Display window. The logo will disappear after a few seconds or as soon as you select your first menu button.

Navigating through Gridgen's menu system is covered in depth in Section 2 of the User Manual. If you do not want to read Section 2 of the User Manual before running this sample session, just remember to use the *right mouse* button to select menu and screen items (assuming that the **Left Pk** has not been set in the DIS-PLAY COMMANDS menu).

The first step in running Gridgen is to select the software (Section 17.1 of the User Manual) to be used in the analysis. This will ensure that the grid generated by Gridgen is consistent with the capabilities of the analysis software. For this tutorial, we will not be performing an analysis using this grid. Therefore, we will use the generic analysis software, which is the default.

The geometry for the sample problem is simple enough we can create it directly without a database. We are ready to begin generating a grid.

2.6   Draw Connectors

Connectors are 3D curves made up of one or more segments connected end-to-end. In other words, a con-

nector is a composite curve. A *segment* is a particular type of curve such as a line or conic section ([Section 6.2](#) of the User Manual). Therefore, a single connector may consist of a line followed by a circle followed by another line. The segments interpolate user specified *control points.*

There are three steps in creating a connector:

1. Define its *shape.*

2. Assign it a *dimension* (number of grid points).

3. *Distribute* the grid points.

The first connector you will create is the circular arc section from point C to point D. The center of the arc is at point A. From Gridgen's MAIN MENU:

1. Select **Connectors**. This will take you to the CONNECTOR COMMANDS menu.

2. Select **Create** (or type *n*) to go to the CREATE CONNECTOR menu.

There are three main techniques for creating connectors:

• **Add Segment** is for general connector creation.

• **2 Point Connectors** is for rapid creation of connectors composed of lines or database constrained lines.

• **On DB Entities** is for turning database curves and edges of database surfaces into connectors in one step.

For this example:

1. Select **Add Segment**.

2. Select Rational **Circle** to begin creating arc CD.

You are now ready to begin creating a circular arc using commands on the CIRCULAR ARC menu. Refer to Figure 18.3 for the following discussion. On Gridgen's screen, immediately above the menu is a rectangular area called the Blackboard window. Gridgen reports topical information in this window. During connector creation, you will see information about the number of segments in the connector, how many grid points have been assigned to it (dimension), and control point locations. The blackboard also shows the x, y, and z coordinates of Gridgen's 3D cursor (the white + in the Display window). The cursor can be moved by holding down the *right mouse* button and moving the mouse (assuming that **Left Pk** has not been set in the DISPLAY COMMANDS menu). As the 3D cursor moves, its coordinates listed in the Blackboard window are updated.

```
        0 Blocks, 0 Domains
        0 Connectors, 0 Nodes
        0 DB ents, generic 3D



     CIRCLE:
     Enter the starting point
     of the circle.


     Connector Segments:    1
     Connector Dimensions:   0
     Cross Position:
        x =  0.000000E+000
        y =  0.000000E+000
        z =  0.000000E+000









     ════════CIRCULAR ARC════════
     Add CP  o │at +        padEnt
               │by Picking   pad0
               │via Keybrd   pad.
               │△xyz via Keybrdo
               │by Linear Proj (
               │at Clsest DB Pt)
     Erase Last Point        del
     Movement Restrictions     m
     Move + to Screen Center   c
     Rotation Pt. To +         t
     Sensit│Decr ←│Reset↓│Incr →
     Done - Save Segment     ent

     Abort - Don't Save      esc
     Display Commands         ^a
     Help                      ?
```

Figure 18.3 The Gridgen Window is Divided into Several Subwindows

Above the Blackboard window is an area called the Message window. This is where Gridgen provides cues for the input it expects from you. In this instance, it should say:

```
CIRCLE:

    Enter the starting point
    of the circle.
```

You have six options for entering the location of the starting point:

Add CP ☐ **at +** will add the point at the current position of the 3D cursor (as shown in the Blackboard window).

Add CP ☐ **by Picking** will add the point at the pickable grid or database point underneath the 3D cursor.

Add CP ☐ **via Keybrd** will allow you to type the x, y and z coordinates of the point in the Text Input window. (The Text Input window is the bar immediately above the Blackboard window.)

Add CP ☐ **△xyz via Keybrd** will allow you to type offsets of the current 3D cursor x, y and z coordinates to position the new control point.

Add CP □ **by Linear Proj** will add the point by projecting from the current 3D cursor location in a direction normal to the plane of the screen until a database entity is hit.

Add CP □ **at Clsest DB Pt** will add the point by projecting from the current 3D cursor location to the closest point on a database entity.

For this tutorial:

1. Select Add CP □ **via Keybrd**

2. Enter `-10, 0, 0`

After entering point `C`'s coordinates, look at the Blackboard window. It should indicate that there is 1 control point and the cross position is (`-10,0,0`). The Message window should be prompting you to enter the ending point of the circle:

> ```
> CIRCLE:
> Enter the ending point
> of the circle.
> ```

1. Select Add CP □ **via Keybrd**

2. Enter `10, 0, 0`.

 Now the Message window should display:

> ```
> EDIT CIRCLE POINT:
> Move the selected point
> and select from the
> options below.
> ```

This is your indication that a third point is needed to define the circular arc. You have a choice of entering a point that lies on the arc or entering the center point of the arc. Since you know the circular arc center point you may enter its coordinates using:

1. Select **Store Center via Kbrd.**

2. Enter `0, -8, 0`

The Gridgen Display window should now look like Figure 18.4.
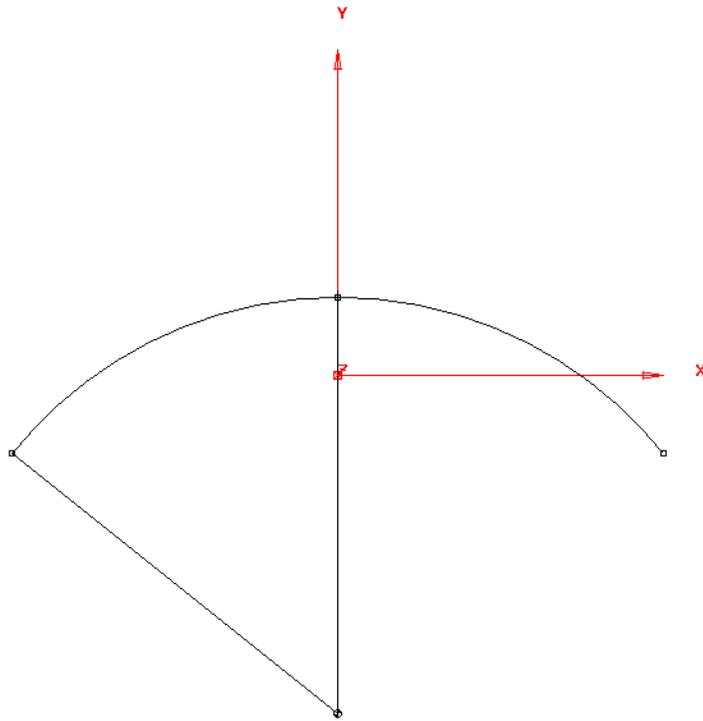
Figure 18.4 Creating the Bump from a Circular Arc

That completes the definition of the circular arc shape.

1. Select **Done - Save Segment** from the EDIT CIRCULAR ARC menu to go back to the CREATE CONNECTOR menu.

2. Select **ReDimension** to assign a number of grid points to the connector. This takes you to the REDI-MENSION CONNECTORS menu.

3. You might want to note that the Apply **dimen** button is set so when you press the From **keybrd,** the Message window should will prompt you for the number of grid points to be assigned to the connector.

4. Enter 15.

5. Finish dimensioning the connector by selecting **Done-ReDimension.**

The final step in connector creation is to distribute the grid points to satisfy whatever clustering requirements you have. For this problem, we will use equal spacing, the default, so no further action is required.

1. Select **Done-Save Connectors** to complete this connector and return to the CREATE CONNEC-TOR menu.

Next, you will create the connector from point B to point C.

1. Select **Add Segment** to go to the ADD SEGMENT menu.

2. Connector <u>BC</u> is a straight line segment, so select 3D Space **Line** to go to the LINE menu.

3. Enter the control point at B by selecting Add CP □ **via Keybrd.**

4. Enter the coordinates of point B, `-20, 0, 0`.

5. Point C was created earlier when you made the circular arc connector. You can take advantage of this fact by moving Gridgen's 3D cursor over point C, hold down the *right mouse* button and move the mouse.

6. Select Add CP □ **by Picking**.

7. Press **Done - Save Segment** to complete the definition of the shape of this connector and return to the CREATE CONNECTOR menu.

8. Select **ReDimension** to assign a number of grid points to this connector.

9. Click on From **keybrd**.

10. Enter 7 for the number of grid points.

11. **Done- ReDimension**

12. The default equal spacing is fine for this connector. Select **Done-Save Connectors** to complete the connector definition and return to the CREATE CONNECTOR menu.

We will create the next connector, the line segment DE, in a slightly different fashion. You should now be at the CREATE CONNECTOR menu.

1. Select **Done Creating Conns** to return to the CONNECTOR COMMANDS menu.

2. Now select **Copy** to go to the COPYING CONNECTORS menu.

3. Move the cursor over connector BC (it will change to a pink color when the cursor is over it).

4. While connector BC is highlighted, press and release the *right mouse* button.

5. Select **Done** to tell Gridgen you do not want to pick any more connectors.

The Message window will display:

```
TRANSLATE
Pick a handle point to
control translation or
select from the options
below.
```

1. Position the pointing finger cursor over the control point at the left end of the connector.

2. Press and release the *right mouse* button. You have selected this node as the translation control point (the handle).

The Message window will display:

```
TRANSLATE
Move the handle point with
the mouse, or select from
the options below.  (Solid
square is initial handle
location)
```

1. Move the point by moving the mouse to the right while holding down the *right mouse* button. You should see the entire connector move as you move the mouse.

2. When the 3D cursor (the +) is over point D, select Set Handle **by Picking** to finish moving the connector.

3. **Done - Translate**

4. Press the **Done - Save Connectors** button to return to the CONNECTOR COMMANDS menu.

If everything is going well, your Display window should look like Figure 18.5.

Figure 18.5 Three Connectors form the Bottom Edge of the Bump

Starting from the Connector menu, create connector <u>BG</u> by:

1. **Create**

2. **Add Segment**

3. 3D Space **Line**

4. Position the 3D cursor over point B.

5. Add CP ☐ **by Picking**

6. Add CP ☐ **via Keybrd**

7. Enter the coordinates of point G, `-20, 20, 0.`

8. **Done - Save Segment**

9. **ReDimension**

10. From **keybrd**

11. Enter `21.`

12. **Done - ReDimension**

13. **Done-Save Connectors**

14. **Done Creating Conns**

Connector <u>EF</u> is identical to connector <u>BG</u> so you can copy <u>BG</u> and translate it to the position of <u>EF</u>.

1. **Copy**

2. Position the cursor over connector BG.

3. Press and release the ***right mouse*** button.

4. **Done**

5. Select the bottom end of connector BG as the translation control point (the handle) by positioning the

cursor over the bottom and pressing and releasing the ***right mouse*** button.

6. Translate the connector copy to its new position by moving the mouse with the right mouse button pressed.

7. Set Handle **by Picking**

8. **Done - Translate**

9. **Done-Save Connectors**

The final connector is <u>GF</u>. Starting from the CONNECTOR COMMANDS menu:

1. **Create**

2. **Add Segment**

3. 3D Space **Line**

4. Position the cursor over point G.

5. Add CP □ **by Picking**

6. Position the cursor over point F.

7. Add CP □ **by Picking**

8. **Done - Save Segment**

9. **ReDimension** to assign a number of grid points to connector <u>GF</u>. Since connector <u>GF</u> will be opposite connectors <u>BC</u>, <u>CD</u>, and <u>DE</u> in the final structured grid, it must have the same total number of grid points.

10. Position the cursor over connector <u>BC</u>.

11. Press and release the ***right mouse*** button.

12. Next, position the cursor over connector <u>CD</u>.

13. Press and release the ***right mouse*** button. (Note how the Blackboard window changes when you do this).

14. Finally, position the cursor over connector <u>DE</u> and press and release the ***right mouse*** button. Note that connector <u>GF</u> and the string of connectors <u>BC</u>, <u>CD</u>, <u>DE</u> have colored triangles at their end points. The color of the triangles provides a directional reference for copying the distribution of grid points. For this symmetric problem, it does not matter which direction the curves are running. If it did, you could use the **Reverse the Direction** button to swap the direction of the string of connectors**.**

15. **Done-ReDimension**

16. **Done - Don't Copy Dist**

17. **Done-Save Connectors**

18. **Done Creating Conns**

This is the last connector you need to create. Let's turn on display of the connector grid points so you can see how the grid points are distributed.

1. Press **Disp** in the Quick Keys.

2. Show **Con GPs**

3. **Done**

The Display window should look like Figure 18.6 on the following page.
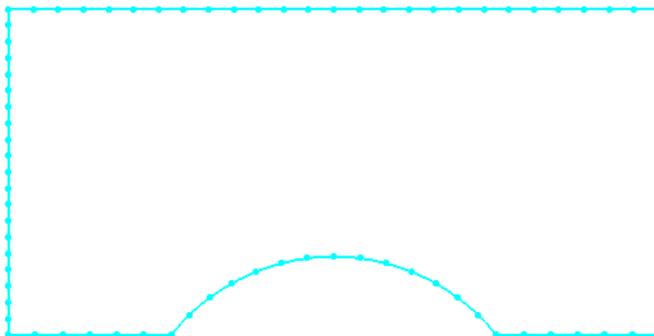
Figure 18.6 Grid Points are Distributed on the Bump's Connectors

2.7   Create a Structured Domain

You will now tell Gridgen how to make a surface grid from the connectors you just created.

1.   Select the quick key **Doms** from the bottom of the Menu window to go to the DOMAIN COMMANDS menu.

2.   Select **Create** to start making a domain.

The Message window will change to

```
CREATE A DOMAIN:
Specify the cell type and
then select a technique
below for constructing a
domain.
```

and the CREATE DOMAIN menu will appear. Make sure that the Cell Type **structured** radio button is selected. Then begin the domain creation process by selecting the loop of connectors that form the perimeter of the domain.

1.   Select **Assemble Edges.**

2.   Check the **Auto Complete** box.

The corners of the domain will be at points B, E, F, and G, so you can pick any connector except <u>CD</u> as the first one. For this sample, start with connector <u>EF</u>.

3.   Position the cursor over connector <u>EF</u>.

4.   Press and release the *right mouse* button.

5.   Connector <u>EF</u> forms one entire edge of the domain, so tell Gridgen you are ready to start defining the next edge by selecting **Next Edge**.

6.   Position the cursor over connector <u>GF</u>.

7.   Press and release the *right mouse* button. This connector is also an entire edge of the domain, so select **Next Edge**.

As soon as you tell Gridgen the second edge is complete, it will recognize that connector <u>BG</u> must be the third edge since it is the only other connector sharing node G. It has the right number of points to complete the third edge. Continuing the process, Gridgen recognizes connector <u>BC</u> must be the beginning of the fourth edge since it is the only connector sharing node B. However, it does not have enough grid points to complete the edge, so Gridgen continues by adding connectors <u>CD</u> and <u>DE</u> to the edge until it is finished. Gridgen then completes the domain and initializes the surface grid using transfinite interpolation (TFI). Your Display window should look like Figure 18.7.
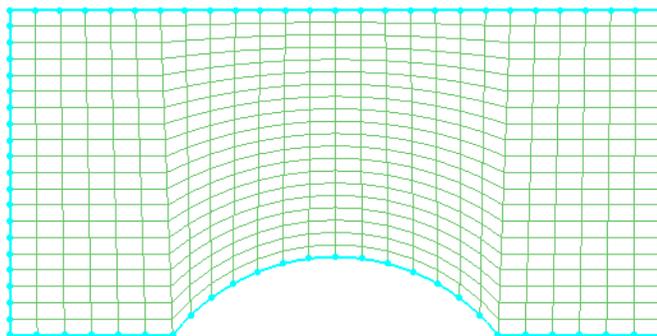
Figure 18.7 The Bump Domain is Initialized by Transfinite Interpolation

1.   Select **Done** to return to the DOMAIN COMMANDS menu.

2.8   *Refine the Structured Domain Grid*

The surface grid created automatically by TFI can often be improved by applying an elliptic PDE-based smoothing to the grid. Gridgen offers a variety of elliptic PDE methods with various controls for boundary spacing, orthogonality, and smoothness.

1.   Select Run Solver **Structured.**

2.   Pick **All**

3.   **Done**

4.   Elliptic Slvr **Run**

This runs the elliptic solver with the default settings. As the elliptic solver iterates, the grid display will be updated so you can monitor the progress of the surface grid.

This is a small grid, so it will not take long to reach a state of equilibrium. When the grid looks something like the one in Figure 18.8 you may stop the solver and save the grid by pressing the following button.

1.   **Done - Save**
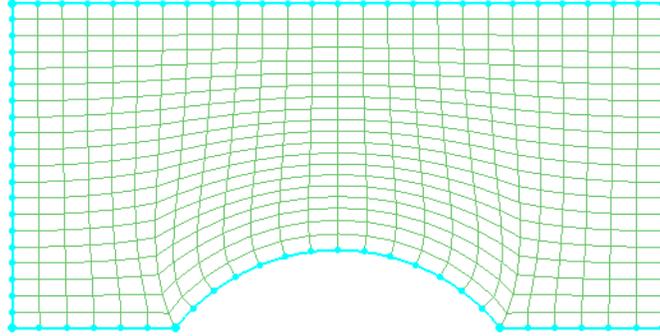
Figure 18.8 The Bump Domain After Running the Elliptic PDE Solver

2.9   Create an Unstructured Domain

It is also easy to create a triangular surface grid for the bump geometry. You can even use the connectors you have already created. Before beginning, delete the structured domain you just created. Starting from the DOMAIN COMMANDS menu:

1. **Delete**

2. Pick **All**

3. **Done**

4. **Delete Domains Only**

You just deleted the domain but kept the connectors, so you are ready to begin creating the unstructured domain immediately.

1. **Create**

2. Cell Type **unstructured**

3. **Assemble Edges**

4. Check **Auto Save.** (Selecting **Auto Save** will tell Gridgen to create a triangular surface grid as soon as it detects that you have chosen a closed loop of connectors.

5. Check **Auto Complete** (Selecting **Auto Complete** tells Gridgen that with each subsequent connector selected, it should try to figure out a possible closed loop of connectors on which to create a domain).

6. Position the cursor over connector <u>BC</u>.

7. Press and release the *right mouse* button.

8. The **Auto Complete** function figures out that a closed loop of connectors is present and creates an unstructured domain which is initialized and saved immediately by the **Auto Save** function.

9. **Done**

When you pick the last connector, Gridgen will initialize the triangular surface grid. Your grid should look something like the one in Figure 18.9.
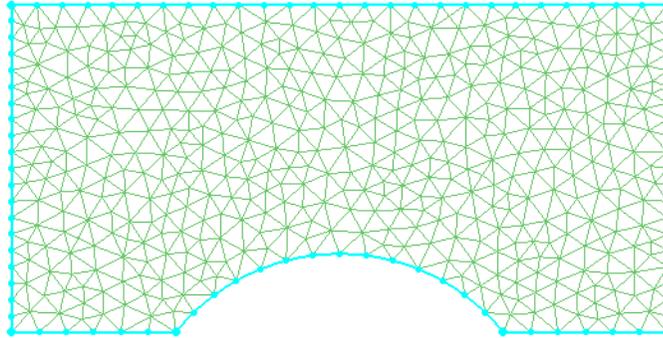
Figure 18.9 An Unstructured Domain for the Bump

2.10   Tutorial Wrap-up

This completes the Gridgen tutorial. If you want to save this grid:

1.   Select **I/O** from the Quick Keys.

2.   Gridgen **Export**

3.   Type In... **Name**

4.   Enter the file name for the grid.

5.   **Done**

When you are done, use **Quit Gridgen** from the MAIN MENU to exit the program.

3. 2D Airfoil: Re-Extrude

3.1   Introduction

This section includes a brief tutorial to help you become familiar with creating basic 2D extruded grids, as well as with Gridgen's Re-Extrude capability. This capability allows you to return to a previously saved extrusion and make changes to it. These possible changes are backing up, continuing from the previously saved final step, and changing attributes.

3.2   Topics Covered

This sample session is a short introduction to using Gridgen. Basic Gridgen skills you will learn are:

• importing a database model using Database **Import**

• setting connector defaults using the SET DEFAULT VALUES menu

• connector creation using **2 Point Connectors**

• creating a C-style airfoil grid using hyperbolic domain extrusion

• changing a saved extrusion using **Modify**, **Re-Extrude**

3.3   Background

You do not need to run any other tutorials before running this one. However, please read Section 2 of the User Manual to familiarize yourself with the operation of the Gridgen user interface before attempting to run this tutorial.

3.4   Geometry

For this introduction to Gridgen's **Re-Extrude** command we will work with a geometry model consisting of two curves forming a simple 2D airfoil. Our final grid will be a C-style 2D mesh. The model is shown in Figure 1.10 below.
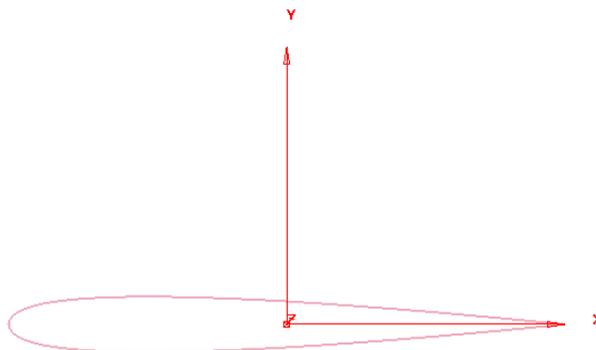
Figure 1.10 Database Geometry Model

This is purely a contrived problem for demonstration purposes; however, the operations to be demonstrated will be common to any models you work with in Gridgen.

3.5   Start Gridgen

Start Gridgen by typing the string `gridgen` on Unix workstations or by simply double-clicking the Gridgen shortcut on your PC's desktop. After a few seconds Gridgen will start, and you will see the Gridgen logo in the Display window. The logo will disappear after a few seconds or as soon as you select your first menu button.

3.6   Database Model Import

To begin the tutorial, import the composite geometry file, `reext.dba`. This file should be available in your Gridgen installation directory under *gridgen_home_path*/`examples/tutorial/reext`. You may wish to copy the file to some other working directory before proceeding. From the MAIN MENU:

1. **Input/Output**

2. **Database Import**

3. Use the Browser to navigate to the directory containing the copy of `reext.dba` and select it.

4. **Done**

In your Display window you should now see the database model as displayed in the previous figure.

3.7   Set Defaults

Before creating your initial connectors, you can save some time later by setting a few connector defaults now. From the MAIN MENU:

1. **Defaults**

2. Con Dim **dimen**

3. `31`

4. Con Dist **Bgn** Δ**s**

5. `0.01`

6. **Done**

All newly created connectors will now have a default dimension (number of grid points) of 31. They will also each have a spacing constraint of 0.01 at their beginning. See Section 3.1 of the User Manual for more details.

3.8   Connector Creation

You will now create three connectors that will eventually be extruded into your new C-style grid. The first two connectors will be the upper and lower airfoil connectors; the third connector will be the wake connector.   Again, from the MAIN MENU:

1. **Connectors**

2. **Create**

3. **2 Point Connectors**

4. Move the 3D Cursor (white cross) to the leading edge (LE) of the airfoil.

5. Add CP □ **by Picking**

6. Move the 3D cursor to the trailing edge (TE) of the airfoil.

7. Add CP □ **by Picking**

Since two database points on the same database entity were picked, Gridgen converts this connector to a database constrained line segment type. If you pick the same two points again, Gridgen will check for an alternative path on a different database entity. Once no alternatives are available, Gridgen will create a straight, 3D line segment type. Pick the same points again now.

8. Move the 3D Cursor to the LE of the airfoil.

9. Add CP □ **by Picking**

10. Move the 3D Cursor to the TE of the airfoil.

11. Add CP □ **by Picking**

You were instructed to place the 3D Cursor at the nose of the airfoil in each case since that is where you want your default clustering value to be applied. Picking these same points again would result in a straight 3D line segment connector since there are only two database curves for Gridgen to use for database constrained segments.

Now create the wake connector continuing with the **2 Point Connector** command. The 3D Cursor should still be at the TE of the airfoil.

12. Add CP □ **by Picking**

13. Add CP □ **via Keybrd**

14. `11`

15. **Done**

16. **Done Creating Conns**

The airfoil length is one unit (1.0), so entering 11 for the length of the wake connector puts the exit boundary at 10 body lengths downstream. This distance is a typical value for many analysis problems. Also, when you entered only a single number, 11, Gridgen used this as the x-coordinate and assumed the default values for y and z. To change only the y-coordinate value, you would enter ",11" so that Gridgen would use the default x and z. Likewise for updating the z-coordinate, enter ",,11".

Your final connector work involves this last wake connector. At its beginning it has the default spacing you set earlier. You really want it to match the spacing at the TE of the airfoil connectors. You can **Modify** it to make this change.

1. **Modify**

2. Pick the wake connector

3. **Done**

4. **ReDistribute**

5. **Begin.** △

6. `-1`

Entering negative one (-1) instructs Gridgen that you wish to copy a spacing from the other side of this particular BreakPoint. It highlights in a deep pink color the candidate connectors for you to select one from. The spacing from the end of the candidate connector you choose will be copied to the connector your are currently modifying.

7. Pick either airfoil connector since both have the same TE spacing.

8. **Done ReDistributing**

9. **Done-Replace Connectors**

This completes the connector creation phase of this tutorial. Use the *r* hot key on the keyboard to have Gridgen resize the view extents for the model and return to the original orientation. Your model should now appear as in the figure below.
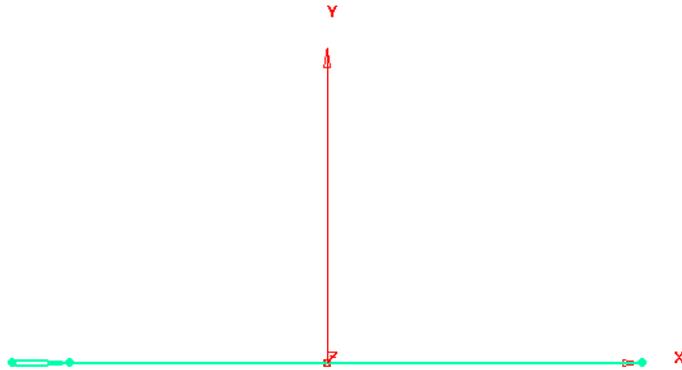
3.9   Domain Extrusion

Now you can create your extruded C-style domain using the three connectors. For this next step you may wish to zoom back up on the airfoil to make connector selection a little easier. Use the *z* hot key to pop up the zoom box and size it around the airfoil to obtain a view similar to the one below. If you make a mistake with the zoom box, use *Shift+z* to undo the previous zoom operation and try again.

Now begin your extrusion work starting from where you left off in the CONNECTOR COMMANDS menu.

1. **Doms**

2. **Create**

3. Cell Type **structured**

4. Extrude **From An Edge**

Now Gridgen wants you to define a single beginning edge which will be extruded into a single domain.

Using the **From An Edge** option allows the wake connector to be used twice in the original marching front and have the marching direction be maintained consistently along the entire front. The alternative, **From Cons**, only allows the wake connector to be used once.

5. Select the wake connector.

6. Select the upper airfoil connector.

7. Select the lower airfoil connector.

8. Select the wake connector again.

9. **Save Edge**

You are now placed in the DOMAIN EXTRUSION menu. Here you will choose the extrusion method, set up attributes, and set some criteria to have the extrusion stop automatically if poor cells are detected.

10. Type **Hyp**

11. **Set Attributes**

Notice immediately that the marching direction vectors rendered around the edge appear to be pointing through the interior of the airfoil. In other words, vectors from the upper airfoil connector are point down and vectors from the lower airfoil connector are point up. Since the same wake connector is used twice, it is impossible to tell what those vectors are doing. However, you can assume that they are consistent with the ones on the airfoil. Fix this problem first.

12. March **Flip**

Keep the default smoothing control settings and initial spacing of 0.01. You should set a boundary condition, though, that will keep the downstream end of the wake at a constant x-coordinate value.

13. **Boundary Conditions**

14. Edges **All**

15. Edges **Set**

16. Symmetry **X**

17. **Done**

Now set up some automatic cell checking using **Set Stop Criteria**.

18. **Set Stop Criteria**

19. Toggle on all four Jacobian checks.

20. **Done**

Now run the extrusion.

21. **Run N**

22. 40

23. **Done**

Obviously no poor Jacobians were encountered during the extrusion, so all 40 steps were completed. You have saved a nice looking extruded domain that should look like the one below after using the *r* hot key to restore orientation.
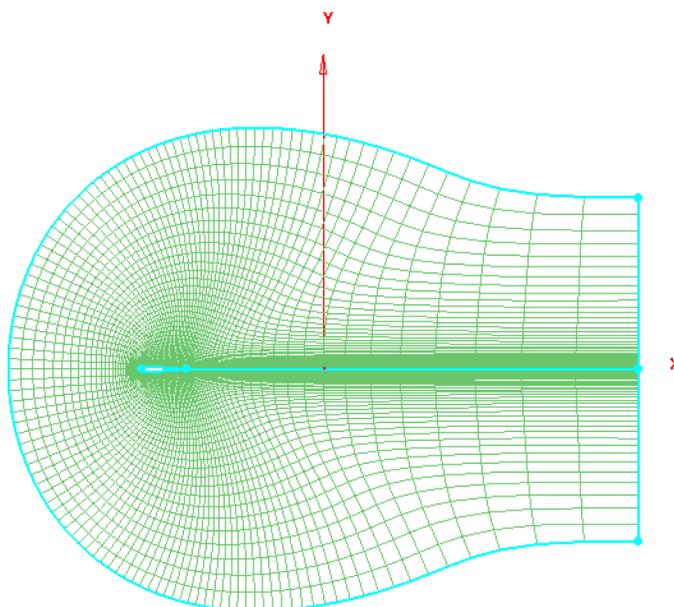
3.10   Re-Extrude

However, after some examination, you realize you likely saved the domain too quickly. The final boundary connector saved from the extrusion does not really look like it is an adequate distance from the airfoil. You set the outflow boundary downstream to 10 units from the airfoil TE. The C-boundary should be a similar distance. Rather than do all of the extrusion settings over again, you can use the **Re-Extrude** command to simply re-enter the extrusion where you left off prior to saving the domain. While still in the DOMAIN COMMANDS menu:

1. **Modify**

2. Select the domain

3. **Done**

4. **Re-Extrude**

You are now back exactly where you were prior to saving the extrusion earlier. Let Gridgen's stop criteria figure out for you how many steps to run.

5. **Set Stop Criteria**

6. Toggle on **Total Height**

7. **Total Height Set**

8. `10`

9. **Done**

10. **Run**

The **Total Height** stop criteria instructs Gridgen to stop automatically when the first grid point on the marching front reaches that distance from its original location. Using the Run command tells Gridgen to continue the extrusion indefinitely, or until a stop criteria has been reached. This extrusion stops automatically at marching step 49 as shown in the Blackboard window in the figure below.

```
Extrusion -   ( RUN )

Marching Step #   49
Input Δr(k+1) :9.702E-001
◆ Max Δr(k+1) :1.409E+000
 Layer Stop Criterion
Total Height :1.057E+001
Min. Jacobian :2.414E-001




═════ SELECT OPTION═════
Pause - delete step      ent

Pause - save step        esc
Continue                 y
Help                     ?
```

You can see also that the first point on the marching front to hit the **Total Height** criteria is actually at 10.57. While the entire boundary is obviously not this distance from the airfoil, this grid is much more in line with your requirements and is likely sufficient. Your choices now are to **Pause - delete step** which will pause the extrusion and remove marching step 49, **Pause - save step** which will pause the extrusion and keep marching step 49, or to **Continue** the extrusion, which will only pause again at marching step 50 as more points meet the **Total Height** criteria. Keep the extrusion as it is.

11. **Pause - save step**

12. **Done**

You now have an opportunity to abort this modification and return to your original extrusion or replace the original with your newly extruded domain. You will choose to replace the original domain.

13. **Replace Domains**

At this point Gridgen replaces the original domain, updating all the connectors as necessary. Your new extruded domain should appear as in the figure below.
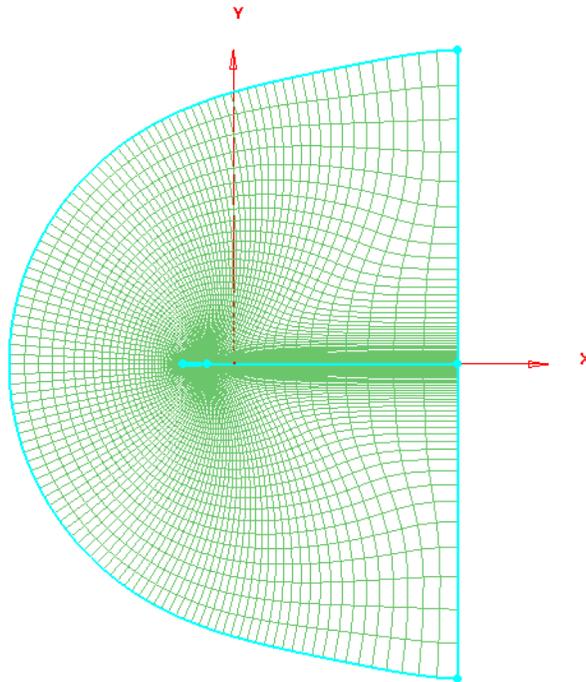
3.11   2D Block Creation and AS/W Settings

To make this a complete grid suitable for analysis in the Fluent 2D analysis software (AS/W), select the AS/W, create the 2D block, set boundary conditions (BC's) and export your analysis data. Keep in mind that Gridgen requires that blocks exist before you can set BC's or export analysis data. Therefore, even for 2D grids, you must create blocks; domains alone are not sufficient. Starting from where you left off in the DOMAIN COMMANDS menu:

1. **AS/W**

2. **Select Analysis S/W**

3. **2D**

4. **FLUENT**

5. **Next Page**

6. **Done**

It was necessary for you to make this setting first, so that Gridgen is now in 2D mode and will allow creation of a 2D block from your single domain.

7. **Blks**

8. **Create**

9. Cell Type **structured**

10. **Assemble Faces**

11. **Add 1st Face**

12. Pick the domain.

13. **Save the Face**

14. **Done-Save Blocks**

Now you are ready to set BC's and export your analysis data.

15. **AS/W**

16. **Set BCs**

Note that the wake connector is in the list twice and has already been set as a Type 1 block connection, which is an inter-block connection. No further settings are necessary for that part of the boundary.

17. Select the two outflow connectors.

18. **Done**

19. **Next Page**

20. **Outflow**

21. Select the C-boundary connector.

22. **Done**

23. **Velocity Inlet**

Note there are only two BC's left to set. Those obviously correspond to the airfoil connectors.

24. Select the remaining unset edge connectors from the Browser list.

25. **Done**

26. **Wall**

27. **Done**

Your new grid is ready for export to a Fluent case file.

28. **Export Analysis Data**

29. Type In... **Name**

30. `airfoil`

You now have a new analysis data file, `airfoil.cas`, ready to run. Congratulations! You have completed this tutorial.

4. Swept Ramp: Basic Multi Block Structured Grid

4.1 Introduction

This tutorial will give you a guided tour of the Gridgen process for creating a simple, two block, structured grid for the flow field over a swept ramp.

Note that working through this tutorial will not make you a Gridgen expert. It will also not make you a grid generation expert. Grid generation is not a trivial task that can be mastered in a couple of hours. Generating a good grid involves both skill and patience and Gridgen has been designed to minimize, as much as possible, the required levels of both.

You should complete this tutorial before attempting any of the other tutorials, since they cover advanced topics and assume you already understand the basics that are covered in this session.

4.2 Topics Covered

This tutorial covers the basics of using Gridgen. Basic Gridgen skills you will learn during this tutorial session are:

- creating connectors, the edge curves on which grid points are distributed, using 3D Space **Line** in the **Add Segment** command
- copying and translating connectors, using the Others **Copy** command
- redistribute grid points on connectors using **ReDistribute**
- creating domains, which are surface grids, using **Assemble Edges**
- elliptically smoothing domains to improve surface grid quality, using the Run Solver **Structured**
- use **Copy** and **Modify** to speed grid generation
- creating blocks, which are volume grids, using AutoSave **Face**
- elliptically smoothing blocks to improve volume grid quality
- use **Examine** to visually examining the volume grid
- setting boundary conditions for analysis software, using the **Analysis S/W** commands

4.3 Background

You do not need to run any other tutorials before running this one. However, please read Section 2 to familiarize yourself with the operation of the Gridgen user interface before attempting to run this tutorial.

4.4 Geometry

The swept ramp configuration you will use for this tutorial is shown in Figure 19.1. Presumably someone will use this grid with a computational fluid dynamics code to compute the flow field over this shape. The CFD solver is not part of Gridgen.
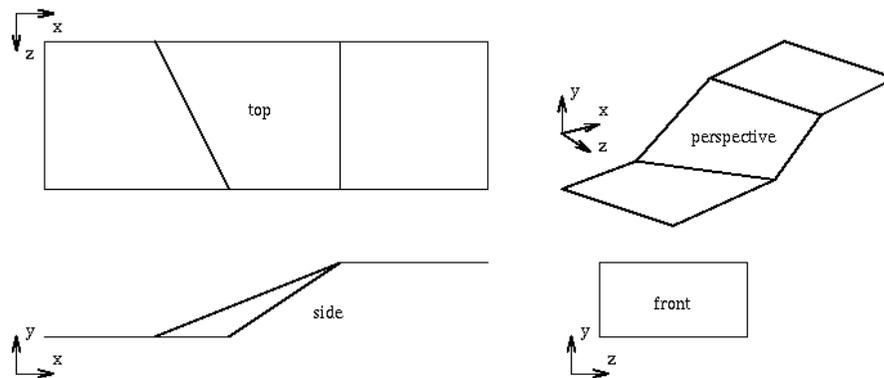
Figure 19.1 The Swept Ramp Geometry, Flow will be in the Positive X Direction

The grid you will generate consists of two blocks arranged end-to-end as shown in Figure 19.2. Within each block one family of grid lines will run in the flow direction, one family of grid lines will run from the ramp to the far field boundary, and one family of grid lines will run in the z direction across the ramp. The choices of blocking system and grid topology are up to you; this particular blocking system is only one of many that could be used for this swept ramp. The requirements of the particular analysis software you are using should be considered when planning the grid.
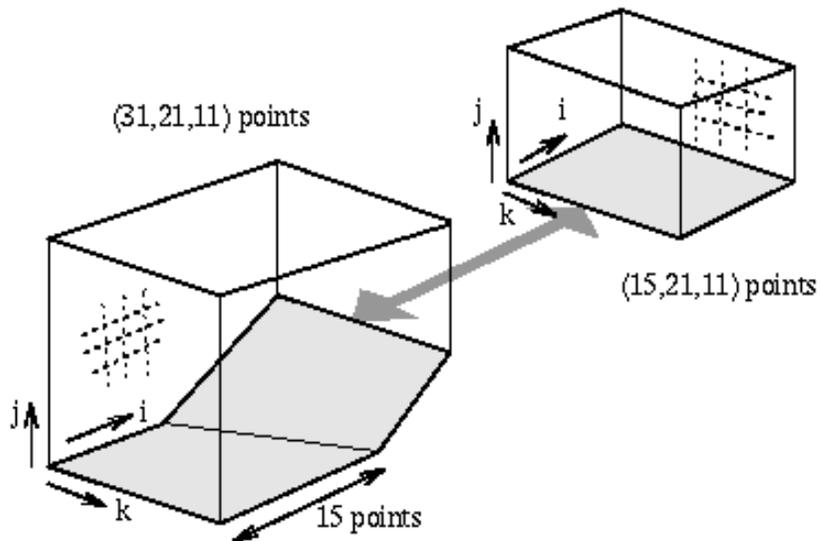
Figure 19.2 The Two Block System to be used for the Swept Ramp

Now that a topology for the blocking system has been chosen, we can begin creating the grid. For convenience we have assigned a naming convention to important points in the grid. The point's labels and coordinate values are shown in Figure 19.3 and followed by a tabular list. Note these particular locations for the outer boundary points may not be adequate for a CFD simulation; the outer boundary may need to be farther away from the ramp in order to minimize the boundary condition's influence on the flow.
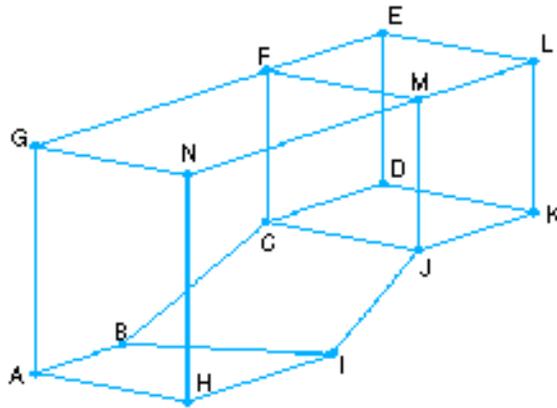
Figure 19.3 The Ramp's Labeled Coordinates

List of Swept Ramp Coordinates

Label	X	Y	Z
a	0.00	0.00	0.00
b	0.75	0.00	0.00
c	2.00	0.50	0.00
d	3.00	0.50	0.00
e	3.00	1.50	0.00
f	2.00	1.50	0.00
g	0.00	1.50	0.00
h	0.00	0.00	1.00
i	1.25	0.00	1.00
j	2.00	0.50	1.00
k	3.00	0.56	1.00
l	3.00	1.50	1.00
m	2.00	1.50	1.00
n	0.00	12.50	1.00

4.5   Getting Started

Begin by starting Gridgen, presumably by typing the string `gridgen` on Unix or by simply double-click-
ing the Gridgen shortcut on your PC's desktop. The Gridgen logo will disappear after a few seconds or as
soon as you invoke your first menu command. The first step in running Gridgen is to select the software
([Section 17.1](#)) to be used in the analysis. This will ensure the grid generated by Gridgen is consistent with

the capabilities of the analysis software. For this analysis we will be using the STAR-CD flow solver (by default Gridgen selects the generic solver). To select STAR-CD do the following starting from the MAIN MENU:

1. **Analysis S/W**

2. **Select Analysis S/W**

3. Press the radio buttons for **3D** and **STAR-CD** (you may have to go to the menu's second page with the **Next Page** button to do this).

4. **Done**

5. **Done**

Use of the **3D** button above tells Gridgen you will be creating a 3D multiple block grid; in other words, the blocks will consists of I x J x K points where K ≥ 2. In order to create 2D multiple block grids (I x J x 1) simply choose the **2D** button instead of the **3D** button. Verify you have selected the 3D STAR-CD flow solver by checking the display in the Diagnostic window.

4.6   Drawing Connectors

You are now ready to begin creating the geometric elements that will comprise the grid. The process begins with the lowest level element in the grid element hierarchy, the connector (Section 6). Connectors are 3D curves made up of one or more segments connected end-to-end. In other words, a connector is a composite curve.

A segment is a particular type of curve such as a line or conic section (Section 6.2). A connector may be composed of several segments, such as a line followed by a circle followed by another line. The segments interpolate a set of user specified control points.

There are three major steps in creating a connector:

1. Define the shape.

2. Specify the number of grid points.

3. Distribute the grid points.

The first connector you will create is the connector AG. From the table of coordinates, you see this line spans from a(0,0,0) to g(0,1.5,0). It is hoped you can also see this particular line may be created with one connector consisting of one Line segment. To draw this connector do the following:

1. **Connectors**

2. **Create**

3. **Add Segment**

4. 3D Space **Line**

5. Add CP ☐ **via Keybrd**

6. 0, 0, 0

7. Add CP ☐ **via Keybrd**

8. 0, 1.5, 0

9. **Done - Save Segment**

You have just completed one third of the connector creation process (shape definition). The connector shape

consists of one segment (Line). The segment was defined by two control points: `(0,0,0)` and `(0,1.5,0)`.

The image on the screen is probably small because Gridgen's default viewport size is larger than the particular grid we are creating. To get a better look at the grid zoom in on the connector.

1. Press and hold the *middle mouse* button.

2. Move the mouse toward you (south on the mouse pad).

3. Continue until the connector is comfortably sized in the screen.

4. Release the *middle mouse* button.

You should have noticed while the *middle mouse* button was pressed the cursor looked like a small magnifying glass, reminding you that you were zooming the image.

Let us continue with the second step of connector creation: assigning a number of grid points (Section 9.1). From Figure 19.2, you can see you need to put `21` grid points on this vertical connector. To enter this information in Gridgen do the following:

1. **ReDimension**

2. From **keybrd**

3. `21`

4. **Done- ReDimension**

The third and final step of connector creation is distributing the grid points along the connector (Section 9.3). Since this grid is to be used for a simulation of the viscous flow over the ramp surface it is reasonable to cluster grid points toward the ramp surface. The requirements of your analysis software and the particular problem you are analyzing determine where and how much to cluster. By default Gridgen will equally space the grid points along the connector. To specify the distance of the first grid point above Point A is `0.01`, do the following:

1. **ReDistribute**

2. **Begin.** Δ

3. `0.01`

4. **Done ReDistributing**

5. **Done - Save Connectors**

6. **Done Creating Conns**

In case you are curious as to how the grid points on this connector appear you can use a display command to turn this on. From the button menu:

1. **Disp**

2. Show **Con GPs**

3. **Done**

This completes the connector creation process. Each of the green dots on the screen is a grid point. There should be 21 grid points and they should be clustered toward the ramp surface as shown in Figure 19.4.

Figure 19.4 Grid Points are Clustered Towards the Ramp Surface for First Connector

Notice the connector is rendered in a light green color signifying that a number of grid points has been assigned (as opposed to a bright green color signifying that no grid points have been assigned). The green dots at either end of the connector are the nodes. Recall nodes are the control points at the ends of a connector.

Create another connector and this time we will look more closely at what Gridgen is doing. The second connector to draw is connector <u>AB</u> in Figure 19.3, the horizontal connector from the leading edge of the grid to the leading edge of the ramp. We will begin by doing the following:

1. **Create**

2. **Add Segment**

3. 3D Space **Line**

Gridgen is now in segment drawing mode. The first item you should identify is the small white cross in the Display window. This cross is Gridgen's 3D cursor and is used to define the control point locations. Also identify the Browser window on the left side of the screen. It will be displaying information about the connector, segment, and control point you are currently creating. In fact, it should currently look like:

```
Connector Segments: 1
Connector Dimensions: 0
Cross Position:
  x = 0.0000000E+00
  y = 1.5000000E+00
  z = 0.0000000E+00
```

The Display window and Browser window both reveal the 3D cursor is located at point G from Figure 19.3. However, we are drawing connector <u>AB</u>. Instead of typing in the control point coordinates as we did on the previous connector, this time we will use another control point method. Specifically we will tell Gridgen to place a control point at the node corresponding to Point A. Do the following:

1. Press and hold the ***right mouse*** button.

2. Move the mouse south and the 3D cursor will move in the negative y direction on the screen.

3. Continue holding down the ***right mouse*** button and moving the mouse until the 3D cursor is over the

node at `(0,0,0)`.

4. Release the ***right mouse*** button.

5. Add CP □ **by Picking**

If the point was picked correctly you will notice that the Browser window changes to:

```
Connector Segments: 1
Connector Dimensions: 0
Control Points: 1
Cross Position:
  x = 0.0000000E+00
  y = 0.0000000E+00
  z = 0.0000000E+00
Distance To Last Point:
D = 0.0000000E+00
```

The important information to note is that the segment now has one control point. If Gridgen's Message window displayed an error message saying that a point had not been picked, you should repeat the above steps until the control point is added. Note after the first control point is added, a small open yellow square appears at the control point location. Also, a large open white circle will appear around the 3D cursor. This is Gridgen's way of telling you the current segment, consisting of the picked control point and the current 3D cursor position, has zero length. In order to complete the shape definition of this connector, we need to add a second control point at Point B, Figure 19.3. To add this control point we will do the following:

1. Add CP □ **via Keybrd**

2. `0.75, 0, 0`

After you do this confirm the Browser window shows:

```
Connector Segments: 1
Connector Dimensions: 0
Control Points: 2
Cross Position:
  x = 7.5000000E-01
  y = 0.0000000E+00
  z = 0.0000000E+00
Distance To Last Point:
D = 0.0000000E+00
```

Once you've defined the shape of this segment we'll save it and proceed with assigning a number of grid points to it ().

1. **Done - Save Segment**

2. **ReDimension**

3. From **keybrd**

4. `15`

5. **Done- ReDimension**

Notice the Browser window now reflects the fact a number of grid points has been assigned to the connector.

```
Connector Segments: 1
Connector Dimensions: 15
```

Finally, let us proceed with grid point distribution (<u>Section 9.3</u>).

1. **ReDistribute**

Before assigning clustering values let us take a closer look at the screen. Locate a large white open circle in the Display window. This is the cursor. Note several menu buttons refer to the cursor (they have a large white open circle on them). Also look at the Browser window text.

```
Connector Segments: 1
Connector Dimensions: 15
SubConnector #1/1 info:
 Dimensions: 15
 Spline: Linear on S
 Distribution: TANH
 beg Δs  input: 0.0000E+00
     Δs actual: 5.3571E-02
 end Δs  input: 0.0000E+00
     Δs actual: 5.3571E-02
 -Δs(8) = 5.357143E-02
 +Δs(8) = 5.357143E-02
Cursor Position:
   x = 3.7500000E-01
   y = 0.0000000E+00
   z = 0.0000000E+00
```

The coordinates of the cursor are shown at the bottom of the Browser window. The 3D cursor may be moved along the connector by:

1. Press and hold the *right mouse* button unless you have **LeftPk** selected in the Display menu, in which case you would press and hold the *left mouse* button.

2. Move the mouse east and west to move the 3D cursor forward and backward along the connector's length.

Note as the cursor moves, the coordinates in the Browser window change accordingly. For this connector we want to cluster at both ends so do the following:

1. **Begin** Δ

2. `0.01`

3. **Ending** Δ

4. `0.01`

Notice every connector has a direction associated with it: the direction in which it was drawn. The connector we are currently working with was drawn in the positive x direction. It is this direction the clustering commands refer to with the terms **Begin** Δ and **Ending** Δ. As we will discuss later in this tutorial, the direction of a connector is irrelevant to the Gridgen process. We could have drawn this connector in the negative x direction without affecting our grid or the amount of work we will have to do. Now save this connector:

1. **Done Redistributing**

2. **Done - Save Connectors**

The next connector to be drawn is connector <u>BC</u> in Figure 19.3. First we will reposition the image in the

Display window to the left side. All of connector <u>BC</u> will be visible as we create it. We will pan the image to the left.

1. Press and hold the *left mouse* button.

2. Move the mouse to the west on the mouse pad. The image will pan to the left on the screen.

3. Release the *left mouse* button when the image has moved to the left side of the Display window.

You probably noticed that the cursor changed to an image of four arrows while the *left mouse* button was pressed. This is to remind you that you are panning the image.

Begin drawing the new connector by:

1. **Add Segment**

2. 3D Space **Line**

Notice the 3D cursor (small white cross) appears at the last control point we placed during connector shape definition (Point B in Figure 19.3). Fortunately this also happens to be one of the control points on the connector we are drawing. Proceed with the segment shape definition using:

1. Add CP □ **by Picking**

2. Add CP □ **via Keyboard**

3. `2, 0.5, 0`

4. **Done - Save Segment**

In case this new connector is not completely visible in the Display window, use the panning and zooming controls to reposition the image so that it is. This connector needs to have 17 points. Do the following to assign the dimension and then begin point distribution:

1. **ReDimension**

2. From **keybrd**

3. `17`

4. **Done - ReDimension**

5. **ReDistribute**

Gridgen provides a tool for ensuring the clustering on either side of a break point is the same (since smooth variation of spacing along an edge is usually required by analysis software). Specify the clustering toward the ramp using the following:

1. **Begin.** Δ

Take note of the Message window text

```
Enter the ΔS value at the beginning of
the subconnector.  Use -1 to copy from
the adjacent subconnector, 0 to
unconstrain the end.
Default = 0.10000E-01
```

By entering −1 you'll be able to pick another connector from which the spacing will be copied. This is a

handy feature to ensure you have the same spacing on either side of a break point.

2. `-1`

3. Move the mouse and position the cursor over the pink image of the connector from which you want to copy the spacing.

4. When the cursor is over the connector's image the pink color will change to salmon.

5. Press and release the *right mouse* button.

The "hand with pointing finger" cursor is a reminder that Gridgen is expecting something to be picked. The candidate items are those colored in pink (in this case connectors sharing a common node). The salmon highlight confirms the item under the cursor is pickable. Pressing and releasing the *right mouse* button picks the connector. The image of the grid points in the Display window has now changed to reflect the new clustering. Also note the Browser window:

```
Connector Segments: 1
   Connector Dimensions: 17
   SubConnector #1 info:
    Dimensions: 17
    Spline: Linear on S
    Distribution: TANH
    beg Δs  input: 1.0000E-02
        Δs actual: 1.0000E-02
    end Δs  input: 0.0000E+00
        Δs actual: 3.1690E-01
     Δs(14,15) = 1.85405E-01
   Cursor Position:
      x = 1.375000E+00
      y = 2.500000E-01
      z = 0.000000E+00
```

It now also reflects the fact the Δs specified at the beginning of the connector is `0.01`.

Complete this connector as follows:

1. **Ending.** Δ

2. `0.01`

3. **Done ReDistributing**

4. **Done - Save Connectors**

Connector <u>CF</u> is to be created next. Do the following:

1. **Add Segment**

2. 3D Space **Line**

3. Add CP □ **by Picking**

4. Add CP □ **via Keyboard**

5. `2, 1.5, 0`

6. **Done - Save Segment**

Figure 19.2 indicates connector <u>CF</u> will be an edge of a surface grid opposite connector <u>AG</u>. In a structured grid this means the two connectors will have the same number of points. In our particular grid it is also reasonable to apply the same type of grid point distribution to both connectors. Therefore we can now do the following:

1. **ReDimension**

2. Using the mouse, move the cursor over the connector corresponding to connector <u>AG</u> in Figure 19.3.

3. When the connector is highlighted in salmon, press and release the ***right mouse*** button.

4. **Done-ReDimension**

5. **Copy △ Values**

6. **Done - Save Connectors**

You just told Gridgen to apply the number of grid points on connector <u>AG</u> to connector <u>CF</u>. You also told the code to copy the distribution constraints (specified clustering, distribution function, etc.).

One more connector must be drawn before the tutorial may proceed to something other than connector creation. This is a good time to point out that typically a large percentage of your work in Gridgen will be expended during connector creation; things get a lot less laborious after this. You can probably guess the next connector we will draw is connector <u>FG</u> in Figure 19.3. Let's finish this one quickly by:

1. **Add Segment**

2. 3D Space **Line**

3. Add CP □ **by Picking**

4. Press and hold the ***right mouse*** button.

5. Move the cursor using the mouse until the 3D cursor is positioned over the node corresponding to Point G.

6. Release the ***right mouse*** button.

7. Add CP □ **by Picking**

8. **Done - Save Segment**

9. **ReDimension**

10. Position the cursor over the connector corresponding to connector <u>AB</u>. Press and release the ***right mouse*** button.

11. Position the cursor over the connector corresponding to connector <u>BC</u>. Press and release the ***right mouse*** button.

12. **Done - ReDimension**

13. **Done - Don't Copy Dist**

14. **Done - Save Connectors**

15. **Done Creating Conns**

16. **Done**

You just drew a straight connector between Points F and G, told Gridgen to put as many grid points on connector <u>FG</u> as there are total on connectors <u>AB</u> and <u>BC</u>, and told Gridgen not to copy the distribution of points but to use the default (equal spacing).

You should now see something similar to Figure 19.5.
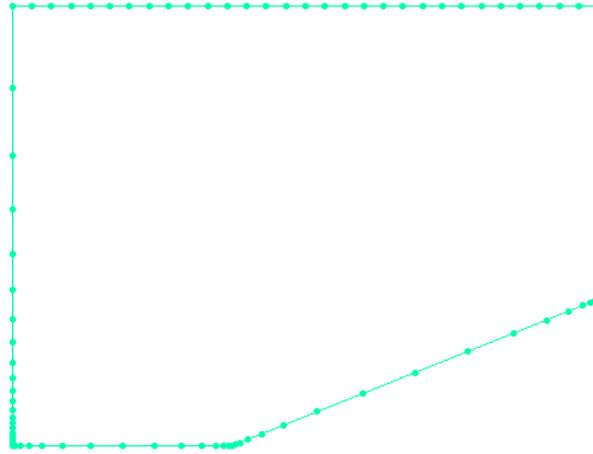
Figure 19.5 All of the Connectors for the Ramp's First Domain

Usually this type of display gets too cluttered so turn off display of connector grid points again.

1. **Disp**

2. Show **Con GPs**

3. **Done**

4.7   Creating A Domain

The connectors you have drawn form the perimeter of one of the surface grids to be used in the blocking system in Figure 19.2 (specifically, we have drawn the perimeter of the ζ_{min} face of the upstream block). Surface grids in Gridgen are known as domains. Since the grid is to be structured, we know each surface grid must map into a four sided region of I x J grid points. That is why it was important for us to ensure the number of grid points on the two vertical connectors was the same. Also, Gridgen requires each edge of a domain be defined by at least one connector so there is a node in each of the corners. This requirement is why we did not create the connector ABCFGA as one connector comprised of six control points: there would not have been nodes in the corners.

The process of creating a domain is rather simple: point to the connectors forming the perimeter of the domain. Begin by doing the following:

1. **Domains**

2. **Create**

3. Make sure the Cell Type **structured** radio button is selected.

4. **Assemble Edges**

Gridgen is now in the mode where it expects you to pick connectors. You can tell this by looking at the Message window. Before proceeding, note the Browser window, shown Figure 19.6.
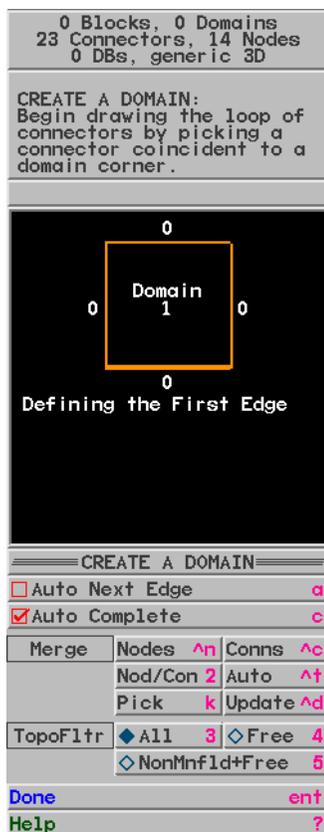
Figure 19.6 The Browser Shows a Schematic of the Ramp's Domain Being Created

This schematic represents the four sided computational space of the domain you create in the Display window. It will help you keep track of which of the four edges you are defining and how many grid points are on each edge. As you add connectors to the domain definition there is also a color coding scheme for the edges. Since the domain is defined by the closed loop of connectors forming its perimeter, you must choose two things before starting: the node at which the perimeter is to begin and end and the direction of the loop. These two choices are irrelevant to the eventual grid index orientation; they are only used in the creation process. For this domain we will start at the node corresponding to Point A in Figure 19.3 and proceed in a counterclockwise direction:

1. Position the cursor over the connector corresponding to connector <u>AB</u>.

2. Press and release the ***right mouse*** button.

3. Move the cursor off the connector.

Take note of the Browser window. It indicates you are "Defining the First Edge" of the domain, and the edge has 15 points. The yellow line in the Browser window (corresponding to edge 1) represents the mapping of the yellow connector in the Display window to physical space. However, this connector is not the only connector that is part of edge 1. To complete the first edge do the following:

1. Position the cursor over the connector corresponding to connector <u>BC</u>.

2. Press and release the ***right mouse*** button.

3. Move the cursor off the connector.

Note the change in the Browser window. The two connectors we have picked make up one of the four edges of this domain. It is time to define the second edge and you must tell Gridgen this by doing the following:

1. **Next Edge**

Note the Browser window now indicates you are "Defining the Second Edge". To continue our counterclockwise loop:

1. Position the cursor over the connector corresponding to connector <u>CF</u>.

2. Press and release the ***right mouse*** button.

3. Move the cursor off the connector.

Notice how the connector is colored red and corresponds to the red edge in the Browser window. This is the only connector in edge 2. To proceed with edge 3 do the following:

1. **Next Edge**

Gridgen knows in a structured grid the number of grid points on opposite edges of the same domain must be the same. Since edge 3 is opposite edge 1 and since we have defined edge 1 to have 31 points the code can complete edge 3 automatically by recognizing connector <u>FG</u> has 31 points and it connects to the appropriate end of edge 2. Likewise, Gridgen recognizes connector <u>AG</u> is the only possible choice for edge 4, so it completes the domain construction and generates the domain interior points using transfinite interpolation. The resulting grid should look like Figure 19.7.
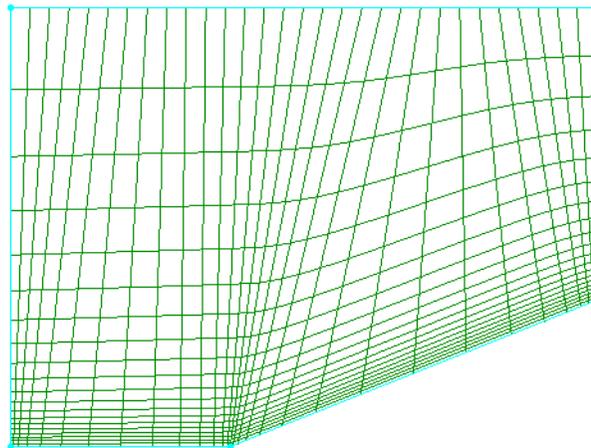
Figure 19.7 Gridgen Automatically Creates the Ramp's Domain Mesh When You Close its Perimeter

By examining the Browser and Display windows you can see we have defined the perimeter of a surface and mapped the surface into a four sided region with 31 x 21 grid points. Furthermore, in the connector creation process we distributed grid points along each of these connectors which provided the input for generating the surface grid.

We recommend you get in the habit of saving your work to a file at regular intervals (in case there's a hardware problem). Let's do that now:

1. **Done**

2. **Done**

3. **Input/Output**

4. Gridgen **Export**

5. Type In... **Name**

6. `ramp`

7. **Done**

You just exported a Gridgen file called ramp.gg (`.gg` is the default extension). The Gridgen file is the main file type used by the code and contains everything from nodes through blocks and display information. Every grid you make will have a Gridgen file. This file can be used to restart the work to continue construction of a grid or to modify an existing grid.

4.8   Running the Elliptic PDE Solver

As mentioned during the domain creation discussion, an algebraic method (specifically, transfinite interpolation) is used to initialize the grid points within a domain. This grid may not be acceptable for your analysis code due to skewness or clustering problems. Because of this Gridgen provides elliptic PDE methods to allow you to make the domain grid smoother, more clustered, or more orthogonal according to your requirements. Gridgen does not judge the quality of your grid. It is up to you to decide whether or not elliptic PDE refinement is necessary. You will apply the elliptic PDE method to the domain you just created.

1. **Domains**

2. Run Solver **Structured**

The first thing you must do is pick the domain or domains to be run in the solver. You can do this by picking from either the Display window (as we have done for connectors) or by picking from the tabular list in the Browser. Let's pick this domain from the Browser.

1. Position the cursor over the entry for domain 1 in the Browser.

2. Note how the domain is highlighted in salmon (indicating that it is pickable).

3. Press and release the *right mouse* button.

4. Note how the domain is highlighted in pink (indicating that it is picked).

5. **Done**

6. Elliptic Slvr **Run**

The elliptic PDE solver is now running using the default attributes. After each iteration the grid is redrawn so that you can monitor the grid quality as the solution proceeds. Notice the Browser window.

```
Ellip PDE Solver - RUN
Iteration #N
Total Residual : nnn
o Max Residual : nnn
```

It indicates how many iterations have been run, the residual of the PDE solution for the entire grid, and the maximum residual. The location of the maximum residual symbol in the Display window indicates the grid point at which the PDE solution is changing most rapidly. We recommend you do not get too hung up on the value or convergence rate of the residual. Instead, by simply watching the grid in the Display window, you will know how the solution is proceeding. It is also important to note Gridgen will not stop the solver. It is up to you to tell the code when to stop the refinement process. Running the elliptic smoother with the default settings should give you a grid that looks like the one in Figure 19.8.
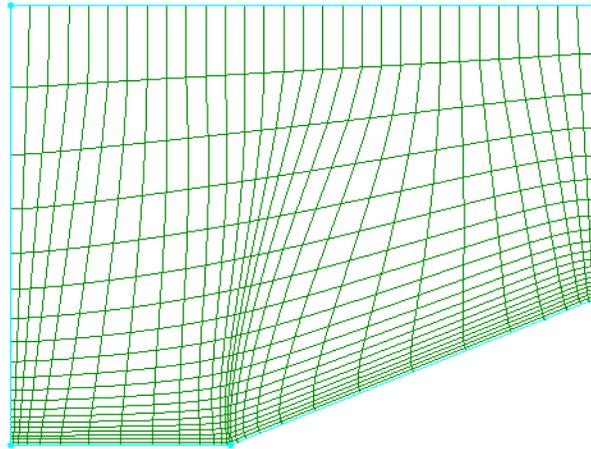
Figure 19.8 Running the Elliptic PDE Solver on the Ramp's Domain Improves the Grid's Quality

When you are ready to stop:

1. **Pause**

Earlier we mentioned that the solver was running with the default attributes. Let us examine the PDE attributes in more detail.

1. **Set Solver Attribs**

The Browser window presents a summary of the PDE attributes for this domain.

```
Domain Num 1 : 31 x 21
TFI: standard
Shape: Free
Relaxation : Optimal
BG ConFun. : Thom.& Midd.
     Δs       |      Angle    |
type decay | type decay | base
-----------------------------
interp   6 | ortho   6  | Hilg
interp   6 | ortho   6  | Hilg
interp   6 | ortho   6  | Hilg
interp   6 | ortho   6  | Hilg
Boundary Conditions :
    Fixed
    Fixed
    Fixed
    Fixed
```

The line containing "`BG ConFun.`" is the background control function. By default Gridgen uses the Thomas-Middlecoff method clustering points on the interior based on how they are clustered on the four edges. The table with the headings "Δs" and "`Angle`" refer to the foreground control functions, influencing the angle of intersection and spacing of transverse grid lines at the four edges. Notice there are four color coded fields in the foreground control function table corresponding to the same colored edges in the Display win-

dow. Also note the line containing "`Relaxation`". Gridgen solves the elliptic PDEs using an explicit SOR algorithm. The Browser window indicates we are using an optimal relaxation parameter by default. Change the relaxation parameter and continue the solver.

1. **Relaxation Parameter**

2. **Use Nominal Relax 0.7**

3. **Done Setting Attributes**

4. Elliptic Slvr **Run**

When ready, stop the solver and save the Gridgen file.

1. **Done - Save**

2. **Done**

3. **Input/Output**

4. Gridgen **Export**

Since the file already exists in the current directory (you saved one before) you can pick the name from the Browser.

1. Position the cursor over the entry for `ramp.gg` in the file Browser.

2. Press and release the *right mouse* button.

3. **Open**

4. **Yes, overwrite file**

5. **Done**

Gridgen makes you confirm your file name choice when exporting to a file already existing (just in case you inadvertently picked the wrong file name).

4.9   More Connectors and Domains

So far you have made one domain corresponding to one face of one of the blocks in the system in Figure 19.2. You can use some of Gridgen's higher level commands to speed-up the remainder of the grid generation process. As you can see from Figure 19.3, the face opposite the one you just made (`hijklmnh`, ζ_{max}) is the same except for the location of the ramp. We will copy the existing domain, translate it in z to the correct location, and then move the ramp point.

1. **Domains**

2. **Copy**

3. Position the cursor over one of the edges of the domain you have already created.

4. Press and release the *right mouse* button.

5. **Done**

6. Position the cursor over Point A.

7. Press and release the *right mouse* button.

8. Set Handle **via Keybrd**

9. `0,0,1`

10. **Done - Translate**

11. **Save Domains**

You just made a copy of the original domain and translated it by moving one of its control points from `(0,0,0)` to `(0,0,1)`. You can confirm that everything is OK by examining the Status window which should look like:

```
0 Blocks, 2 Domains
10 Connectors, 10 Nodes
0 DBs, STAR-CD 3D
```

To verify the scene graphically, first change from an orthonormal (default) to perspective viewing transformation.

1. **Disp**

2. Wndw **Ortnrml** (This is a toggle button that switches between orthonormal and perspective viewing modes each time you press it.)

To further graphically illustrate the two domains rotate the image.

1. Press and hold the *PAD /* key.

2. Release the *PAD /* key after about 45° of rotation.

3. Press and hold the *PAD ** key.

4. Release the *PAD ** key after about 45° of rotation.

Recall we need to somehow change the shape of the new domain to reflect the fact Point I in Figure 19.3 is at a different x value than Point B (to account for the sweep of the ramp). We do this by editing the shape of one of the connectors sharing that node. Edit the connector corresponding to connector HI.

1. **Done**

2. **Done**

3. **Connectors**

4. **Modify**

5. Position the cursor over the connector corresponding to connector HI in Figure 19.3.

6. Press and release the *right mouse* button.

7. **Done**

8. Segment **Edit**

9. Position the cursor over the control point corresponding to Point I in Figure 19.3.

10. Press and release the *right mouse* button.

11. Store CP □ **via Keyboard**

12. `1.25, 0, 1`

13. **Done Editing**

14. **Done - Replace Connectors**

The process to this point has been rather straightforward. You modified a connector by changing the shape

of its segment by moving a control point to a new location. Now when you save the new connector shape you have to tell Gridgen how to treat the new shape. Specifically, the connectors corresponding to connectors HI and IJ in Figure 19.3 share a common point - we have just moved the common point. We need to tell Gridgen whether to unlink the two connectors or to keep them attached by changing the shape of IJ to reflect the change in position of point I. Since these connectors are part of a domain and since the domain must be bounded by a closed loop of connectors we choose to keep the connectors linked.

1. Maintain Linkages

You have now changed the shape of the domain but, as the Display window shows, the grid within the domain needs to be updated. Gridgen will attempt to regenerate a grid that has been run in the elliptic PDE solver by interpolating the existing grid distribution. If the grid had been generated using *only* the algebraic methods then Gridgen would have algebraically regenerated it automatically.

The interpolated grid should be pretty close to what you would get from the elliptic solver, but go ahead and run the elliptic PDE solver as was done on the first domain to make sure. Note when you copied the domain, Gridgen also copied all of the original domain's PDE solver attributes.

1. **Done**

2. **Domains**

3. Run Solver **Structured**

4. Pick domain 2.

5. **Done**

6. Elliptic Slvr **Run**

7. Run solver until you think the grid looks OK.

8. **Done - Save**

You should now have something that looks like Figure 19.9.
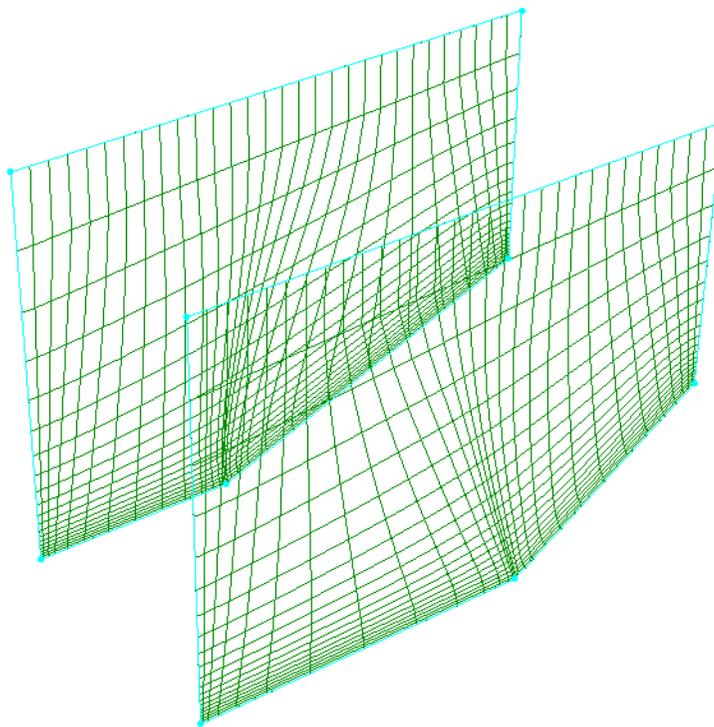
Figure 19.9 Re-apply the Elliptic Solver After Modifying the Ramp's Domain Shape

Save this latest version of the grid to a file.

1. **Done**
2. **Input/Output**
3. Gridgen **Export**
4. Pick file name `ramp.gg`
5. **Open**
6. **Yes, overwrite file**
7. **Done**

Five more domains have to be made before you can create the first block. Before making the domains you will have to draw several z-directed connectors. Start by creating the connector corresponding to connector AH.

1. **Connectors**
2. **Create**
3. **Add Segment**
4. 3D Space **Line**
5. Position the cursor over the node corresponding to Point A in Figure 19.3.

6. Add CP □ **by Picking**

7. Position the cursor over the node corresponding to Point H in Figure 19.3.

8. Add CP □ **by Picking**

9. **Done - Save Segment**

10. **ReDimension**

11. From **keybrd**

12. 11

13. **Done- ReDimension**

14. **Done - Save Connectors**

Instead of redrawing the z-directed connector <u>GN</u> as we did for this last connector, copy connector <u>AH</u> and translate it to the new position. This process is very similar to what you have already done in copying and translating a domain.

1. **Add Segment**

2. Others **Copy**

3. Pick the connector corresponding to connector <u>AH</u> in Figure 19.3.

4. Pick the node corresponding to Point A in Figure 19.3.

5. Press and hold the ***right mouse*** button.

6. Move the cursor over the node corresponding to Point G in Figure 19.3.

7. Release the ***right mouse*** button.

8. Set Handle **by Picking**

9. **Done - Copy Connector**

10. **Copy Break Point Info**

11. **Done - Save Connectors**

Now copy this last connector to positions <u>FM</u> and <u>CJ</u>.

1. **Add Segment**

2. Others **Copy**

3. Pick the connector corresponding to connector <u>GN</u> in Figure 19.3.

4. Pick the node corresponding to Point G in Figure 19.3.

5. Press and hold the ***right mouse*** button.

6. Move the cursor over the node corresponding to Point F in Figure 19.3.

7. Release the ***right mouse*** button.

8. Set Handle **by Picking**

9. **Done - Copy Connector**

10. **Copy Break Point Info**

11. **Done - Save Connectors**

12. **Add Segment**

13. Others **Copy**

14. Pick the connector corresponding to connector <u>FM</u> in Figure 19.3.

15. Pick the node corresponding to Point F in Figure 19.3.

16. Press and hold the ***right mouse*** button.

17. Move the cursor over the node corresponding to Point C in Figure 19.3

18. Release the ***right mouse*** button.

19. Set Handle **by Picking**

20. **Done - Copy Connector**

21. **Copy Break Point Info**

22. **Done - Save Connectors**

The final connector, connector <u>BI</u> corresponding to the leading edge of the ramp, will be drawn using a straight line segment.

1. **Add Segment**

2. 3D Space **Line**

3. Position the cursor over the node corresponding to Point B in Figure 19.3.

4. Add CP □ **by Picking**

5. Position the cursor over the node corresponding to Point I in Figure 19.3.

6. Add CP □ **by Picking**

7. **Done - Save Segment**

8. **ReDimension**

9. From **keybrd**

10. 11

11. **Done- ReDimension**

12. **Done - Save Connectors**

13. **Done Creating Conns**

You should now have all the connectors shown in Figure 19.10.
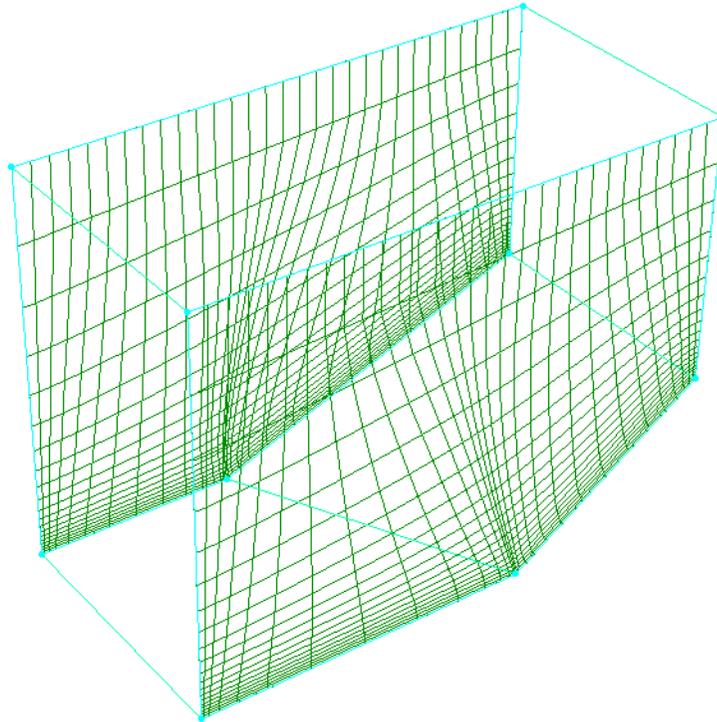
Figure 19.10 Copying and Translating Connectors Speeds the Ramp's Grid Generation Process

Now that all of the connectors relevant to the upstream block have been created, you can continue with the domain creation process. Start with the upstream (ξ_{min}) and far field (ζ_{max}) domains.

1. **Done**
2. **Domains**
3. **Create**
4. **Assemble Edges**
5. **Auto Next Edge**
6. Pick connector <u>AH</u>
7. Pick connector <u>HN</u>. (At this point, Gridgen should automatically create domain ahng).
8. Pick connector <u>GN</u>.
9. Pick connector <u>NM</u>. (This completes domain <u>GNMF</u>).
10. **Done**

The Display window is getting pretty cluttered. It is a good idea to turn off the display of the domain grids to make the new grids easier to see.

1. **Disp**
2. Edit Dspla **Dom**

3. Pick **All**

4. **Done**

5. **Off**

6. **Done - Apply Atts**

7. **Abort**

You should now see only connectors as shown in Figure 19.11.
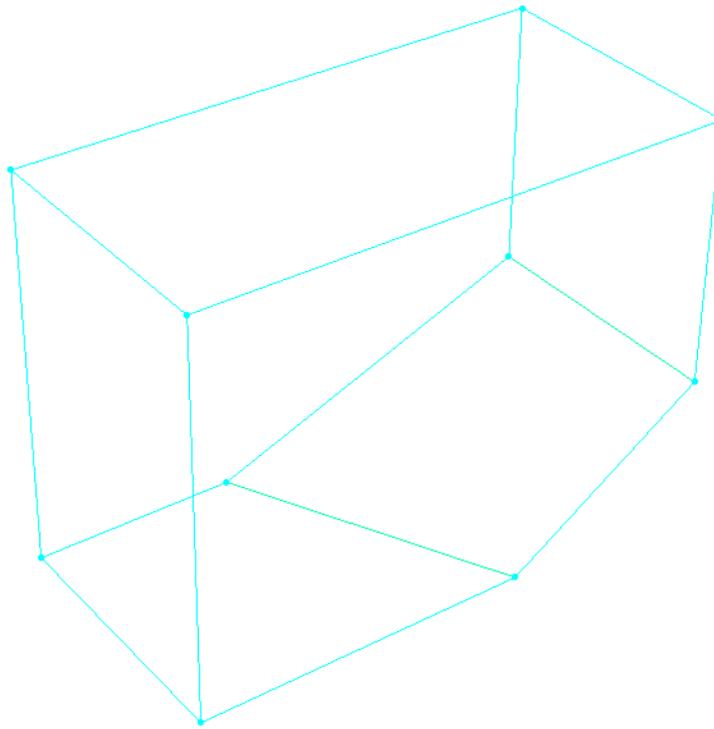
Figure 19.11 Turning Off the Display of Existing Ramp Domains Reduces Screen Clutter

The three domains that still need to be created are the outflow domain and the two domains on the ramp surface.

1. **Done**

2. **Create**

3. **Assemble Edges**

4. Pick connector FM.

5. Pick connector MJ. (This completes the outflow domain.)

6. Pick connector JC.

7. Pick connector CB. (This completes the ramp surface domain.)

8. Pick connector BI.

9. Pick connector <u>IH</u>. (This completes domain <u>BIHA</u>.)

10. **Done**

11. **Done**

All of the domains necessary for defining the perimeter of the upstream block are now created. Before proceeding, save another Gridgen file.

1. **Input/Output**

2. Gridgen **Export**

3. Pick file name `ramp.gg`.

4. **Open**

5. **Yes, overwrite file**

6. **Done**

4.10 Creating a Block

You can now proceed with construction of the highest level element of the grid hierarchy: the block. Block creation proceeds along the same lines as domain creation (albeit volume rather than surface). You pick the domains comprising of each of the six faces of the block. Begin by:

1. **Blocks**

2. **Create**

3. **Assemble Faces**

4. **Add 1st Face**

Gridgen is now waiting for you to pick all the domains that make up one face of this block. You can have any number of domains on a face. Some of the reasons you may have more than one domain on a face are:

• To apply more than one boundary condition type on a face. Each domain can have only a single boundary condition type. By grouping multiple domains on a face, you can have multiple boundary condition types on a face.

• For proper block-to-block connectivity. Each domain must be fully contained in the blocks on either side, i.e. a domain cannot span multiple block faces.

• For more control over grid point distribution. Sometimes a face is broken into smaller pieces to give you more control over the grid.

We are free to start with any face. Let's choose the ζ_{min} face.

1. Position the cursor over connector <u>GF</u>. Press and release the ***right mouse*** button.

2. Move the cursor off the domain.

Notice the Browser window contains a schematic of this face. Since this face consists of only one domain save it and proceed with the second face:

1. **Save the Face**

2. Position the cursor over connector <u>CJ</u>. Press and release the ***right mouse*** button.

3. **Save the Face**

4. Position the cursor over connector <u>AH</u>. Press and release the ***right mouse*** button.

5. **Save the Face**

6. Position the cursor over connector <u>NM</u>. Press and release the ***right mouse*** button.

7. **Save the Face**

8. Position the cursor over connector <u>FG</u>. Notice even though this connector is part of the domain corresponding to the η_{max} face, the domain <u>GFMNG</u> wasn't highlighted. The ambiguity is caused by the fact connector <u>FG</u> is used in several domains and Gridgen guessed which one it should highlight. When there are multiple pickable items at the cursor location (as there are in this case), you can use the ***x*** hot key to scroll through all of the pickable items until the one you want (the top face) is highlighted.

9. Press and release the ***x*** hot key.

10. Press and release the ***right mouse*** button.

11. **Save the Face**

The final face corresponding to the ramp surface is comprised of two domains.

1. Position the cursor over connector <u>AH</u>.

2. Without moving the cursor press and release the ***x*** key.

Repeat this procedure several times. Each time the ***x*** key is pressed Gridgen moves to the next candidate item under the cursor and makes it pickable. Continue this process until domain <u>AHIBA</u> is highlighted.

1. Make sure the correct domain is highlighted.

2. Press and release the ***right mouse*** button.

3. Move the cursor off the connector.

Remember this domain is only part of the face of the block. Before saving the face, we must pick the other domain.

1. Position the cursor over connector <u>IJ</u>.

2. Without moving the cursor press and release the ***x*** key.

3. Make sure the correct domain is highlighted.

4. Press and release the ***right mouse*** button.

5. Move the cursor off the connector

Examine the schematic of this face in the Browser window. It shows how the two domains will be combined into a single face. Figure 19.12 shows how your screen should look.

```
        0 Blocks, 7 Domains
    15 Connectors, 10 Nodes
        0 DB ents, STAR-CD 3D



ASSEMBLE 1ST FACE OF THE BLOCK
Use the cursor to add a domain to the
face or choose from the menu options
below. (use x to cycle picks).
```
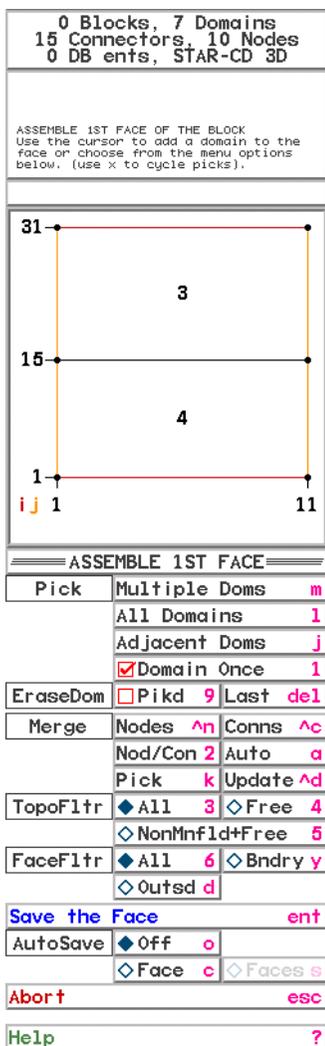
Figure 19.12 The Browser Shows How the Two Domains Form One Face of the Ramp's First Block

1. **Save the Face**

When you save the sixth face, Gridgen assembles all of the domains and places their surface grids on the exterior of a volume grid. Gridgen also chooses a default orientation of the computational axes which are displayed according to a red=ξ, orange=η, yellow=ζ scheme.

Since Gridgen's selection of computational axes may not be what you wanted it lets you manually redefine the axes if you want. In this block we want ξ to run in the x direction, η in y, and ζ in z. Try this now:

1. **ReSpecify** ξ, η, ζ

2. Position the cursor over any of the four block edges that run in the ξ direction.

3. Press and release the ***right mouse*** button.

4. ξ **direction**

5. Position the cursor over any of the four block edges that run in the η direction.

6. Press and release the ***right mouse*** button.

7. η **direction**

8. ζ **direction**

It is hoped you noticed each of the color coded connectors has an arrow pointing in the positive index direction.

Each block must be given a unique name when using the STAR-CD analysis software. Do that now:

1. **ReName**

2. `upstream`

3. **Done - Save Blocks**

Save the current grid to a Gridgen file.

1. **Done**

2. **Input/Output**

3. Gridgen **Export**

4. Pick file name `ramp.gg`.

5. **Open**

6. **Yes, overwrite file**

7. **Done**

Just as was the case for domains, Gridgen automatically initializes the volume grids using an algebraic method. To view the volume grids do the following:

1. **Blocks**

2. **Examine**

3. Pick **All**

4. **Done**

5. **Topo** ξ

Now use the *left arrow* and *right arrow* keys to increment and decrement the ξ index of the plane being displayed, respectively. To view the η constant planes do:

1. **Topo** η

A similar command is available for ζ constant planes.

Now create the second block in this system. In order to keep the display as uncluttered as possible, turn off the display of the domain grid points.

1. **Abort**

2. **Disp**

3. Edit Dspla **Dom**

4. Pick all domains that currently use a wireframe display style.

5. **Done**

6. **Off**

7. **Done - Apply Atts**

8. **Abort**

9. **Done**

10. **Done**

4.11   Creating the Second Block

It is hoped at this point in the tutorial you can see several ways to continue with the construction of the second block. The technique you will use is to copy the downstream domain JMFCJ and translate it in the x direction to the position of KLEDK. One x directed connector will be drawn between Points C & D and copied to positions FE, ML, and JK. Four new domains will be created and the downstream block constructed. Since these steps have all been covered in detail during construction of the first block, the commentary here will be kept to a minimum.

First, copy the domain and move it downstream:

1. **Domains**

2. **Copy**

3. Pick the downstream domain defined by JMFCJ.

4. **Done**

5. Pick the node corresponding to Point C.

6. Set Handle **via Keybrd**

7. 3 (Notice when this last coordinate was entered, only the x coordinate was actually typed and the default values for y and z were used. The default values for all three of the components are the current cursor position shown in the Browser window. Because of this, it is only necessary to enter the value of the component(s) that change.)

8. **Done - Translate**

9. **Save Domains**

10. **Done**

Now create an x directed connector. For practice, we will create this connector in a different way than the previous connectors - as a 2 Point Connector. 2 Point Connectors are created by picking two end points. If both end points lie on a common database entity a DB cnstr **Line** connector is created between the two points. Otherwise, a 3D Space **Line** connector is created between the two points. In this example, we do not have any database entities, so we will use **2 Point Connectors** to create a straight line segment. Before creating the connector, you will set a default number of grid points to place on any new connectors that are created.

1. **Defaults**

2. Con Dim **dimen**

3. 15

4. **Done**

5. **Connectors**

6. **Create**

7. **2 Point Connectors**

8. Add a control point by positioning the 3D cursor and picking the node corresponding to Point C.

9. Add a control point by positioning the 3D cursor and picking the node corresponding to Point D.

10. **Done**

11. **Done Creating Conns**

12. **Set ∆s Vals**

13. Pick the end of connector <u>CD</u> at Point C.

14. **Done**

15. `0.01`

16. **Abort**

Note a new command, **Set ∆s Vals**, was used to redistribute the grid points on connector <u>CD</u>. Using this command you can pick multiple connector ends and set their spacing constraints all at once. In this example, you only want to set the spacing constraints at one end of the connectors. You can pick as many as you want before setting the spacing.

Make three copies of the original x-directed connector.

1. **Create**

2. **Add Segment**

3. Others **Copy**

4. Pick the connector corresponding to connector <u>CD</u> in Figure 19.3.

5. Pick the node corresponding to Point C in Figure 19.3.

6. Move the 3D cursor to the Point F in Figure 19.3.

7. Set Handle **by Picking**

8. **Done - Copy Connector**

9. **Copy Break Point Info**

10. **Done - Save Connectors**

11. **Add Segment**

12. Others **Copy**

13. Pick the connector corresponding to connector <u>FE</u> in Figure 19.3.

14. Pick the node corresponding to point F in Figure 19.3.

15. Move the 3D cursor to Point M in Figure 19.3.

16. Set Handle **by Picking**

17. **Done - Copy Connector**

18. **Copy Break Point Info**

19. **Done - Save Connectors**

20. **Add Segment**

21. Others **Copy**

22. Pick the connector corresponding to connector <u>ML</u> in Figure 19.3.

23. Pick the node corresponding to Point M in Figure 19.3.

24. Move the 3D cursor to Point J in Figure 19.3.

25. Set Handle **by Picking**

26. **Done - Copy Connector**

27. **Copy Break Point Info**

28. **Done - Save Connectors**

29. **Done Creating Conns**

30. **Done**

Create four new domains.

1. **Domains**

2. **Create**

3. **Assemble Edges**

4. If **Auto Next Edge** and **Auto Complete** are not enabled, then enable them now.

5. Pick connector <u>CD</u>.

6. Pick connector <u>DE</u>. (You should see domain <u>CDEFC</u> appear now.)

7. Pick connector <u>CD</u>.

8. Pick connector <u>DK</u>. (You should see domain <u>CDKJC</u> appear now.)

9. Pick connector <u>JK</u>.

10. Pick connector <u>KL</u>. (You should see domain <u>JKLMJ</u> appear now.)

11. Pick connector <u>LM</u>.

12. Pick connector <u>MF</u>.   (You should see the domain <u>LMFEL</u> appear now.)

13. **Done**

14. **Done**

Create the new block.

1. **Blocks**

2. **Create**

3. **Assemble Faces**

4. **Add 1st Face**

5. Turn on the AutoSave **Face** toggle.

6. Pick domain <u>CDEFC</u>.

7. Pick domain <u>DKLED</u>.

8. Pick domain <u>DKJCD</u>.

9. Pick domain <u>CJMFC</u>.

10. Pick domain <u>JKLMJ</u>.

11. Pick domain <u>ELMFE</u>.

12. **ReName**

13. `downstream`

14. **Done - Save Blocks**

15. **Modify**

16. Pick **All**

17. **Done**

18. **Align** ξ, η, ζ

19. Pick the upstream block with the right mouse button.

20. **Done - Replace Blocks**

21. **Done**

Save the two block system to a file.

1. **Input/Output**

2. Gridgen **Export**

3. Pick file name `ramp.gg`.

4. **Open**

5. **Yes, overwrite file**

6. **Done**

4.12   Running the Elliptic PDE Solver on Volume Grids

At this point, you have created two blocks with the volume grids initialized using an algebraic method. As was the case with the surface grids, we will apply an elliptic PDE method to the volume grids to improve the grid's smoothness, clustering, and orthogonality.

1. **Blocks**

2. Run Solver **Structured**

3. Pick **All**

4. **Done**

5. Elliptic Slvr **Run**

6. Let the elliptic solver run about 25 iterations.

7. **Pause**

8. **Done - Save**

The **Examine** function in the solver allows you to view a topological grid surface while the elliptical solver is running.

4.13   Volume Grid Inspection

Inspect the results of Gridgen 3D's elliptic PDE refinement of the two block volume grid.

1. **Examine**
2. Pick **All**
3. **Done**
4. **Topo** ξ
5. Use the *left arrow* and *right arrow* keys to scroll through the grid planes.

Figure 19.13 shows the ξ=9 grid surface after elliptic smoothing. The volume grid appears to be clustered and orthogonal where we wanted it. There is additional diagnostic information printed in the Browser window. At the top it shows global information about the grid function we are currently examining. The bottom section of information shows the location and grid spacing at the point currently highlighted. Note turning the Hiliter **extend** toggle on allows you to scroll through the grid planes for both blocks and thus view the grid surface across the chosen topological plane.
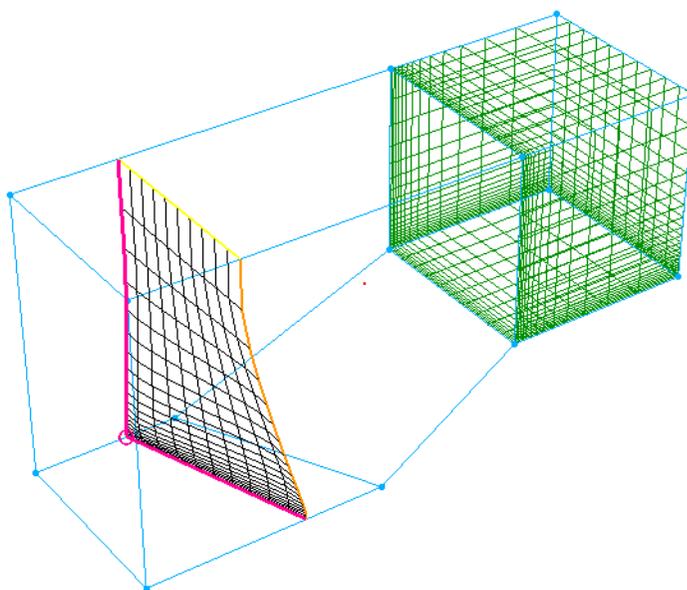
Figure 19.13 Block Examine Lets You Assess the Ramp's Volume Grid Quality

The volume grid looks good. Let's go to the final part of the tutorial, generating the input files for a CFD flow solver.

1. **Abort**
2. **Done**

4.14   *Customizing the Grid for the Analysis Software*

The Gridgen process is nearly complete. The volume grid is done and ready to be used. The final task is to set-up the grid and boundary condition information for the analysis software. If you check the beginning of this tutorial we told Gridgen we would be using the STAR-CD flow solver to compute the flow field on this grid. The Status window should also show this. We need to set a STAR-CD flow solver boundary condition on each domain in the system and then export STAR-CD's files. To begin setting boundary conditions do the

following.

1. **Analysis S/W**

2. **Set BCs**

The Browser has an entry for each domain on the block is shown in Figure 19.13.

Figure 19.14 Domains in the Browser for BC Selection

Each domain entry in the Browser lists the block coordinate that is constant on the domain, the domain number in parentheses, and the boundary condition set on that domain. Notice only two domains have an entry for a boundary condition. Let us examine it further.

1. Position the cursor over the Browser entry containing the text "Type 1 - (2)".

Notice in the Display window the domain corresponding to this entry is highlighted. It is the domain shared between the two blocks. Gridgen is telling you it has detected a point-to-point (Type 1) inter-block connection on this domain and has automatically set the boundary condition. In general, Gridgen will automatically set all inter-block connections. All you have to do is specify the boundary conditions corresponding to flow boundaries.

1. Position the cursor over the Browser entry containing the text "1, ζmin".

2. Press and release the *right mouse* button.

3. Position the cursor over the Browser entry containing the text "2, ζmin".

4. Press and release the *right mouse* button.

5. Position the cursor over the Browser entry containing the text "1, ζmax".

6. Press and release the *right mouse* button.

7. Position the cursor over the Browser entry containing the text "2, ζmax".

8. Press and release the *right mouse* button.

9. **Done**

You are now presented with a menu of boundary conditions used by the STAR-CD flow solver. These domains are xy planes of symmetry so choose

1. **Symmetry**

Notice how the boundary condition you just set is now reflected in the Browser. To set the rest of the boundary conditions.

1. Pick the three Browser entries that contain the text "ηmin".

2. **Done**

3. **Wall**

4. Pick the two Browser entries that contain the text "ηmax".

5. **Done**

6. **Free Stream**

7. Pick the Browser entry that contain the text "1, ξmin".

8. **Done**

9. **Inlet**

10. Pick the Browser entry containing the text "2, ξmax".

11. **Done**

12. **Outlet**

13. **Done**

14. **Done**

Now you can save another Gridgen file.

1. **Input/Output**

2. Gridgen **Export**

3. Pick file name ramp.gg

4. **Open**

5. **Yes, overwrite file**

6. **Done**

Finally, export the files needed to run the STAR-CD code:

1. **Analysis S/W**

2. **Export Analysis Data**

3. Type In... **Name**

4. ramp

5. **Done**

6. **Quit Gridgen**

7. **Quit Gridgen**

At last you are ready to run the STAR-CD code using the files ramp.inp, ramp.bnd, ramp.cel and ramp.vrt. The grid including nodes, connectors, domains, blocks, surface and volume grids, display settings, and analysis software boundary conditions is saved in file ramp.gg.

5. Intersecting Pipes: Pole Grid Construction (Bottom Up)

5.1   Introduction

This tutorial will introduce you to several special Gridgen capabilities while you build a grid for two pipes intersecting in a Y-junction.

5.2   Topics Covered

Gridgen skills you will learn during this tutorial session are:

- using the **Points**, Curve, and Surface database creation tools

- finding intersections between database entities using **Intersect**

- creating pole connectors, pole domains, and blocks containing poles

5.3   Background

You should complete the tutorials in Section 2 and Section 4 before beginning this lesson, since this tutorial assumes you already have the skills covered in those sections.

5.4   Geometry

The intersecting pipe configuration you will use for this tutorial is shown in Figure 20.1.

Figure 20.1 Geometry for Intersecting Pipes Tutorial

The grid you will generate consists of two blocks. One block fills the main pipe. The other block is in the branch section. The block topology you will build is shown in Figure 20.2. Within each block one family of grid lines will run in the flow direction; however, the cross section grid topology will be different for the two blocks.

In the main pipe, an O-topology is used with a pole running down the center of the pipe. This topology makes efficient use of the cells, allows you to cluster tightly in the pipe wall boundary layer, and keeps grid lines normal to the walls.

In the branch section, an H-topology is used. This is not as efficient as the O-topology, but it makes interfacing with the main pipe grid easier.

Figure 20.2 Two Block Grid Topology for the Intersecting Pipes

Now that a topology for the blocking system has been chosen, you can begin creating the grid. For convenience we have assigned a naming convention to important points in the grid. The points' labels and coordinate values are shown in the following table.

Table of Intersecting Pipes Tutorial Coordinates

Label	X	Y	Z
A	-10	0	0
B	0	0	0
C	10	0	0
D	-5	10	0

5.5   Getting Started

Begin by starting Gridgen, either by typing the string `gridgen` on Unix workstations or simply double-clicking the Gridgen shortcut on your PC's desktop. Usually, the first step in running Gridgen is to select the software (Section 17.1) to be used in the analysis. This will ensure that the grid generated by Gridgen is consistent with the capabilities of the analysis software. Since setting the analysis software is not the focus of this tutorial, you can use the default generic analysis software setting. No action on your part is necessary.

5.6   Creating a Database

You will begin creating the pipes by entering the points that define important locations on the geometry. Then you will build curves followed by surfaces. The final step in geometry creation will be to find the intersection curve between the two pipes.

From the Gridgen MAIN MENU:

1. **Database**

2. **Create**

3. **Points**

You should now be in the CREATE POINTS menu shown below.

```
══════CREATE POINTS══════
Store Pnt │at +          padEnt
          │by Picking    pad0
          │via Keybrd    pad.
          │Δxyz via Keybrdo
          │Δcp via Keybrd^o
          │by Linear Proj (
          │at Clsest DB Pt)
Erase Last Point          del
Sensit│Decr ←│Reset↓│Incr →
Move + to Screen Center     c
Rotation Pt. To +           t
☑Keep Pts  1│☐Keep Avg   3
Save                      ent

Abort                     esc

Help                        ?
```

Make sure that the **Keep Pts** toggle button is enabled and the **Keep Avg** toggle button is disabled. Create points A, B, C, and D now while referring to Figure 20.3 for the proper layout of the points:
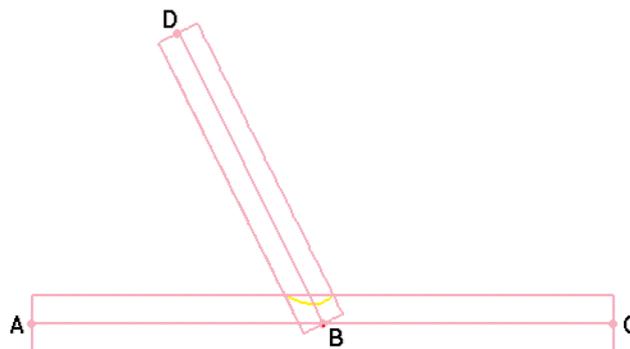
Figure 20.3 Labels for Starting Points

1. Store Pnt **via Keybrd**

2. -10, 0, 0

3. Store Pnt **via Keybrd**

4. 0, 0, 0

5. Store Pnt **via Keybrd**

6. `10, 0, 0`

7. Store Pnt **via Keybrd**

8. `-5, 10, 0`

9. **Save**

```
═══════LINE DB ENTITY═══════
Add CP □   at          padEnt
           by Picking   pad0
           via Keybrd   pad.
           Δxyz via Keybrd□
           by Linear Proj (
           at Clsest DB Pt)
Erase Last Point         del
Edit Existing Curve        q
Cnvrt to│Curve  ^c│Akima  ^h
Smooth Control Points      s
Movement Restrictions      m
Sensit│Decr ←│Reset↓│Incr →
Move    to Screen Center   c
Move Rotation Pt. To       t
Save DB curve            ent

Abort - Don't Save       esc
Display Commands          ^a
Help                       ?
```

Next, create the lines that define the center line of each pipe. Starting from the DB CREATION menu:

1. Curve **Line**

2. Position the 3D cursor over point A.

3. Add CP □ **by Picking**

4. Position the 3D cursor over point C.

5. Add CP □ **by Picking**

6. **Save DB curve**

7. Curve **Line**

8. Position the 3D cursor over point B.

9. Add CP □ **by Picking**

10. Position the 3D cursor over point D.

11. Add CP □ **by Picking**

12. **Save DB curve**

```
═══CREATE OFFSET CURVE═══
Store Pnt │at            padEnt
          │      by Picking  pad0
          │      via Keybrd  pad.
          │      Δxyz via Keybrdo
          │      Δcp via Keybrd^o
          │      by Linear Proj (
          │      at Clsest DB Pt)
Sensit│Decr ←│Reset↓│Incr →
Move    to Screen Center    c
Rotation Pt. To             ↑
ReDefine Generatrix       del
Set Fit Tolerance           2
Reverse the Offset          s
Enter Offset Distance       d
Save                      ent

Abort                     esc

Help                        ?
```

Next you will create an edge curve for each pipe by creating offset curves from the two existing curves. The edge curves are offset from the center line by the pipe radius. It is important to remember when using the **Offset** command that the offset operation is performed in the plane of the screen. Therefore it is essential that your view orientation be perpendicular to the plane in which you wish the offset operation to take place. The radius of the main pipe is 1.0, and the radius of the branch section is 0.75.

1. Curve **Offset**

2. Position the cursor over line AC and press and release the ***right mouse*** button (assuming you are using the default mouse button configuration. Otherwise, swap "***right mouse***" for "***left mouse***" in the remainder of this tutorial).

3. **Enter Offset Distance**

4. `1.0`

5. **Save**

6. Curve **Offset**

7. Position the cursor over line BD and press and release the ***right mouse*** button.

8. **Enter Offset Distance**

9. `0.75`

10. **Reverse the Offset**

11. **Save**
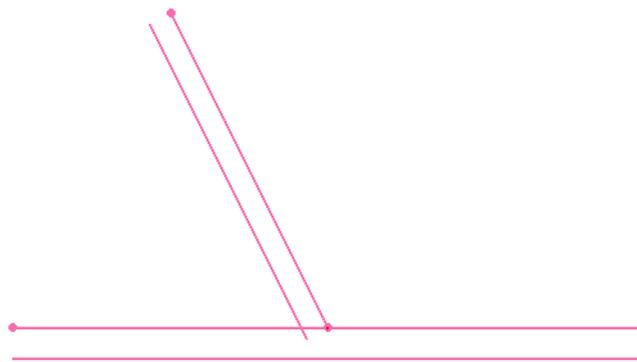
Figure 20.4 Database Curves for the Intersecting Pipes

You should be at the DB CREATION menu, and the Display window should look like Figure 20.4. You are now ready to create the surfaces of the two pipes.

You will construct the surfaces of the two pipes as surfaces of revolution. Surfaces of revolution are created by specifying two curves: a generatrix and an axis. The generatrix is revolved around the axis to produce the surface. By default, Gridgen revolves the generatrix a full 360 degrees around the axis. For this tutorial, you will only use 180 degrees of rotation.

```
=CREATE SURF OF REVOLUTION=
Reselect Axis Curve        del
Reselect Both Curves         0
(Re)Set Fit Tolerance        2
(Re)Set the Angle            a
Done-Save Surf of Revol    ent

Abort-Don't Save Surface   esc

Help                         ?
```

Starting from the DB CREATION menu:

1. Surface **Revolution**

2. Position the cursor over the edge curve of the main pipe (the curve you offset from AC) and press and release the ***right mouse*** button.

3. Position the cursor over line AC and press and release the ***right mouse*** button.

4. **(Re)Set the Angle**

5. 180

6. **Done - Save Surf of Revol**

7. Surface **Revolution**

8. Position the cursor over the edge curve of the branch pipe (the curve you offset from BD) and press and release the ***right mouse*** button.

9. Position the cursor over line BD and press and release the ***right mouse*** button.

10. **(Re)Set the Angle**

11. -180

12. **Done - Save Surf of Revol**

13. **Done**

The final piece of the database is the intersection curve between the two branches. When finding intersections, Gridgen prompts you to pick two groups of database entities. It computes the intersections of entities in the first group with entities in the second group. For this tutorial, the first group contains only the surface of the main pipe, and the second group contains only the surface of the branch section.

Starting from the DATABASE COMMANDS menu:

1. **Intersect**

2. **Select Group A**

3. Select the surface of the main pipe from the screen or from the Browser list.

4. **Done**

5. **Select Group B**

6. Select the surface of the branch section.

7. **Done**

8. **Run**

9. **Done**

Gridgen computes the intersection curves when you hit the **Run** button. After some image manipulations (Section 2.7), you should be able to see the intersection curve in the Display window as shown in Figure 20.5.
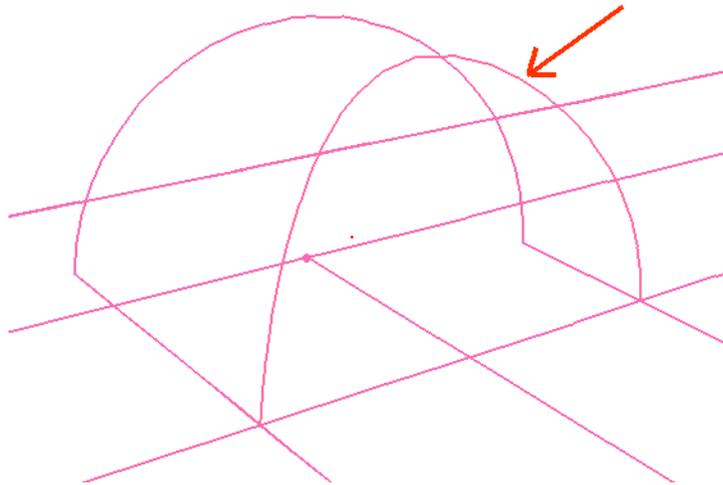
Figure 20.5 Intersection Curve Computed by Gridgen

This completes the geometry creation portion of the tutorial. It would be a good idea for you to save the database as a Gridgen Composite format database file before proceeding further.

Starting from the MAIN MENU:

1. **Input/Output**
2. Database **Export**
3. **composite**
4. **Done**
5. Type In... **Name**
6. `pipes`

Note that Gridgen will automatically append the correct file extension (.dba).

5.7   Drawing Connectors

The geometry you just created will be used to determine the shape of the grid you will create next. Before starting to build the grid, set a default number of grid points to put on each new connector we create.

Starting from the MAIN MENU:

1. **Defaults**
2. Con Dim **dimen**
3. `31`
4. **Done**

The connector topology you will create is shown in exaggerated scale in Figure 20.6 along with labels that will be used for reference in the discussion that follows. This figure will be repeated every few pages for convenient reference.
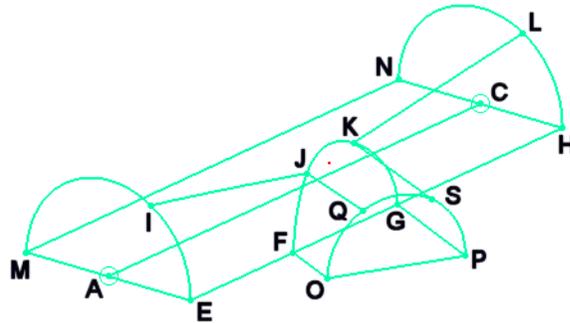
Figure 20.6 Connector Topology

Begin by creating some connectors on the surfaces of the two pipes, the center line of the main pipe, and the intersection curve. From the MAIN MENU:

1. **Connectors**

2. **Create**

3. **On DB Entities**

4. Select the two surfaces of revolution, the axis of the main pipe, and the intersection curve between the two pipes from the Browser or Display window.

5. **Done**
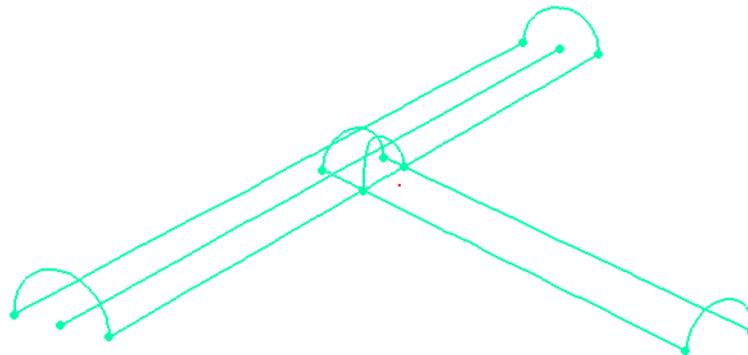
Figure 20.7 Connectors Created With On DB Entities Command

The connectors you have just created are displayed in Figure 20.7.

You can see that the **On DB Entities** command puts connectors along the length of the selected database curves and around the perimeter of the selected database surfaces. It also checks to see if any of the connectors are duplicates of other connectors or if they intersect. Any duplicates are deleted and the remaining connectors are split at the intersection locations. For example, connector EH is automatically split into connectors EF, FG, and GH.

The connectors on the end of the branch pipe that is inside the main pipe will not be part of the final grid, so we can delete them.

Starting from the CONNECTOR COMMANDS menu:

1. **Delete**

2. Select the connectors on the end of the branch pipe inside the main pipe using the cursor and *right mouse* button.

3. **Done**

4. **Done**

We also need to redimension connectors <u>EF</u>, <u>FG</u>, and <u>GH</u> because they need a different number of points than the default setting.

1. **ReDimension Grids**

2. Select connector <u>EF</u> using the *right mouse* button.

3. `11`

4. Select connector <u>FG</u> using the *right mouse* button.

5. `11`

6. Select connector <u>GH</u> using the *right mouse* button.

7. `11`

8. **Done - Apply New Dims**

To fit the topology we chose for this grid, we also need to split the connectors at the ends of the pipes and on the intersection curve.

1. **Modify**

2. Select connector <u>EM</u>.

3. **Done**

4. **Split**

5. ❍ **to GP** ● **via Keybd**

6. `21`

7. **Split at Cursor** ❍

8. **Abort - Don't Split**

9. **Done-Replace Connectors**

10. **Modify**

11. Select connector <u>HN</u>.

12. **Done**

13. **Split**

14. ❍ **to GP** ● **via Keybd**

15. `21`

16. **Split at Cursor** ❍

17. **Abort - Don't Split**

18.   **Done-Replace Connectors**

The ends of the branch pipe will be split into 3 pieces.

1.   **Modify**

2.   Select connector <u>GKJF</u> using the *right mouse* button.

3.   **Done**

4.   **Split**

5.   ○ **to GP ● via Keybd**

6.   11

7.   **Split at Cursor** ○

8.   ○ **to GP ● via Keybd**

9.   11

10. **Split at Cursor** ○

11. **Abort - Don't Split**

12. **Done - Replace Connectors**

13. **Modify**

14. Select connector <u>OQSP</u> using the *right mouse* button.

15. **Done**

16. **Split**

17. ○ **to GP ● via Keybd**

18. 11

19. **Split at Cursor** ○

20. ○ **to GP ● via Keybd**

21. 11

22. **Split at Cursor** ○

23. **Abort - Don't Split**

24. **Done - Replace Connectors**
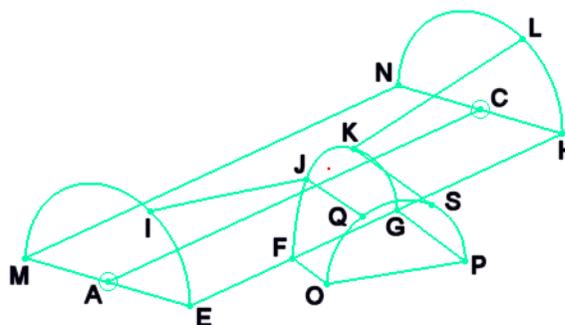
Now it is time to create a few more connectors.

1.   **Create**

2.   **2 Point Connectors**

3.   Hold down the *right mouse* button and move the 3D cursor over point J.

4.   Add CP □ **by Picking**

5.   Hold down the *right mouse* button and move the 3D cursor over point Q.

6.   Add CP □ **by Picking**

7.   Hold down the *right mouse* button and move the 3D cursor over point S.

8.   Add CP □ **by Picking**

9.   Hold down the *right mouse* button and move the 3D cursor over point K.

10. Add CP □ **by Picking**

The next two connectors you create will be pole connectors. A pole connector is a connector that is collapsed to a point. There are two pole connectors in this grid, at points A and C. You create them as a connector with two control points at the same location. You should still be in the 2 POINT LINE menu.

1.   Hold down the *right mouse* button and move the 3D cursor over point A.

2.   Add CP □ **by Picking**

3.   Add CP □ **by Picking**

4.   Hold down the *right mouse* button and move the 3D cursor over point C.

5.   Add CP □ **by Picking**

6.   Add CP □ **by Picking**

7.   **Done**

8.   **Done Creating Conns**

Now you should have created all the connectors shown in Figure 20.8. Notice the special symbology Gridgen uses for a pole connector - a dot with a circle around it.
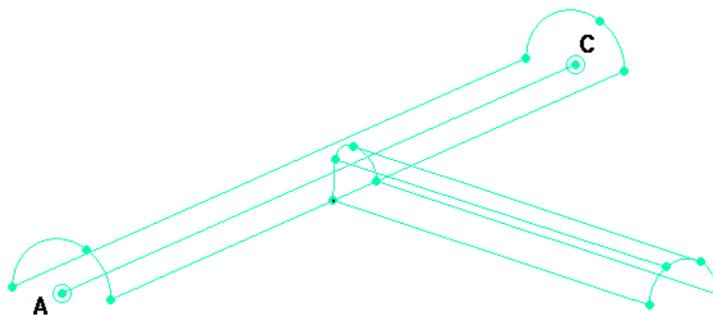
Figure 20.8 Pole Connectors for the Two Pipes Are Located at A and C

All the connectors we have created so far have had 31 grid points assigned to them initially. We still have a few more connectors to make, but the remainder will have 11 grid points on them. Go to the SET DEFAULT VALUES menu and set that up.

1.   **Done**

2.   **Defaults**

3.   Con Dim **dimen**

4.   `11`

5.   **Done**

6. **Connectors**

You are ready to put the remaining connectors on the main pipe. They will all be **2 Point Connectors**. Rather than tell you step-by-step how to create them, we are going to assume you got the hang of it while creating the previous **2 Point Connectors** and can try it on your own. So, here goes. Create **2 Point Connectors** for the point pairs AE, AM, CN, CH, IJ, and KL.

Since connectors IJ, KL, JQ, and KS do not exactly traverse parallel to their respective pipes, it will be prudent to project them all to the underlying database surfaces to ensure they match the database shape. This will also allow Gridgen to automatically surface constrain the domains on both pipes later.

From the CONNECTOR COMMANDS menu:

1. **Modify**

2. Select connectors IJ, KL, JQ, and KS from the Display window or from the Browser.

3. **Done**

4. **Project**

5. **Closest Pt**

6. **Done - Project**

7. **Done - Replace Connectors**

8. **Maintain Linkages**

Only one more connector to create! The final connector is on the plane of symmetry of the smaller pipe between points O and P. You could create it using **2 Point Connectors**, but Gridgen will try to put it on any adjacent database entities in that case and all we want is a straight line. (Actually, Gridgen will create the straight line connector we want if all the available on-database paths between the selected points are already filled. It is easier in this case to create it explicitly as a straight line.)

From the CONNECTOR COMMANDS menu:

1. **Create**

2. **Add Segment**

3. 3D Space **Line**

4. Position the 3D cursor over point O.

5. Add CP □ **by Picking**

6. Position the 3D cursor over point P.

7. Add CP □ **by Picking**

8. **Done - Save Segment**

9. **Done - Save Connectors**

10. **Done Creating Conns**
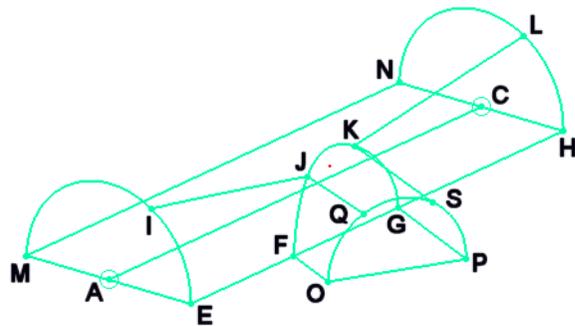
5.8 Creating Domains

Go to the DOMAIN COMMANDS menu to begin creating domains. The first domain you create will be a pole domain. A pole domain is a domain that is collapsed into a line. It consists of two pole connectors and a straight line connector. The straight line connector is used twice in the definition of the pole domain. In

this case, the pole domain is the center line of the main pipe.

Starting from the DOMAIN COMMANDS menu:

1. **Create**

2. Cell Type **structured**

3. **Assemble Edges**

4. **Auto Next Edge**

5. Position the cursor over connector AC and, when it highlights, press the *right mouse* button.

6. Position the cursor over point A.

7. Hit the *x* key until the pole connector AA is highlighted.

8. Press the *right mouse* button.

After you select the second connector, Gridgen will automatically create the pole domain. This is thanks to the **Auto Complete** feature which always tries to complete domains automatically once the first two edges are defined. You will not be able to see the new domain, since it has no area; however, it will be rendered in a light blue color, indicating that it is indeed a domain.

Next, create two of the domains on the main pipe. You should still be at the CREATE A DOMAIN menu:

1. Position the cursor over connector LN and, when it highlights, press the *right mouse* button.

2. Position the cursor over connector NM and, when it highlights, press the *right mouse* button.

3. Position the cursor over connector MI and, when it highlights, press the *right mouse* button.

4. Position the cursor over connector IJ and, when it highlights, press the *right mouse* button. Gridgen will then recognize that connectors JK and KL are needed to complete this domain and will add them.

5. To start the next domain, position the cursor over connector HC and, when it highlights, press the *right mouse* button.

6. Position the cursor over connector CA and, when it highlights, press the *right mouse* button.

7. Position the cursor over connector AE and, when it highlights, press the *right mouse* button.

8. Position the cursor over connector EF and, when it highlights, press the *right mouse* button. Gridgen will then recognize that connectors FG and GH are needed to complete this domain and will add them.

Note that we left **Auto Next Edge** turned on while creating these domains, even though each has an edge with three connectors on it. You can do this as long as the first two domain edges you pick each have only one connector on them. After that Gridgen knows enough about the domain to keep you out of trouble.

The next most interesting domains are the end caps of the main pipe. They each have a pole connector on one edge. While still in CREATE A DOMAIN:

1. Position the cursor over connector AM and, when it highlights, press the *right mouse* button.

2. Position the cursor over point A and hit the *x* key until the pole connector AA highlights.

3. Press the *right mouse* button.

4. Position the cursor over connector AE and, when it highlights, press the *right mouse* button. Gridgen will then recognize that connectors EI and IM are needed to complete this domain and will add them.

5. Position the cursor over connector CN and, when it highlights, press the *right mouse* button.

6. Position the cursor over point C and hit the *x* key until the pole connector CC highlights.

7. Press the *right mouse* button.

8. Position the cursor over connector <u>CH</u> and, when it highlights, press the *right mouse* button. Gridgen will then recognize that connectors <u>HL</u> and <u>LN</u> are needed to complete this domain and will add them.

The remaining domains are all pretty easy. They each have only one connector per edge. Hopefully, you can tell which domains are left by looking at the Display window. Just in case you need help, here are the remaining domains: <u>AMNC</u>, <u>EFJI</u>, <u>FGKJ</u>, <u>GHLK</u>, <u>FGPO</u>, <u>GKSP</u>, <u>JKSQ</u>, <u>FJQO</u>, and <u>OPSQ</u>. Create them now.

Note that some of your new domains are rendered in purple and some in green. Purple domains have been initialized using either Parametric or Parametric Fit TFI, and green domains have been initialized using Gridgen's Standard TFI method. Parametric domains automatically take the shape of the underlying database surface. Standard domains do not. Various criteria must be met for a domain to be initialized using a parametric method. See Section 10.1 of the User Manual for further details. Domain color is only relevant at the time of initialization. The color will not be changed automatically to reflect later modifications to a domain.

5.9   Creating Blocks

Finally, we are ready to create blocks. We will create the main pipe block first. Remember that it uses an O-type topology with a pole at the center line of the pipe. Go to the BLOCK COMMANDS menu and:

1. **Create**

2. Cell Type **structured**

3. **Assemble Faces**

4. **Add 1st Face**

5. AutoSave **Face**

6. Position the cursor over the pole domain (along line AC) and, when it highlights, press the *right mouse* button.

7. Leave the cursor over the pole domain and hit the *x* key until domain <u>AEFGHC</u> highlights, then press the *right mouse* button.

8. Leave the cursor over the pole domain and hit the *x* key until domain <u>AMNC</u> highlights, then press the *right mouse* button.

9. Position the cursor over connector <u>EI</u> and, when domain <u>AEIM</u> highlights, press the *right mouse* button.

10. Position the cursor over connector <u>HL</u> and, when domain <u>CHLN</u> highlights, press the *right mouse* button.

11. Pick **Multiple Doms**

12. Using the cursor and *right mouse* button, pick the four domains on the outer surface of the main pipe.

13. **Done**

14. At this point you will receive two messages regarding possible topologies and whether or not the block is right-handed. This is normal; simply hit the space bar for each and continue.

15. **Done - Save Blocks**

The key points to remember when creating grids with poles are:

• A pole connector is a zero length connector with grid points on it. You create it as a line connector with both end points at the same location.

• Pole connectors are shown in the Display window as dots with circles around them.

- A pole domain contains two pole connectors and one other connector. The line connector is used twice in the domain, either as edges 1 and 3 or as edges 2 and 4.

- The pole domain is used as one of the faces in the block that contains the pole. The connectors in the pole domain must be shared with the neighboring faces in the block.

If you run into trouble when creating pole grids, it is usually caused by having duplicate connectors, i.e. two connectors that lie on top of one another. You can search for and eliminate these types of coincident connector occurrences using Gridgen's Merge utilities (Section 14.10).

Although the grid is not complete (there is one more block to build), this is the end of the tutorial. The construction of the second block uses skills you have learned in this and previous Gridgen tutorials. If you would like to complete this grid for practice please do.

6. Intersecting Pipes: Pole Grid Construction (Top Down)

6.1 Introduction

This tutorial will introduce you to several special Gridgen capabilities while you build a grid for two pipes intersecting in a Y-junction using a top down approach.

The original tutorial was provided by Mr. John Stone, CFD Technologies, Sheffield, UK.

6.2 Topics Covered

Gridgen skills you will learn during this tutorial session are:

• using **Glyph** commands to execute an existing script

• matching block topology to geometry using **Block Modify** and **Connector Modify**

• Using **Domain Modify** to project domains onto database geometry

6.3 Background

You should complete the tutorials in Section 2, Section 4, and Section 5 before beginning this lesson, since this tutorial assumes you already have the skills covered in those sections.

6.4 Geometry

The intersecting pipe configuration you will use for this tutorial is shown in Figure 20.1.

Figure 20.1 Geometry for Intersecting Pipes Tutorial

The grid you will generate consists of two blocks. One block fills the main pipe. The other block is in the branch section. The block topology you will build is shown in Figure 20.2. Within each block one family of grid lines will run in the flow direction; however, the cross section grid topology will be different for the two

blocks.

In the main pipe, an O-topology is used with a pole running down the center of the pipe. This topology makes efficient use of the cells, allows you to cluster tightly in the pipe wall boundary layer, and keeps grid lines normal to the walls.

In the branch section, an H-topology is used. This is not as efficient as the O-topology, but it makes interfacing with the main pipe grid easier.

Figure 20.2 Intersecting Pipes Grid Topology

6.5   Getting Started

Begin by starting Gridgen, either by typing the string `gridgen` on Unix workstations or simply double-clicking the Gridgen shortcut on your PC's desktop. Usually, the first step in running Gridgen is to select the software (Section 17.1 of the User Manual) to be used in the analysis. This will ensure that the grid generated by Gridgen is consistent with the capabilities of the analysis software. Since setting the analysis software is not the focus of this tutorial, you can use the default generic analysis software setting. No action on your part is needed.

6.6   Importing the Database

Now we need to import the intersecting pipes database into Gridgen. There are two possible ways that you may import the database into Gridgen. The first option is if you completed the tutorial in Section 6, you should already have the interesting pipes database available saved in a location on your computer as `pipes.dba`. You can read in the database in from the MAIN MENU:

1. **Input/Output**

2. Database **Import**

3. Navigate to the appropriate directory and select `pipes.dba` from the Browser.

4. **Open**

5. **Done**

The second option for importing the database is to load the Intersecting Pipes tutorial from the MAIN MENU and restart Gridgen, keeping only the database. From the Gridgen MAIN MENU:

1. **Tutorials**

2. **Intersecting Pipes**

3. **Done**

4. **Restart Gridgen**

5. Toggle off **Clear DB**

6. **Restart**

Before you go much further you may want to change the display attributes of the database. Turning off the display of domains and placing the database entities in outline mode will make the database easier to see and work with.

1. **Disp**

2. Toggle off **Doms**

3. Edit Dspla **DB**

4. Pick **All**

5. **Done**

6. render mode **Outline**

7. **Done - Apply Atts**

8. **Abort**

If you chose the first option for importing the database into Gridgen, you may have to rotate the image to get it to the orientation seen below (Section 2.7.3 of the User Manual). If you chose the second option for loading the database, the model should already be in this orientation.
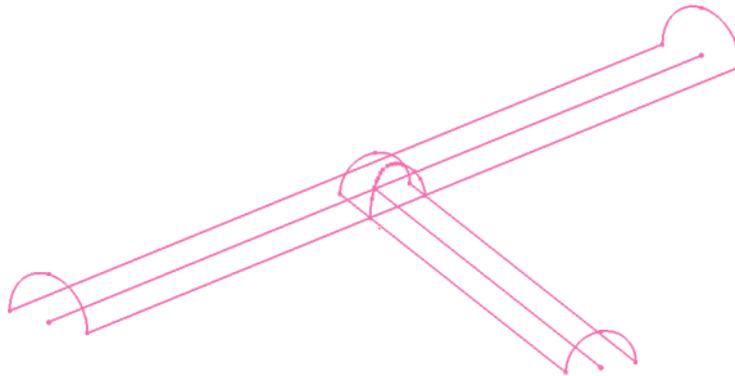
Figure 20.3 Pipes database

6.7   Building Polar Block for Main Pipe

The geometry you just imported will be used to determine the shape of the grid you will create next. Before starting to build the grid, set a default number of grid points to put on each new connector created. For convenience, we will specify a grid size of 2. From the MAIN MENU:

1. **Defaults**

2. Con Dim **dimen**

3. 2

4. **Done**

The blocks you will create are shown labeled in Figure 20.4. The below figure is an exaggerated version of the blocks and the topology. This will be used for reference in the discussion that follows.
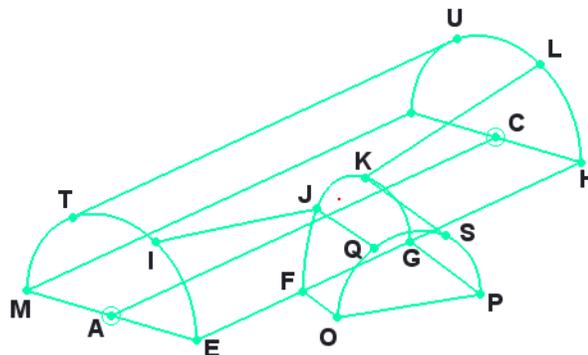
Figure 20.4 Block Topology Labels

You will create a polar block for the main pipe by running the glyph script `block-gui.glf`. It is located in the *gridgen_home_path*/`examples/tutorial/pipestd` directory. You may want to copy it to a local directory before trying to execute it in Gridgen.

The script was designed to offer three different types of block topologies, H-Block, Polar, and T-Block, for block creation. A script such as this is relatively easy to construct and essential to the process of constructing a mesh through a top down method. Of these three types of blocks offered, only the H-Block and the Polar block will be used for this tutorial. The block topology which will note be used, the T-Block, consists of two H-Blocks which adjoin at one face to form a topology roughly in the shape of a "T". Please feel free, once you have completed this tutorial, to experiment and try this block and the others offered on other database models.

For the main pipe, we will want to use a polar block to create the O-grid topology mentioned earlier. Assuming that you have copied the file `block-gui.glf` into your current working directory, from Gridgen's MAIN MENU:

1. **Glyph**

2. **Execute Script**

3. `block-gui.glf`

A small interface will appear as seen in the following figure.

Steps 4-5 require you to make your selections from within this interface.

4. Polar

5. **Run**

6. **Done**

The polar block should now be loaded in the Display window as seen in the following figure.
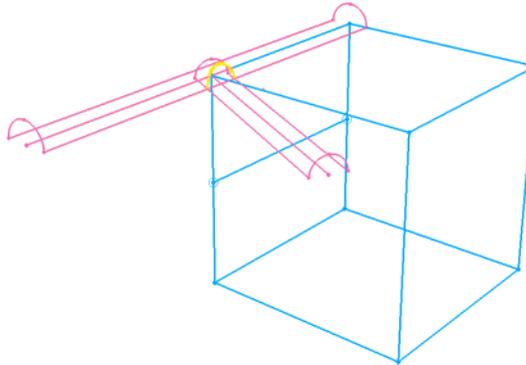
Figure 20.5 Polar Block

The next few steps you take will be to scale and align this block to fit the geometry. The polar block created by this script is 10 units in length along each edge with the pole running from $x = 0$, $y = 0$, $x = 5$ to $x = 10$, $y = 0$, $z = 5$. The main pipe has a radius of 1 unit and a length of 20 units. Since the block is 10 units long, a scale factor of 2 in the x-direction will be necessary to make the block fit the length of the main pipe. The main pipe's radius of 1 unit requires that the polar block will need to be 2 units wide in the y-direction and 1 unit tall in the z-direction to fit the half-cylinder of the main pipe. The scaling factors of 0.1 and 0.2 will be used to fit the block in the y- and z-directions respectively.   From the MAIN MENU:

1. **Blocks**

2. **Modify**

3. Pick **All**

4. **Done**

5. **Scale**

6. **Enter Anchor via Keybrd**

7. 0, 0, 5

8. **Enter Scaling Factors**

9. 2, 0.1, 0.2

10. **Done - Scale**

11. **Done - Replace Blocks**

12. **Maintain Linkage**

The polar block should be scaled down around the point $x = 0$, $y = 0$, and $z = 5$ and running parallel to

the main pipe. We now need to translate it over to the main pipe, then rotate it. You will move the polar block by setting the handle you will move it by to the end of the pole that you scaled it about. Once you have set this handle, you will move the block to point A, the end of the main pipe pole (see Figure 20.4). The polar block will then be rotated about point A. From the BLOCK COMMANDS menu:

1. **Modify**

2. Pick **All**

3. **Done**

4. **Translate**

5. Select the end of the polar block pole corresponding to x = 0, y = 0, z = 5.

6. Drag the polar block to point A, the end of the main pipe pole.

7. Set Handle **by Picking**

8. **Done - Translate**

9. **Rotate**

10. **Use X-principal Axis**

11. **Enter Rotation Angle**

12. -90

13. **Done - Rotate**

14. **Done - Replace Blocks**

15. **Maintain Linkage**

The polar block should now appear as to contain the main pipe of the intersecting pipes as seen in the following figure.
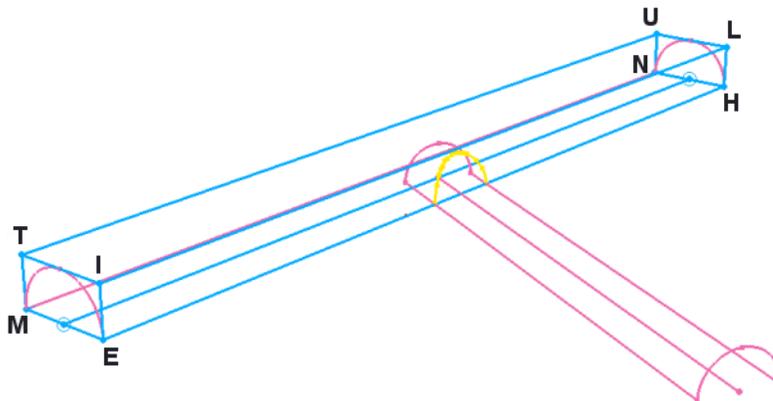
Figure 20.6 Main Pipe Polar Block

In Figure 20.6, the block nodes have been labeled for reference in the block's fit to the geometry. To fit the block to the geometry, you will need to split domain <u>EILH</u> into three domains in the area of the intersection curve seen in the following figure.
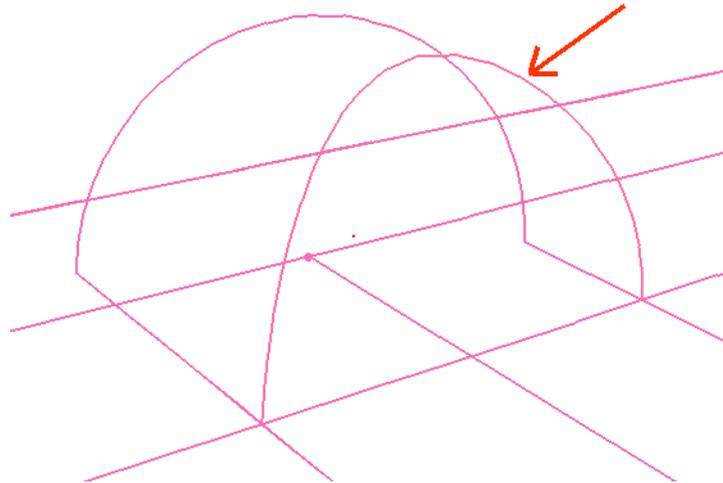
Figure 20.7 Intersection Curve Computed by Gridgen

The middle domain formed from the domain split will eventually form the end of the branch pipe block. Splitting the domain <u>EILH</u> will require Gridgen to also split the domains connectors. In Gridgen, a connector must have three or more points before it can be split. Because the default connector dimension was set to 2, the connectors running the length of the pipe were created with a dimension of 2. All four of these connectors need to be re-dimensioned to a larger value before domain <u>EILH</u> may be split. We will re-dimension them to a value close to their final dimension.

Before redimensioning these connectors, it is important to understand how the **Redimension Grids** button works when selecting a connector that is already a part of a structured domain or block. In cases where such a connector is chosen, **Redimension Grids** is configured to automatically propagate that connector's dimension changes to all connectors within the domain or block that must maintain the same dimensions ([Section 9.2](#) of the User Manual). This allows you to quickly change the dimensions of a grid by only selecting a few connectors for redimensioning instead of all connectors. As such, you will only have to select connector <u>TU</u> to redimension all the lengthwise connectors.   From the MAIN MENU:

1. **Connectors**

2. **Redimension Grids**

3. Select connector <u>TU</u>.

4. `101`

5. **Done - Apply New Dims**

Now you are ready to split domain <u>EILH</u>. You will want to split the domain just slightly to either side of the intersection curve. The end result is that the intersection curve will be entirely contained within one small domain. From the MAIN MENU:

1. **Domains**

2. **Modify**

3. Select domain <u>EILH</u> from the Display window or from the Browser list (entity 8).

4. **Done**

5. **Split**

6. **I Lines**

7. Move the Hilite in the Display window to a point just to the right of where the branch pipe intersects the main pipe.

8. **Done - Split at Hilite**

The domain EILH has now been split into two domains, EIKG and GKLH respectively, seen in Figure 20.8.
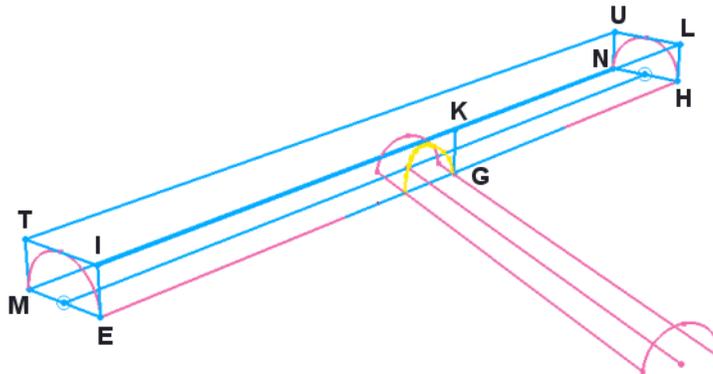
Figure 20.8 Domains EIKG and GKLH

Now you need to split domain EIKG at the other location that the branch pipe intersects the main pipe. From the MAIN MENU:

1. **Domains**

2. **Modify**

3. Select domain EIKG from the Display window or from the Browser list (entity 9).

4. **Split**

5. **I Lines**

6. Move the Hilite in the Display window to a point just to the left of where the branch pipe intersects the main pipe.

7. **Done - Split at Hilite**

The three new domains EIJF, FJKG, and GKLH should now appear as in Figure 20.9.
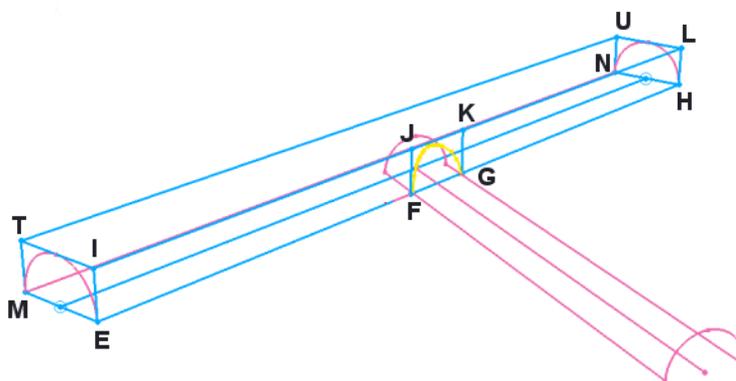
Figure 20.9 Split Domains of Main Pipe Polar Block

You now need to prepare domain <u>FJKG</u> for the next block you will create with the block creation glyph script (<u>Section 6.8</u>). This next block will be fit over the branch pipe which will require domain <u>FJKG</u> to be one of its faces. To accomplish this, you will need to take advantage of the manner in which Gridgen treats connectors and domains. If two connectors in Gridgen have the same endpoints, number of grid points, and distribution, those connectors are considered identical. One of the connectors is thrown away and the remaining connector assumes the linkages of both connectors. Gridgen treats domains in a similar manner.

Changing the dimension of connectors <u>JK</u> and <u>FG</u> to 2 will make all of domain <u>FJKG</u>'s connectors match those of the corresponding face of the new branch pipe block once the four corners of the block's face have been moved to the points F, J, K and G. This means that Gridgen will consider those connectors and domains identical which will result in one domain that has linkages to the main pipe and branch pipe blocks.

For your grid, connector TU will also need to be re-dimensioned to maintain computationally balanced system.

To re-dimension connectors <u>JK</u>, <u>FG</u>, and <u>TU</u> from the MAIN MENU:

1. **Connectors**
2. **Redimension Grids**
3. Select connector <u>JK</u>.
4. 2
5. Select connector <u>TU</u>.
6. 93
7. **Done - Apply Dims**
8. **Done**

6.8   Building H-Block for Branch Pipe

Generating the H-block for the branch pipe will require you to run the glyph script "`block-gui.glf`" once more. From the MAIN MENU:

1. **Glyph**
2. **Re-execute Last Script**

Steps 3-4 should be completed from within the Ready Made Blocks interface.

3. H-Block

4. **Run**

5. **Done**

You should now have an H-block positioned at the origin. You will be connecting one face of this H-block to the end of the branch pipe and the opposite face to domain FJKG. Figure 20.10 illustrates the points that will be referenced when connecting a face of the H-block to the end of the branch pipe.

Figure 20.10 H-Block for the Branch Pipe

To attach the domain OQSP to the end of the branch pipe, you will need to translate point O to point O', point Q to point Q', point S to point S', and point P to point P'. You will then need to scale the H-Block to fit the branch pipe. Since the H-Block is 4 units in length along any edge and the branch pipe has a radius of 0.75 units, you will need to use scaling factors of 0.4 in the x-direction and 0.2 in the z-direction. For the y-direction, we will use a scaling factor of 1 to leave the block's size in the y-direction unchanged. Please note that these scaling factors are not exact, but merely approximations to make fitting the block to the pipe slightly easier. From the MAIN MENU:

1. **Blocks**

2. **Modify**

3. Select the H-block from the Display window or the Browser list (entity 2).

4. **Translate**

5. Move point O to point O' by selecting point O with your *right mouse* button and dragging it to point O'.

6. Set Handle **by Picking**

7. **Done - Translate**

8. **Scale**

9. Select point O.

10. **Enter Scaling Factors**

11. `0.4, 1, 0.2`

12. **Done - Scale**

13. **Done - Replace Blocks**

14. **Maintain Linkage**

Now you will need to attach point P of the domain OQSP to point P' on the branch pipe using the Segment: **Edit** command found within the CONNECTOR COMMANDS menu. From the MAIN MENU:

1. **Connectors**

2. **Modify**

3. Select connector OP.

4. **Done**

5. Segment **Edit**

6. Move point P to point P' by selecting point P with your ***right mouse*** button and dragging it to point P'.

7. Store CP **by Picking**

8. **Done Editing**

9. **Done - Replace Connectors**

10. **Maintain Linkages**

Your branch pipe block should now appear linked to the branch pipe as seen in Figure 20.11.

Figure 20.11 H-Block Connected to the Branch Pipe

The same procedure used to connect point O to O' and point P to P' should will now be used to connect point Q to Q' and point S to S'. Since the points Q' and S' do not physically exist you will need to use an option to pick the closest database point instead of picking a pre-existing point. From the MAIN MENU:

1. **Connectors**

2. **Modify**

3. Select connector <u>QS</u>.

4. **Done**

5. Segment **Edit**

6. Move point Q to Q' by selecting point Q with your *right mouse* button and dragging it to just beyond the branch pipe in that area.

7. Store CP **at Clsest DB Pt**

8. Edit... **Pick**

9. Select point S.

10. Move point S to S' by selecting point S with your *right mouse* button and dragging it to just beyond the branch pipe in that area.

11. Store CP **at Clsest DB Pt**

12. **Done Editing**

13. **Done - Replace Connectors**

14. **Maintain Linkages**

Next, you need to connect opposite face of the block opposite to domain <u>FJKG</u> using the same technique you used to connect points O and P to their corresponding branch pipe locations. You will use the Segment **Edit** and the Store CP **by Picking** commands to move connector endpoints to their corresponding locations on the <u>FJKG</u> domain. This is left as an exercise for you to complete since you have already used these steps once before. When you have finished, your block should appear as seen in the following figure.
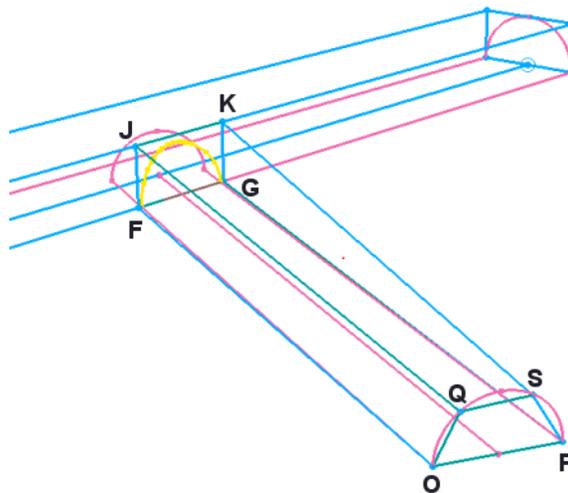
Figure 20.12 H-Block Connected to Branch Pipe and Polar Block

Both the Polar Block and the H-Block share the domain <u>FJKG</u>. This domain now has to be modified to follow the shape of the intersection curve. First, move points F and G to the end points of the intersection curve using the Segment **Edit** command in the CONNECTOR COMMANDS menu. From the MAIN MENU:

1. **Connectors**

2. **Modify**

3. Select connector <u>FG</u>.

4. **Done**

5. Segment **Edit**

6. Select point F.

7. Keep the *right mouse* button pressed and drag point F to the end of the intersection curve.

8. Store CP **by Picking**

In Figure 20.13, in the close up view of the domain and the intersection curve, you can see the arrows which point to the ends of the curve. These point to the locations that you want to move points F and G to respectively.
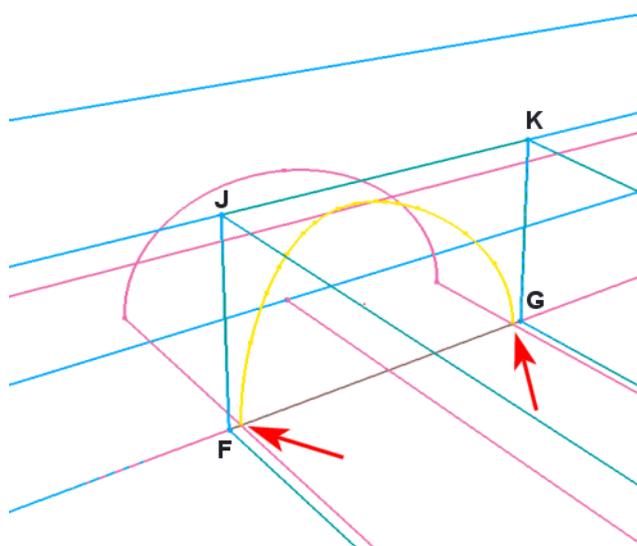
Figure 20.13 Main and Branch Pipe Intersection Curve

Now that you have moved point F, you will need to move point G to the other endpoint of the intersection curve.

9. Edit... **Pick**

10. Select point G.

11. Keep the *right mouse* button pressed and drag point G to the end of the intersection curve.

12. Store CP **by Picking**

13. **Done Editing**

14. **Done - Replace Connectors**

15. **Maintain Linkages**

Points J and K now need to be moved to follow the intersection curve. To make the connectors <u>FJ</u>, <u>JK</u>, and <u>KG</u> follow the intersection curve exactly, you will need to split the intersection curve into three sections. From the MAIN MENU:

1. **Database**
2. **Modify**
3. Select the intersection curve in the Display window or in the Browser list (entity 11).
4. **Done**
5. **Maintain Linkage**
6. **Split**
7. **Move O via Keybrd**
8. `0.3`
9. **Split at Hiliter**
10. Select **Split It?**
11. **Modify**
12. Select the remainder of the intersection curve from the Browser list (entity 12).
13. **Done**
14. **Split**
15. **Move O via Keybrd**
16. `0.6`
17. **Split at Hiliter**
18. Select **Split It?**
19. **Done**

Now that the intersection curve has been split into three sections, you will need to create a connector of dimension two on each section.   At the beginning of this tutorial, you set the default connector dimension to 2, so now when you create your three new connectors they will already be dimensioned for you. From the MAIN MENU:

1. **Connectors**
2. **Create**
3. **On DB Entities**
4. Select the intersection curves three database entities from the Display window or from the Browser list (entities 11, 12, and 13).
5. **Done**
6. **Done**

The three new connectors should now appear in green along the intersection curve as seen in the following figure.
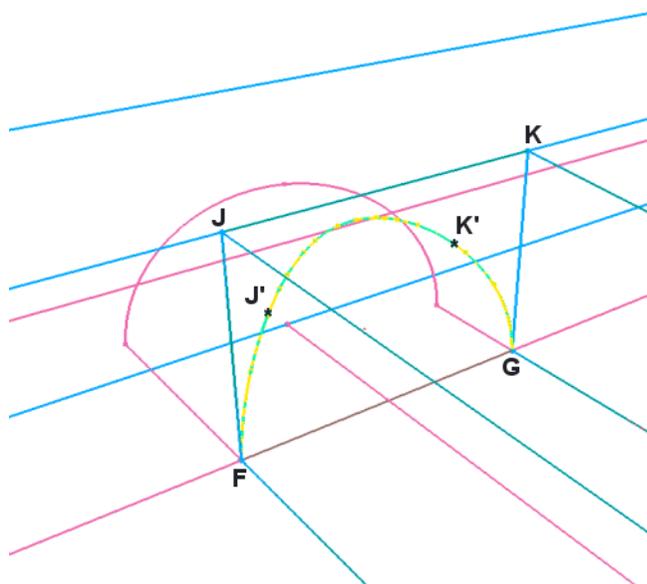
Figure 20.14 Intersection Curve Connectors

Now that connectors FJ', J'K', and K'G have the same dimension as connectors FJ, JK, and KG, you can connect their nodes so that they will be automatically merged together by Gridgen. Since you want the H-Block to conform to the shape of the branch pipe, you will need to move point J to J' and point K to K'. From the MAIN MENU:

1. **Connectors**

2. **Modify**

3. Select connector JK.

4. **Done**

5. Segment **Edit**

6. Move point J to point J' by selecting point J with your *right mouse* button and dragging it to point J'.

7. Store CP **by Picking**

8. Edit... **Pick**

9. Select point K.

10. Move point K to point K' by selecting point K with your *right mouse* button and dragging it to point K'.

11. Store CP **by Picking**

12. **Done Editing**

13. **Done - Replace Connectors**

14. **Maintain Linkages**

15. **Done**

Domain FJKG should now following the intersection curve as seen in Figure 20.15.
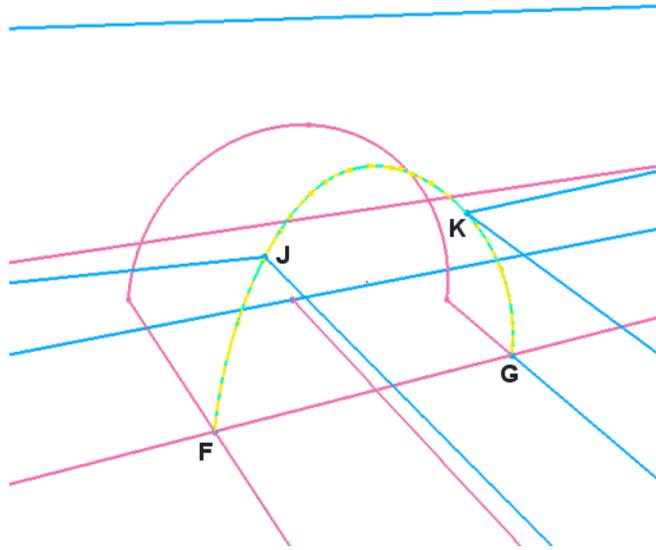
Figure 20.15 Domain FJKG Conformed to the Pipe Intersection

The blocks are now ready for the final stage of grid construction - projecting the block faces onto the database surfaces.

6.9   Connector and Domain Projection

Projecting the block faces onto the database surfaces is the last step in the top down grid construction process. Before projecting the faces, all connectors must be redimensioned via the **Redimension Grids** button. Afterwhich, the connectors, then the domains are projected to the database.
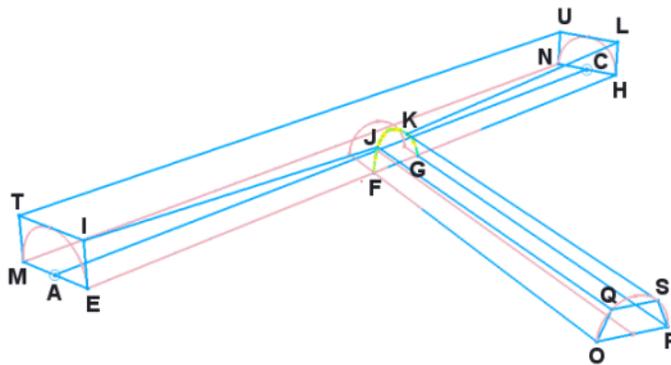
Figure 20.16 Pipes Labeled Blocks

For your grid, you will only need to redimension seven connectors for the entire grid to be redimensioned appropriately. The following table lists the connectors that need to be selected and their new dimensions.

Please reference Figure 20.16 to identify which connectors to select.

Connectors Selected for Redimensioning

Connector	Old Dimension	New Dimension
EM	2	11
TI	2	11
MT	2	7
IE	2	7
SQ	2	11
TU	93	102
FO	2	44

From the MAIN MENU:

1. **Connectors**
2. **Redimension Grids**
3. Select connector EM.
4. `11`
5. Select connector TI.
6. `11`
7. Select connector MT.
8. `7`
9. Select connector IE.
10. `7`
11. Select connector SQ.
12. `11`
13. Select connector TU.
14. `102`
15. Select connector FO.
16. `44`
17. **Done - Apply New Dims**
18. **Done**

You are now ready to project your connectors onto the database. From the MAIN MENU:

1. **Connectors**

2. **Modify**

3. Select connector <u>MT</u>, <u>TI</u>, <u>IE</u>, <u>NU</u>, <u>UL</u>, <u>LH</u>, <u>OQ</u>, <u>QS</u>, <u>SP</u>, <u>TU</u>, <u>IJ</u>, <u>KL</u>, <u>JQ</u>, <u>KS</u>.

4. **Done**

5. **Project**

6. **Closest Pt**

7. Select **Nodes Too**.

8. **Done - Project**

9. **Done - Replace Connectors**

10. **Maintain Linkages**

11. **Done**

Once you have finished projecting your connectors, you will need to project your domains in the same fashion. From the MAIN MENU:

1. **Domains**

2. **Modify**

3. Select domains <u>MTUN</u>, <u>ITUL</u>, <u>EIJF</u>, <u>GKLH</u>, <u>FJQO</u>, <u>QJKS</u>, <u>GKSP</u>.

4. **Done**

5. **Project**

6. **Closest Pt**

7. Select **Cons Too**

8. **Done - Project**

9. **Replace Domains**

10. **Maintain Linkage**

11. **Done**

To improve the corner cells of domains <u>OQSP</u> and <u>FJKG</u> you may want to smooth those domains using the Structured Domain Solver. From the MAIN MENU:

1. **Domains**

2. Run Solver **Structured**

3. Select domains <u>OQSP</u> and <u>FJKG</u>.

4. **Done**

5. **Run N Iters**

6. `11`

7. **Done - Save**

8. **Done**

Your grid is now complete! To examine the grid, you will want to turn on the display of your domains and edit their display attributes. From the DISPLAY COMMANDS menu:

1. Toggle on Show **Doms**

2. Edit Dspla **Dom**

3. Pick **All**

4. **Done**

5. display style **Shaded/Wireframe**

6. solid/shade color **All**

7. **Done - Apply Atts**

8. **Abort**

9. **Done**

The intersecting pipes grid you have constructed should now appear as in the following figure.
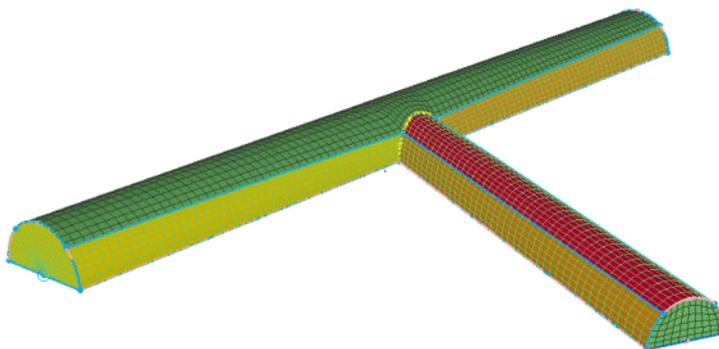
Figure 20.17 Completed Intersecting Pipes Grid

Before you finish the tutorial, you will want to save your work. To save your database (.dba) and gridgen restart (.gg) files, you will need to go to the INPUT/OUTPUT COMMANDS menu accessible from the MAIN MENU:

1. **Input/Output**

2. Database **Export**

3. Make sure **composite** is selected.

4. Done

5. Type In... **Name**

6. `pipestd`

7. Gridgen **Export**

8. Type In... **Name**

9. `pipestd`

Please note that, due to database references in the Gridgen restart file, it is always a good idea to export your database model prior to exporting your Gridgen restart file.

You have now completed the tutorial! You should be able to use the techniques and methods you have learned in this tutorial for your own geometries. For practice, you may want to try gridding the other geometries found in the remaining tutorials in this workbook using the top-down approach.

7. B-747 Nacelle: Working With An IGES Database

7.1 Introduction

This tutorial will give you a guided tour of the Gridgen process for creating a simple two block structured grid for the internal flow field of a 747 inlet geometry imported from an IGES formatted database file. We encourage you to complete this tutorial before using IGES databases of your own to create more complex grids.

7.2 Topics Covered

This tutorial covers the basics of using an IGES database in Gridgen. Gridgen skills you will cover during this tutorial session are:

- creating database constrained connectors using **On DB Entities**
- creating connectors using DB cnstr **Line** segments
- creating database constrained connectors with branch segments
- modifying and redimensioning connectors
- using the SET DEFAULT VALUES menu and the **Redimension Grids** command
- creating database constrained domains
- using **Project** to put domains onto database entities
- elliptically smoothing database constrained domains using Run Solver **Structured**
- creating blocks

7.3 Background

This tutorial covers advanced topics and assumes that you already understand the basics that are covered in previous tutorials, particularly Section 2 and Section 4.

7.4 Geometry

The 747 inlet configuration you will use for this tutorial is shown in Figure 21.1. The portion of the geometry that we are using consists of a central hub and a nacelle. For simplicity, we are not modeling the engine or the mounting hardware for the nacelle. Since the geometry is symmetric about a vertical plane only half the geometry is included. We can do the same thing for the grid. If both sides of the grid are needed, say for a yawed case, it is easy to copy and mirror the grid about the symmetry plane after it is completed.
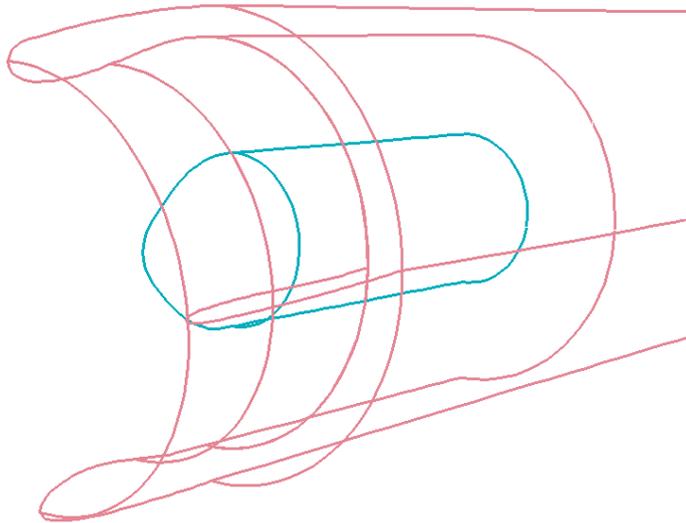
Figure 21.1 The 747 Inlet Configuration To Be Gridded

The grid you will generate consists of two blocks arranged as shown in Figure 21.2. One block will be essentially rectangular and will be attached to the nose of the central hub. This block will run forward to the inlet inflow plane. For this first block i-direction grid lines will run in the positive x-direction, j-direction grid lines will run in the positive y-direction, and k-direction grid lines will run in the positive z-direction.

The second block will be generated in a "C-Grid" style and will wrap around the central hub and the internal hub nose block. For the second block i-direction grid lines will run in the positive x-direction, j-direction grid lines will run around the hub from bottom to top, and k-direction grid lines will run from the hub out to the inner surface of the inlet. The choices of blocking system and grid topology are up to you; this particular blocking system is only one of many that could be used for this inlet configuration. The requirements of the particular flow solver or other analysis software you are using should be considered when planning the grid.

Now that a topology for the blocking system has been chosen we can begin creating the grid. For convenience we have assigned a naming convention to the connector node points which will be used to create the grid. The node labels are shown in Figure 21.2 (for clarity connectors and nodes are shown only; the database model is not visible in this image). Note that in order to accurately model this internal flow field, one would likely grid and solve the region external to the inlet as well. To satisfy the goals of this tutorial, however, it is sufficient to grid only the internal region of the inlet. For further practice you are free to develop an external grid for this database geometry.
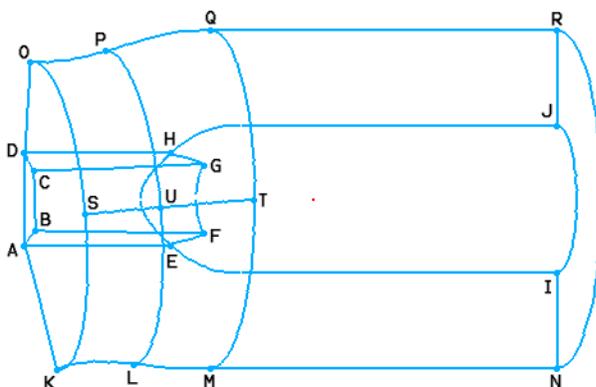
Figure 21.2 Two Block Grid Topology for the Inlet

7.5   *Getting Started*

Begin by starting Gridgen, either by typing the string `gridgen` on Unix workstations or simply double-clicking the Gridgen shortcut on your PC's desktop. The first step in running Gridgen is to select the software (Section 17.1 of the User Manual) to be used in the analysis. This will ensure that the grid generated by Gridgen is consistent with the capabilities of the analysis software. For this analysis we will be using the WIND flow solver (by default Gridgen selects the generic solver). To select WIND do the following:

1. **Analysis S/W**

2. **Select Analysis S/W**

3. **Next Page**

4. **WIND**

5. **3D**

6. **Next Page**

7. **Done**

8. **Done**

7.6   *Importing the Database*

The 747 inlet geometry we will use for this example is in an IGES file named `747_new.iges`. It is located in the *gridgen_home_path*/`examples/tutorial/747` directory. You may want to copy it to a local directory before trying to import it into Gridgen.

Assuming that you have copied `747_new.iges` to your current working directory, the procedure to read it in to Gridgen starting from the MAIN MENU is:

1. **Input/Output**

2. Database **Import**

3. Select `747_new.iges` from the Browser.

4. **Open**

5. **Done**

After the file is read, rotate the image in the Display window until the model orientation is similar to that in Figure 21.2. To remove some clutter from the display, we will disable database entities that will not be necessary for this tutorial. There are other advantages to disabling unnecessary database entities. Gridgen's rendering speed will be improved. There will be fewer items to pick from when creating connectors on database entities. And domain projections will be more efficient since Gridgen will not attempt to project onto disabled entities. To disable the database surfaces on the exterior of the nacelle:

1. **Database**

2. **En/Disable**

3. Select or toggle entities 3, 4, 6, and 7.

4. **Done**

5. **Done**

To simplify connector creation we are going to modify the database a little bit. It will keep the same shape, but we are going to change its topology slightly.

First, split the surface on the front of the centerbody by:

1. **Database**

2. **Modify**

3. Select the centerbody nose surface (entity 2).

4. **Done**

5. **Split**

6. **V**

7. **Move ○ via Keybrd**

8. `0.5`

9. **Split at Hiliter**

10. Confirm the split with **Split It?**.

The nose surface is now split into two separate surfaces. Next, split the upper surface at the front of the inlet nacelle.

1. **Modify**

2. Select the front upper nacelle surface (entity 11).

3. **Done**

4. **Split**

5. **U**

6. **Move ○ via Keybrd**

7. `0.75`

8. **Split at Hiliter**

9. Confirm the split with **Split It?**.

There is one more surface to split - the lower surface at the front of the inlet nacelle.

1. **Modify**

2. Select the front lower nacelle surface (entity 9).

3. **Done**

4. **Split**

5. **U**

6. **Move ○ via Keybrd**

7. `0.25`

8. **Split at Hiliter**

9. Confirm the split with **Split It?**.

10. **Done**

Notice that we split the lower and upper surfaces at very different u-values because they are parametrized differently. We did not split them at exactly the same location, but it is pretty close. These new and existing surface gaps will require us to make some adjustments to our model's tolerances in the SET DEFAULT VALUES menu. Your geometry should now look like the one in Figure 21.3.
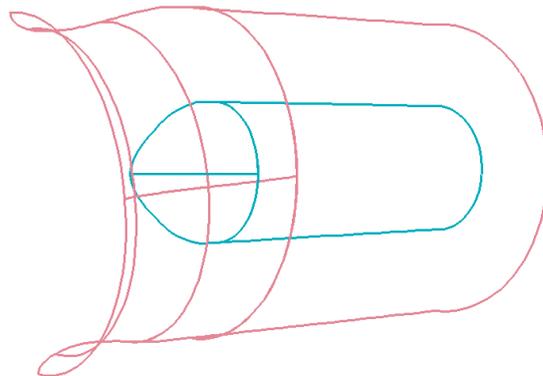
Figure 21.3 Inlet Geometry After Modification

Later you will create grid entities on the modified geometry. It is important to save the modified geometry so that the association with the grid can be preserved in future sessions. Save your database model now.

1. **Input/Output**

2. Database **Export**

3. type **composite**

4. **Done**

5. Navigate using the file Browser to a suitable directory where you have permission to save this new file.

6. Type In... **Name**

7. `747_mod`

8. **Done**

We have completed all of the database tasks required. You can now proceed to building the connectors for

this grid.

Finally, to help speed grid construction, we will set a default value for connector dimension and change the connector and node tolerances using the SET DEFAULT VALUES menu option. To make these changes:

1. **Defaults**
2. Con Dim **dimen**
3. 30
4. **Tolerances**
5. **Connector**
6. 0.02
7. **Done**
8. **Done**

Any connectors you create after this point will automatically have 30 points assigned to them. If you want a different number of grid points on any connector you will have to change the dimension manually or change the default value. We changed the connector tolerance to a higher value than the usual default so that later in the tutorial Gridgen will automatically grid over gaps in the database by removing coincident connectors. Note that this change automatically increased the node tolerance as well. The node tolerance may not be less than the connector tolerance.

7.7   Creating The First Block

The first connectors you will create correspond to lines <u>AB</u>, <u>BC</u>, <u>CD</u>, and <u>DA</u> in Figure 21.2. Their shapes will different than those in the figure when you first create them. We will take care of the shape later.

Starting from the MAIN MENU:

1. **Connectors**
2. **Create**
3. **2 Point Connectors**
4. Add CP □ **via Keybrd**
5. 0, 0, -6
6. Add CP □ **via Keybrd**
7. 0, -6, -6
8. Add CP □ **by Picking**
9. Add CP □ **via Keybrd**
10. 0, -6, 6
11. Add CP □ **by Picking**
12. Add CP □ **via Keybrd**
13. 0, 0, 6
14. Add CP □ **by Picking**
15. Position the 3D cursor over Point A.
16. Add CP □ **by Picking**

17. **Done**

18. **Done Creating Conns**

We have four connectors, so go ahead and create a domain.

1. **Doms**

2. **Create**

3. Cell Type **structured**

4. **Assemble Edges**

5. Toggle on **Auto Next Edge** and **Auto Complete**.

6. Pick connector <u>AB</u>.

7. Pick connector <u>BC</u>. Gridgen will then automatically complete the domain.

8. **Done**

This domain will be the front face of the first block of our grid. The back face will be similar to the front but projected onto the centerbody. Copy the first domain and project it onto the surface by starting from the DOMAIN COMMANDS menu and:

1. **Copy**

2. Select the domain.

3. **Done**

4. **Abort - Don't Translate**

5. **Project**

6. **Linear**

7. **Cons Too**

8. Rotate the image in the Display window so that you are looking down the x-axis. Because we are using linear projection we need to align the view with the direction in which we want the grid projected. An easy way to get the right view is to first hit the *r* hotkey to restore the default view. Then, while holding down the *Page Down* key, press the *Pad /* key to get a 90 degree rotation about the y-axis.

9. **Done - Project**

10. **Save Domains**
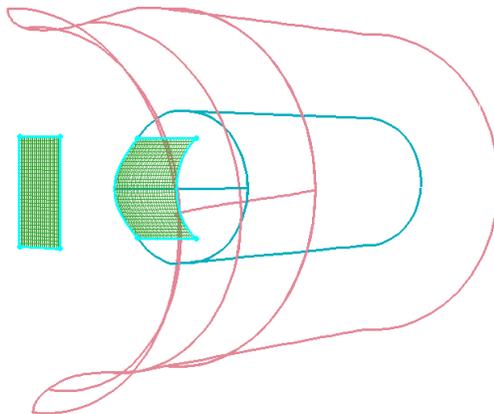
Figure 21.4 First Two Domains of Block 1

You should now have the two domains shown in Figure 21.4. Rotate the view to verify that the grid points and connectors of the domain you projected are on the database surface. The procedure you just went through is one of the ways that you can conform the grid to the shape of the geometry. We will look at some others during the construction of the second block.

You are on your own for the remainder of the construction of the first block. (You should have already completed the previous tutorials!) Create the four remaining connectors for this block (<u>AE</u>, <u>BF</u>, <u>CG</u>, and <u>DH</u>) as **2 Point Connectors**. Use them to create the four other domains (<u>ABEF</u>, <u>BCGF</u>, <u>CDHG</u>, and <u>DAEH</u>). Finally, put the block together using one domain for each face. Save the grid after you complete the first block.

7.8   Creating Block Two

Now you are ready to begin the construction of the second block. Before beginning, turn off the display of the domains in Block 1, so that the Display window is less cluttered.

1. **Disp**

2. Edit Dspla **Dom**

3. Pick **All**

4. **Done**

5. display style **Off**

6. **Done - Apply Atts**

7. **Abort**

8. **Done**

Now you are ready to begin creating some more database constrained connectors. This time you will use the DB cnstr **Line** segment type to create some connectors that span multiple database entities on the center-body surface.

1. **Cons**

2. **Create**

3. **Add Segment**

4. DB cnstr **Line**

5. DB Ent **Pick**

6. Select the centerbody upper nose surface.

7. Position the 3D cursor over Point H. If you are having trouble moving the 3D cursor try rotating the view. Gridgen converts your mouse movements into database constrained screen movements of the cursor. This works best when the database surface at the cursor location is approximately in the plane of the screen. It can have trouble when the surface is perpendicular to the view.

8. Add CP □ **by Picking**

9. Position the 3D cursor at the top back corner of the surface at (u,v) = (0,0).

10. Add CP □ **by Picking**

11. DB Ent **Pick**

12. Select the main (aft) surface of the centerbody.

13. Note that the 3D cursor is now on the new database surface, but it is still at the same physical location. The `Distance to Last Point` shown in the Blackboard window should be 0. Move the 3D cursor a slight distance aft, until the `Distance to Last Point` is approximately 0.1.

14. Add CP □ **at +**

15. Position the 3D cursor at the top back corner of the surface at (u,v) = (1,0).

16. Add CP □ **by Picking**

17. **Done - Save Segment**

18. **Done - Save Connectors**

You have now completed a parametrically defined connector that spans two different database entities. In order for these DB Cnstr segment types to be successful, a minimum of two control points must be saved on each database entity. The small piece of the connector you made approximately 0.1 in length is referred to as a "branch segment" (Section 6.2.2 of the User Manual). That portion of the connector is not constrained to either database entity and must have a small but finite length in order for Gridgen to not convert it to a pole segment or singularity. Next make a connector that goes around the centerbody from the top to the bottom.

1. **Add Segment**

2. DB Cnstr **Line**

3. Add CP □ **by Picking**

4. Position the 3D cursor at the back bottom corner of the same surface.

5. Add CP □ **by Picking**

6. **Done - Save Segment**

7. **Done - Save Connectors**

Make database constrained connector EI in the same way:

1. **Add Segment**

2. DB cnstr **Line**

3. Add CP □ **by Picking**

4. Position the 3D cursor at the front bottom corner of the same surface.

5. Add CP □ **by Picking**

6. DB Ent **Pick**

7. Select the lower centerbody nose surface.

8. Move the 3D cursor slightly forward (until the `Distance to Last Point` is approximately 0.1).

9. Add CP □ **at +**

10. Position the 3D cursor at Point E.

11. Add CP □ **by Picking**

12. **Done - Save Segment**

13. **Done - Save Connectors**

14. **Done Creating Conns**

We are almost ready to make another domain. First, we need to redimension some connectors in the first block.

1. **Redimension Grids**

2. Select connector EF.

3. 8

4. Select connector FG.

5. 16

6. **Done - Apply New Dims**

Now make the domain:

1. **Doms**

2. **Create**

3. **Assemble Edges**

4. Make sure **Auto Next Edge** and **Auto Complete** are toggled on.

5. Pick connector HJ.

6. Pick connector JI. Gridgen will recognize that connector IE is the next edge and will add it to the domain.

7. Pick connector EF. Gridgen will recognize that connectors FG and GH are needed to complete the domain and will add them.

8. **Done**

The Display window should now look like Figure 21.5. Rotate the image and note that the domain does not lie on the database surfaces. It is actually easy in Gridgen to know immediately upon creation which domains are database constrained automatically and which are not. When they can be constrained automatically, Gridgen renders them in a purple color; if not, they are green. See the end of Section 7.2 of the User Manual for more details regarding the algebraic methods used to initialize structured domains. Gridgen was not able to automatically constrain this domain because it spans multiple database surfaces. Before proceeding, put it on the database by projecting it.

1. **Modify**

2. Select the newly created domain (domain 7).

3. **Done**

4. **Project**

5. **Cylindrical**

6. **Use X-principal Axis**

7. **Proj Out**

8. **Int Pts**

9. **Done - Project**

10. **Replace Domains**

11. **Maintain Linkage**
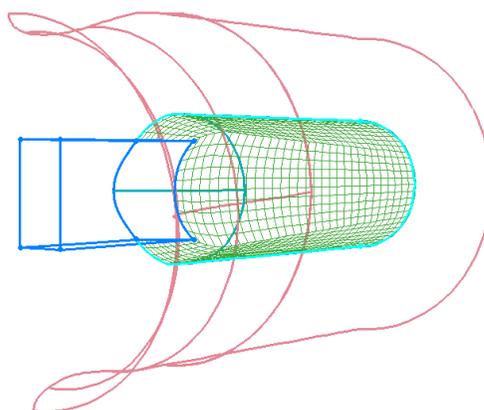
Figure 21.5 Centerbody Domain Before Projecting

Now when you rotate the image you should see that the grid is on the database.

The domains on the centerbody can be improved by refining them with Gridgen's elliptic PDE methods. We will modify some of the elliptic solver attributes to ensure that the domains stay on the database surfaces and to let the boundary between the two domains on the centerbody float to a new position resulting in higher grid quality.

1. Run Solver **Structured**

2. Select the two domains on the centerbody surface (domains 2 and 7).

3. **Done**

4. **Set Solver Attribs**

5. Apply to **All**

6. **Solver Bound. Conds.**

7. **Floating**

8. **Done Setting Attributes**

9. Elliptic Slvr **Run N Iters**

10. `30`

11. **Done - Save**

Your grid should now look like the one in Figure 21.6. Note that the connectors between the two domains are a different shape now than they were before running the elliptic solver. By setting the **Floating** boundary condition, you told Gridgen to let their shape be computed as part of the PDE solution just like they were interior grid lines. Not all of the connectors moved even though we used Apply to **All** for the floating boundary condition because it only applies to connectors that are shared between domains that are currently being elliptically refined. For this case, the only connectors that we wanted to float were also the only connectors between the two domains we were refining. Now would be another good opportunity to save your grid.
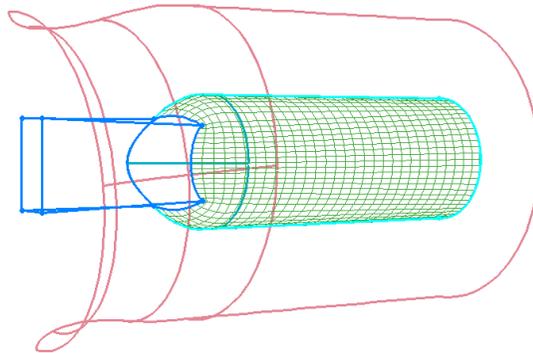
Figure 21.6 Centerbody Domain After Elliptic Refinement

Now back to creating connectors and domains. The next domains we will create are the ones on the interior wall of the nacelle. We will use another way of creating connectors and domains constrained to databases, **On DB Entities**, which lets you pick a set of database surface entities on which to create domains. For any database surface that you pick, Gridgen will create a connector for each edge of the surface using the default connector dimension and spacing constraints and will then attempt to put these connectors together in the definition of a domain. This command will only be successful in completing structured domains from these connectors when the database surfaces have exactly four edges. (If you are setting the connector dimensions based on average spacing, the opposite edges may not have the same number of grid points, and Gridgen will not be able to put a structured domain on that surface.)

Try it now:

1. **Doms**

2. **Create**

3. Toggle on **Auto Merge**.

4. **Set Toler**

5. `0.1`

6. **On DB Entities**

7. Select the 5 surfaces on the nacelle wall corresponding to OPUS, SULK, PQTU, UTML, and QRNM.

8. **Done**

Now you have a lot more connectors and five more domains! Note the purple color of the domains indicating they were automatically database constrained. However, the domain on the aft-most surface, QRNM, has the same dimension on all edges as the forward four domains, just as we instructed Gridgen. Since the interfacing connectors along the boundary between QRNM and PQTU/UTML do not match dimensionally, Gridgen does not automatically merge them. Therefore, we now have coincident connectors along this circumferential boundary between the larger QRNM and the two domains in front of it. Make the necessary dimension changes and Merge the coincident connectors now.

1. **Redimension Grids**

2. Select connector QT.

3. `16`

4. Select connector TM.

5. `15`

6. **Done-Apply New Dims**

7. **Cons**

8. Merge **Auto**

9. `0.1`

10. **Doms**

You may have already noticed that the grid on domain UTML looks a little strange. Figure 21.7 shows a view of this domain from the bottom, looking up. Gridgen recognized that all the connectors for this domain lie on the same database entity, so it created the grid using the parametric transfinite interpolation method (Section 10.1.5 of the User Manual). This is usually a nice property because it ensures that the grid points are on the database surface. In fact, it worked well on all the other surfaces of the nacelle. Occasionally you will get a database surface that has an odd parametrization. That is the case here and is the cause of the interior point distribution problem.
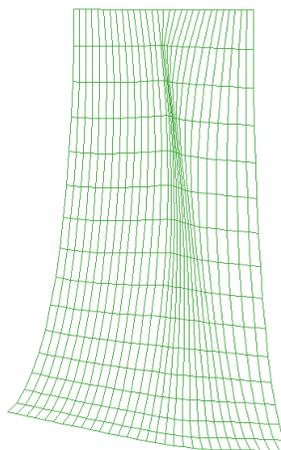
Figure 21.7 Domain Interior Problem Caused By Database Surface Parametrization

The simple fix to this problem is to elliptically refine this domain.

1. Run Solver **Structured**

2. Select domain UTML.

3. **Done**

4. Elliptic Slvr **Run N Iters**

5. `30`

6. **Done - Save**

There are only four connectors left to create: <u>OD</u>, <u>RJ</u>, <u>AK</u>, and <u>IN</u>. Go ahead and create them now on your own as **2 Point Connectors**.

Turn off display of all the domain interior grids to reduce the clutter on the screen.

1. **Disp**

2. Edit Dspla **Dom**

3. Pick **All**

4. **Done**

5. **Off**

6. **Done - Apply Atts**

7. **Abort**

8. **Done**

Now your Display window should look like Figure 21.8.
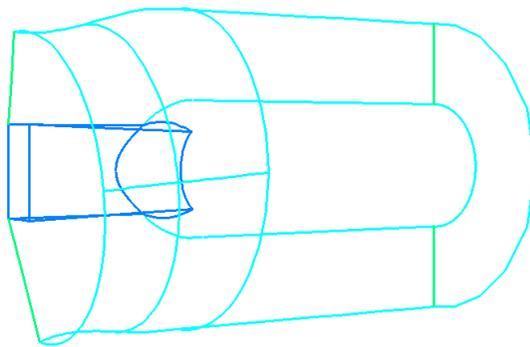
Figure 21.8 All Connectors for the Two Blocks

We have four domains left to create, so finish them now. We will redimension a couple connectors before starting.

1. **Doms**

2. **Redimension Grids**

3. Select connector <u>PQ</u>.

4. `11`

5. Select connector <u>QR</u>.

6. 20

7. **Done-Apply New Dims**

8. **Create**

9. **Assemble Edges**

10. Make sure **Auto Next Edge** and **Auto Complete** are toggled on.

11. Pick connector JR.

12. Pick connector RN. Gridgen will complete domain JRNI.

13. Toggle off **Auto Next Edge**.

14. Pick connector DO.

15. **Next Edge**

16. Pick connector OS.

17. Pick connector SK.

18. **Next Edge**

19. Pick connector KA.

20. Pick connector AB. Gridgen will complete the domain.

21. Pick connector DO.

22. **Next Edge**

23. Pick connector OP.

24. Pick connector PQ.

25. Pick connector QR.

26. **Next Edge**

27. Pick connector RJ. Gridgen will complete the upper symmetry domain.

28. Pick connector AK.

29. **Next Edge**

30. Pick connector KL.

31. Pick connector LM.

32. Pick connector MN.

33. **Next Edge**

34. Pick connector NI. Gridgen will complete the lower symmetry domain.

35. **Done**

The grid spacing changes abruptly between some of the domains. Reduce the abruptness using **Set Δs Vals**.

1. **Cons**

2. **Set Δs Vals**

3. Select the end of connector PQ at P.

4. Select the end of connector LM at L.

5. Select the end of connector HJ at H.

6. Select the end of connector EI at E.

7. **Done**

8. `0.5`

9. **Abort**

This will give you a smoother variation in grid spacing as shown in Figure 21.9. This also affects the two domains on the centerbody since they include connectors HJ and EI. Use the structured solver to reinitialize them using **Initialize via TFI**, and then use the elliptic solver for about 20 iterations with the attributes still set the way they were previously.
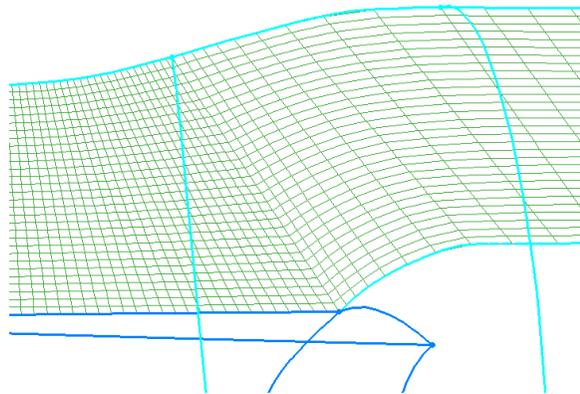
Figure 21.9 Use Set Δs Vals Command to Change Grid Clustering

You can improve the grid distribution on the symmetry plane domains as well by elliptically refining them using the default solver attributes.

1. **Doms**

2. Run Solver **Structured**

3. Select the two domains on the plane of symmetry.

4. **Done**

5. Elliptic Slvr **Run N Iters**

6. `30`

7. **Done - Save**

You can see in Figure 21.10 (and in your Display window) that the default settings in the elliptic PDE solver try to make the grid lines orthogonal to the domain edges. This is usually a desirable property.
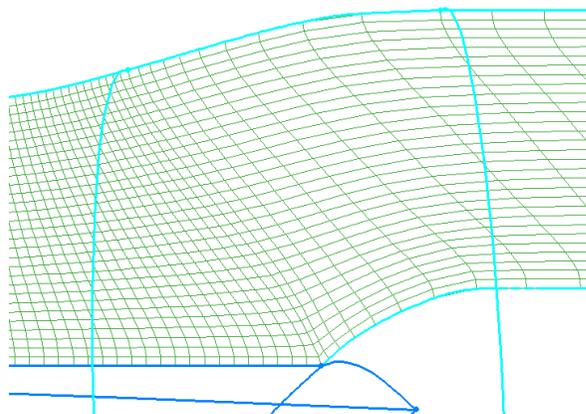
Figure 21.10 Symmetry Domain After Running the Elliptic PDE Solver

You can also improve the two inflow plane domains using elliptic refinement. This time use the floating boundary condition to let the connectors between the domains move.

1. Run Solver **Structured**

2. Select the two inflow plane domains.

3. **Done**

4. **Set Solver Attribs**

5. Apply to **All**

6. **Solver Bound. Conds.**

7. **Floating**

8. **Done Setting Attributes**

9. Elliptic Slvr **Run**

10. Let the solver run for approximately 100 iterations.

11. **Done - Save**

Now you are ready to build the second block.

1. **Blks**

2. **Create**

3. **Assemble Faces**

4. **Add 1st Face**

5. Pick domain OPQRJHD.

6. **Save the Face**

7. Pick domain RNIJ.

8. **Save the Face**

9. Pick domain AEINMLK.

10. **Save the Face**

11. Pick domain <u>HJIEFG</u>.

12. Pick domain <u>DHGC</u>.

13. Pick domain <u>CGFB</u>.

14. Pick domain <u>AEFB</u>.

15. **Save the Face**

16. Pick domain <u>OSKABCD</u>.

17. **Save the Face**

18. Pick domain <u>QRNM</u>.

19. Pick domain <u>PQTU</u>.

20. Pick domain <u>UTML</u>.

21. Pick domain <u>OPUS</u>.

22. Pick domain <u>SULK</u>.

23. **Save the Face**

24. **Done-Save Blocks**

The block is complete and you have completed this tutorial! Save a Gridgen file before doing anything else. After that you can examine the two blocks you have created. See if you can think of any ways to improve this grid.

8. LDI Combustor: Advanced Structured Grid

8.1 Introduction

This tutorial will give you a guided tour of the Gridgen process for creating a grid for a lean-direct injection (LDI) combustor with discrete jet swirlers.

The original tutorial was provided by Dr. S. L. (Jason) Yang and Chen Yen Teo, Mechanical Engineering - Engineering Mechanics Department, Michigan Technological University.

8.2 Topics Covered

This sample session is a short introduction to using Gridgen. Basic Gridgen skills you will learn are:

- Creating database constrained connectors using **On DB Ents** command
- Creating connectors using **2 Point Connector** command
- Creating domains using **Assemble Edges**
- **Copy** and **Rotate** topology
- Elliptically smooth domains using Run Solver **Structured** command
- Use the **Extrude** command to create blocks
- Use the **ReSpecify** ξ,η,ζ to align the computational axes

8.3 Background

We encourage you to complete all previous tutorials in the Gridgen User Manual before attempting this one, since it covers advanced topics and assumes you already understand the basics covered in previous Gridgen tutorials. You should also read Section 12 and Section 13 to familiarize yourself with the advanced topics covered in this tutorial.

8.4 Geometry

The multi-point LDI combustor as shown in Figure 22.1 has a rectangular inlet section 0.5 inches long, and is upstream of nine groups of swirlers arranged in a 3 by 3 pattern. Each group of swirlers comprises eight passages, all co-rotated at 35° in the tangential direction. A 7.5 inches long rectangular burner duct is downstream of swirler section. The combustor has a cross section of 3 inches by 3 inches, as indicated by a square box placed around the swirlers.
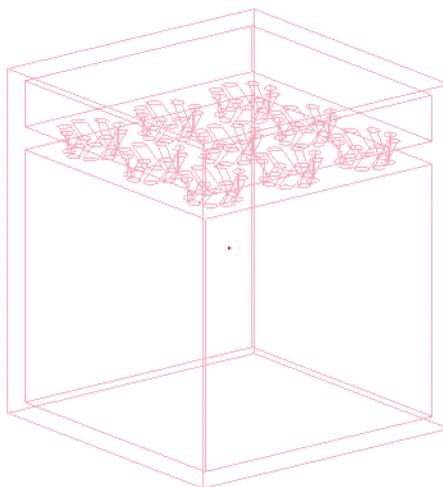
Figure 22.1 The Multi-Point LDI Combustor Geometry

For simplicity, you do not have to model the whole combustor since the geometry can be considered as a grouping of nine sets of identical discrete combustors.

The topology is divided into a single combustor section consisting of an inlet section, eight passages, and a burner section as shown in Figure 22.2. The blocks' I-, J-, and K-directed grid lines will align in the positive x-, y-, and z-directions, respectively. This tutorial will guide you through creating the inlet section block, and the passages but leave the burner section for your own practice.
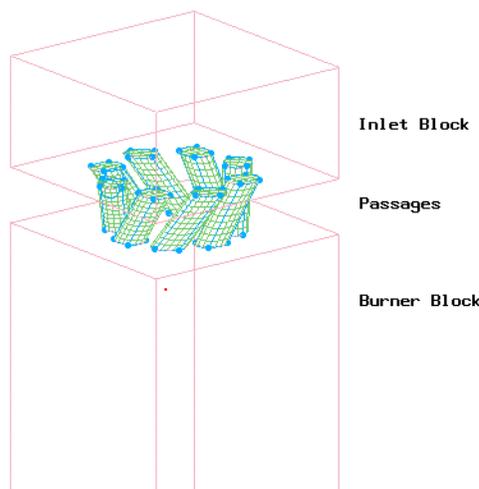
Figure 22.2 A Single Combustor Divided into Ten Blocks

8.5 Getting Started

Start Gridgen by typing `gridgen` on Unix workstations or by simply double-clicking the Gridgen shortcut on your PC desktop. For this tutorial, we will not be performing any analysis using this grid so it will not be necessary to select a particular solver from the Analysis Software menu.

8.6 Importing a Database

You will need to import a database file for this geometry. This will normally be located under *gridgen_home_path*/ `examples/tutorial/ldi`. Load it into Gridgen as follows, starting from the MAIN MENU:

1. **Database**

2. **Import**

3. Select `ldi.dba` from the Browser window.

4. Open

5. Done

6. Done

After importing the file, the Display window should look like Figure 22.3.
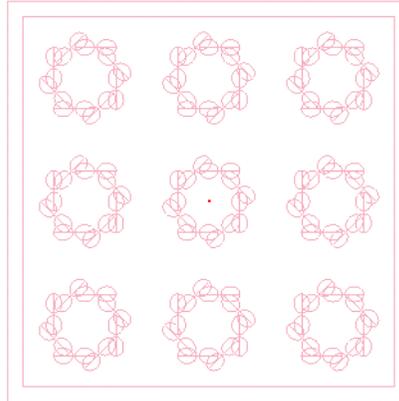
Figure 22.3 Default View of the LDI Combustor Database

8.7 Setting Defaults

Since you will be creating a structured grid, you will begin the process by first setting up default values for the number of grid points and connector tolerances. From the MAIN MENU:

1. Defaults

2. Con Dim **dimen**

3. 5

4. Tolerances

5. Connector

6. 0.01

7. Done

8. Done

You have now set up the grid attributes which will be used throughout this session, unless otherwise stated.

8.8 Displaying Swirler Passages

You will only create a one inch square grid for the combustor at the lower-left corner. To simplify the geometry we want to only display the database curves in the area of the lower left corner. From the MAIN MENU:

1. Database

2. En/Disable

3. Pick **in Box**

4. In the Display window, draw a box around the swirler passages in the lower left corner.

5. Pick **Tog**

6. Done

7. Done

Press the keyboard hotkey *r* to reset the view limits of the Display window. The database entities of the eight passages are shown in Figure 22.4 on the next page.
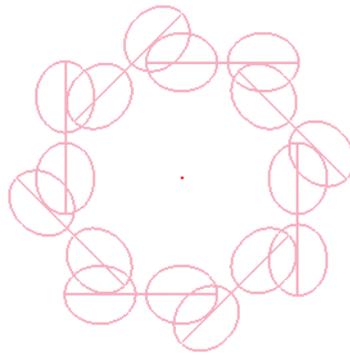
Figure 22.4 Default View of the Swirler Passages

8.9   Connectors at the Inlet Plane of the Passages

You will be creating a multi-block structured grid for the inlet block of the burner geometry. For the purposes of this tutorial, the chosen topology on the inlet plane to the swirler passages, is shown in Figure 22.5. The passage entity 9 has been labeled for the first step in the grid construction process.
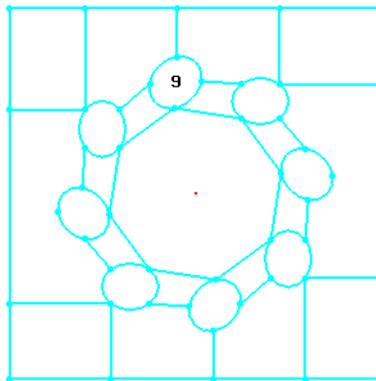
Figure 22.5 Multi-block Grid Topology for the Inlet Block

Now that the topology for the blocking system has been chosen, you will begin by creating connectors on the swirler passage entity 9. These will then be copied and rotated to form the connectors for the other seven passages. From the MAIN MENU:

1. **Connectors**

2. **Create**

3. **On DB Entities**

4.   Pick the database curves for the passage entity 9.

5. **Done**

The Display window should look like Figure 22.6.
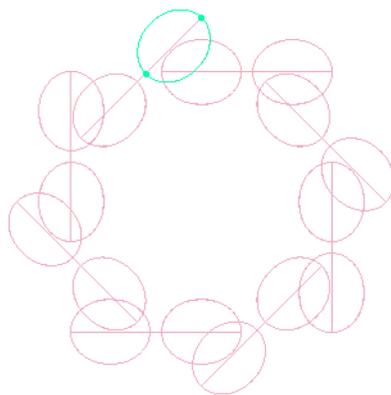
Figure 22.6 Connectors on Passage Entity 9

You must now split the connectors to create the four edges defining the domain of the structured grid for the swirler passage. From the CONNECTOR COMMANDS menu:

1. **Modify**

2. Select either connector.

3. **Done**

4. **Split**

5. ❍ **to Prev Grid Pt** ●

6. **Split at Cursor** ❍

7. ❍ **to Next Grid Pt** ●

8. **Split at Cursor** ❍

9. Press *Enter* on the keyboard, acknowledging the warning message.

10. Repeat steps 1 - 9 for the other connector of passage entity 9.

Zoom into the area around the passage entity 9 and the Display window should look like Figure 22.7 on the next page.
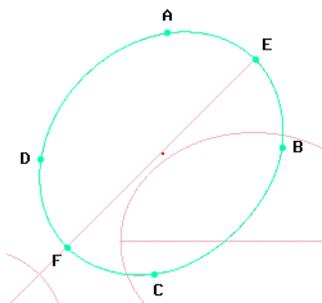
Figure 22.7 Passage Entity 9 Connectors

You now need to join the connectors <u>AE</u>, <u>EB</u> to define the edge <u>AB</u> and segment connectors <u>CF</u>, <u>FD</u> to define the edge <u>CD</u>. From the CONNECTOR COMMANDS menu:

1. **Modify**

2. Pick connector <u>AE</u>.

3. **Done**

4. **Join**

5. Pick connector <u>EB</u>.

6. Abort

7. Done - Replace Connectors

8. Repeat steps 1 - 7 to join segment connectors <u>CF</u> with <u>FD</u> to form the connector <u>CD</u>.

This completes the connector creation for the passage entity 9. The Display window is shown in Figure 22.8.
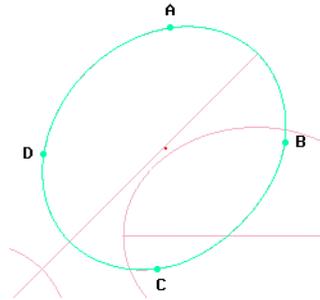
Figure 22.8 Joined Connectors on Passage Entity 9

You now need to redimension the connectors. From the CONNECTOR COMMANDS menu:

1. Modify

2. Pick **All**

3. Done

4. ReDimension

5. From **keybrd**

6. 5

7. Done - ReDimension

8. Done - Replace Connectors

9. Maintain Linkages

Since the connectors for the other seven passages are identical to those of passage entity 9, you will use the copy and rotate commands to create them. From the CONNECTOR COMMANDS menu:

1. Copy

2. Pick **All**

3. Done

4. Abort - Don't Translate

5. Rotate

6. Enter Axis_1 via Keybrd

7. 0.5, -2.5, 0

8. Use Z-directed Axis

9. Enter Rotation Angle

10. 45

11. Done - Rotate

12. Done-Save Connectors

The Display window should look like Figure 22.9 on the following page.
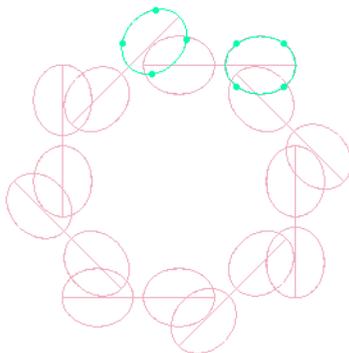
Figure 22.9 Passage 9 Connectors Copied and Rotated

Use the same commands to create the other passage connectors. From the CONNECTOR COMMANDS menu:

1. **Copy**

2. Pick **All**

3. **Done**

4. **Abort - Don't Translate**

5. **Rotate**

6. **Enter Axis_1 via Keybrd**

7. `0.5, -2.5, 0`

8. **Use Z-directed Axis**

9. **Enter Rotation Angle**

10. `.90`

11. **Done - Rotate**

12. **Done-Save Connectors**

The Display window should look like Figure 22.10 on the following page.
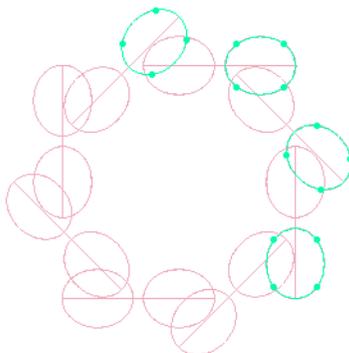
Figure 22.10 Copied and Rotated Connectors at Passage Inlets

Finally, create the last four sets of connectors for the swirler passages. From the CONNECTOR COMMANDS menu:

1. **Copy**

2. Pick **All**

3. **Done**

4. Abort - Don't Translate

5. Rotate

6. Enter Axis_1 via Keybrd

7. 0.5, -2.5, 0

8. Use Z-directed Axis

9. Enter Rotation Angle

10.180

11. Done - Rotate

12. Done-Save Connectors

This completes the connector creation for the inlets to the swirler passages. The finished topology is shown in Figure 22.11. The labeling convention is also shown for the next step in the construction process.
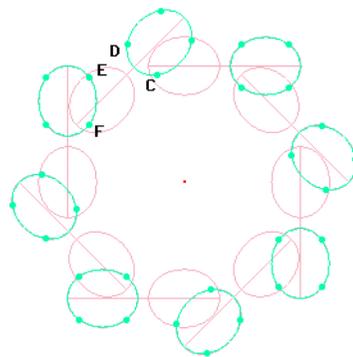
Figure 22.11 Connectors at the Inlet Plane of Swirler Passages

8.10   Domain Topology Between Passages

You will now create the connectors <u>CF</u> and <u>DE</u>, which define the topology between the swirler passages. From the CON-NECTOR COMMANDS menu:

1. Create

2. 2 Point Connectors

3. Position the 3D cursor over node C.

4. Add CP □ **by Picking**

5. Position the 3D cursor over node F.

6. Add CP □ **by Picking**

7. Position the 3D cursor over node D.

8. Add CP □ **by Picking**

9. Position the 3D cursor over node E.

10. Add CP □ **by Picking**

11. Done

12. Done Creating Conns

Redimension connectors <u>CF</u> and <u>DE</u>. You should still be in the CONNECTOR COMMANDS menu.

1. ReDimension Grids

2. Select connector <u>CF</u>.

3. 6

4. Select connector <u>DE</u>.

5. 6

6. Done - Apply New Dims

7. Done

The Display window with connectors <u>CF</u> and <u>DE</u> is shown in Figure 22.12.
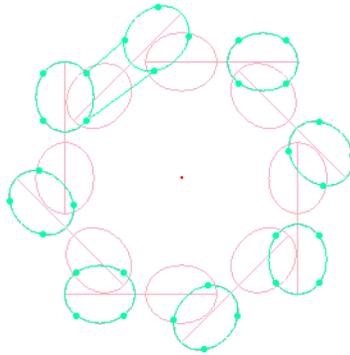
Figure 22.12 Connectors Between Swirler Passages

You will create a domain bound by the connectors <u>DC</u>, <u>CF</u>, <u>FE</u> and <u>ED</u>. From the MAIN MENU:

1. Domains

2. Create

3. Assemble Edges

4. Turn on the **Auto Next Edge** toggle.

5. Pick connector <u>DC</u> followed by <u>CF</u>.

6. Done

The Display window should shown in Figure 22.13 on the following page.
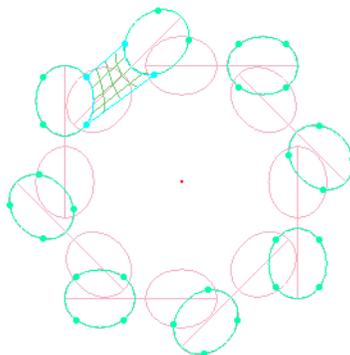
Figure 22.13 Domain Topology Between Passages

You now copy and rotate the domain you have just created in much the same way as Section 8.9 in the tutorial. From the DOMAIN COMMANDS menu:

1. Copy

2. Pick **All**

3. **Done**

4. **Abort - Don't Translate**

5. **Rotate**

6. **Enter Axis_1 via Keybrd**

7. `0.5, -2.5, 0`

8. **Use Z-directed Axis**

9. **Enter Rotation Angle**

10. `45`

11. **Done - Rotate**

12. **Save Domains**

13. **Done**

Repeat steps 1 - 13 for 90° and 180° angles of rotation, each time selecting all domains. The Display window should look like Figure 22.14 on the next page.
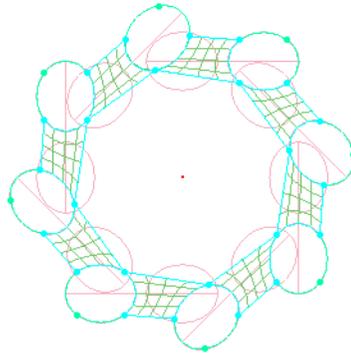
Figure 22.14 Copied Domains Between Passages

You are now ready to proceed to the next phase of the grid construction process.

8.11   Creating Boundary Topology

You are now going to create the connectors defining the topology around the swirler passages and the burner boundary. We will be referring to the labeling conventions shown in Figure 22.15.
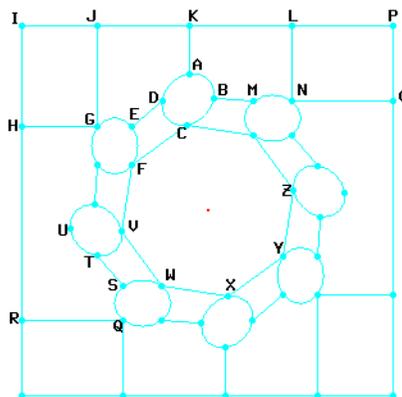
Figure 22.15 Labeling Convention for Boundary Domains

First set a new default number of grid points from the MAIN MENU:

1. **Defaults**

2. Con Dim **dimen**

3. `10`

4. **Done**

Now begin creating the connectors:

1. **Connectors**

2. **Create**

3. **2 Point Connectors**

Connector GH:

1. Place the 3D cursor over point G.

2. Add CP □ **by Picking**

3. Add CP □ **via Keybrd**

4. `0`

Connector HI:

1. Place the 3D cursor over point H.

2. Add CP □ **by Picking**

3. Add CP □ **via Keybrd**

4. `0,-2`

Connector GJ:

1. Place the 3D cursor over point G.

2. Add CP □ **by Picking**

3. Add CP □ **via Keybrd**

4. `, -2`

Connector IJ to close domain GHIJ:

1. Place the 3D cursor over point I.

2. Add CP □ **by Picking**

3. Place the 3D cursor over point J.

4. Add CP □ **by Picking**

Connector AK:

1. Place the 3D cursor over point A.

2. Add CP □ **by Picking**

3. Add CP □ **via Keybrd**

4. `, -2`

Connector JK to close domain ADEFJK:

1. Place the 3D cursor over point J.
2. Add CP ☐ **by Picking**
3. Place the 3D cursor over point K.
4. Add CP ☐ **by Picking**

Connector NL:

1. Place the 3D cursor over point N.
2. Add CP ☐ **by Picking**
3. Add CP ☐ **via Keybrd**
4. , -2

Connector KL to close Domain AKLNMB:

1. Place the 3D cursor over point K.
2. Add CP ☐ **by Picking**
3. Place the 3D cursor over point L.
4. Add CP ☐ **by Picking**

Connector NO:

1. Place the 3D cursor over point N.
2. Add CP ☐ **by Picking**
3. Add CP ☐ **via Keybrd**
4. 1

Connector OP:

1. Place the 3D cursor over point O.
2. Add CP ☐ **by Picking**
3. Add CP ☐ **via Keybrd**
4. , -2

Connector LP to close domain LNOP:

1. Place the 3D cursor over point L.
2. Add CP ☐ **by Picking**
3. Place the 3D cursor over point P.
4. Add CP ☐ **by Picking**

Connector RQ:

1. Place the 3D cursor over point Q.
2. Add CP ☐ **by Picking**
3. Add CP ☐ **via Keybrd**
4. 0

Connector HR:

1. Place the 3D cursor over point H.

2. Add CP □ **by Picking**

3. Place the 3D cursor over point R.

4. Add CP □ **by Picking**

5. **Done**

6. **Done Creating Conns**

You can turn off the database entities to view the topology from the DISPLAY COMMANDS menu. The Display window should look like Figure 22.16.
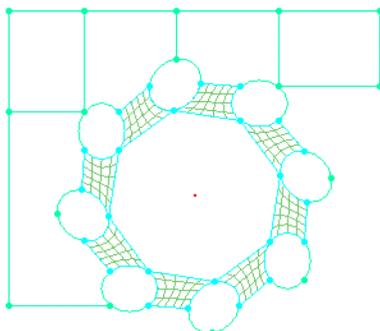
Figure 22.16 Connectors Forming the Boundary Topology

Redimension the connectors to ensure a structured grid can be obtained from the connectors. From the CONNECTOR COMMANDS menu:

1. **Redimension Grids**

2. Select connector JK.

3. `14`

4. Select Connector KL.

5. `14`

6. Select Connector HR.

7. `27`

8. **Done-Apply New Dims**

9. **Done**

8.12   Structured Domains on Inlet Plane

You will now tell Gridgen how to make a surface grid from the connectors you just created. From the MAIN MENU:

1. **Domains**

2. **Create**

3. **Assemble Edges**

4. Turn on the **Auto Next Edge** toggle.

Domain GHIJ:

1. Pick connector GH.

2. Pick connector HI.

Domain <u>ADEGJK</u>:

1. Pick connector <u>GJ</u>.

2. Pick connector <u>JK</u>.

3. Pick connector <u>AD</u>.

Domain <u>ABMNLK</u>:

1. Pick connector <u>AK</u>.

2. Pick connector <u>KL</u>.

3. Pick connector <u>LN</u>.

4. Pick connector <u>NM</u>.

Domain <u>NOPL</u>.

1. Pick Connector <u>NO</u>

2. Pick Connector <u>OP</u>.

Domain <u>GHRQSTU</u>:

1. Pick connector <u>GH</u>.

2. Pick connector <u>HR</u>.

3. Connector <u>RQ</u> is picked automatically.

4. Pick connector <u>QS</u>, followed by <u>ST</u>, then finally <u>TU</u>.

5. Done

Gridgen completes the domain and initializes the surface grid using transfinite interpolation (TFI). Your Display window should look like Figure 22.17.
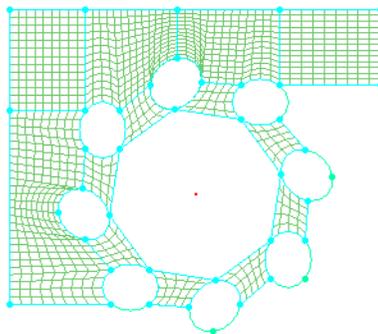
Figure 22.17 Domains are Initialized by Transfinite Interpolation

You now need to create the domain at the center of the topology bounded by the connectors <u>CF</u>, <u>FV</u>, <u>VW</u>, <u>WX</u>, <u>XY</u>, and <u>YZ</u>. From the DOMAIN COMMANDS menu:

1. Create

2. **Assemble Edges**

3. Turn off the **Auto Next Edge** toggle.

Domain <u>CFVWXYZ</u>:

1. Pick the connector <u>CF</u> followed by <u>FV</u>.

2. Next Edge

3. Pick the connector <u>VW</u> followed by <u>WX</u>.

4. Next Edge

5. Pick the connector <u>XY</u> followed by <u>YZ</u>.

6. Done

The Display window should look like Figure 22.18.
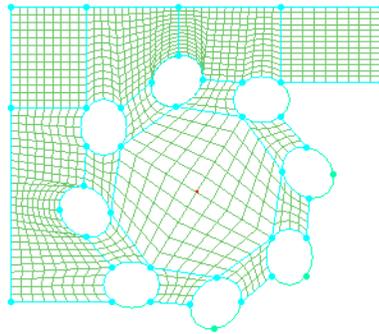
Figure 22.18 Center Domain Topology

Use the copy and rotate commands to create the opposite boundary domains on the inlet plane. From the DOMAIN COMMANDS menu:

1. Copy

2. Pick domains <u>GHIJ</u>, <u>ADEGJK</u>, <u>ABMNLK</u>, <u>NOPL</u> and <u>GHRQSTU</u>.

3. Done

4. Abort - Don't Translate

5. Rotate

6. Enter Axis_1 via Keybrd

7. 0.5, -2.5, 0

8. Use X-directed Axis

9. Enter Rotation Angle

10.180

11. Done - Rotate

12. Save Domains

Your Display window should look like Figure 22.19.
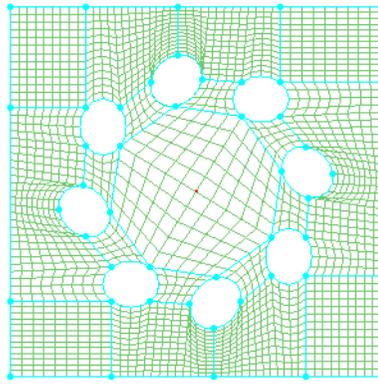
Figure 22.19 Domain Topology on Inlet Plane to Swirler Passages

8.13   *Running the Elliptic PDE Solver*

The surface grid created automatically by TFI can often be improved by applying an elliptic PDE-based smoothing to the grid. Gridgen offers a variety of elliptic PDE methods with various controls for boundary spacing, orthogonality, and smoothness.

You will run the elliptic solver from the DOMAIN COMMANDS menu:

1.   Run Solver **Structured**

2.   Pick **All**

3.   Done

4.   Set Solver Attributes

5.   Apply to **All**

6.   Background Cont Funcs

7.   LaPlace

8.   Foreground Cont Funcs

9.   Form **Off**

10. Set Other Attributes

You will finally set the floating boundary condition on all domains as shown in Figure 22.20 on the following page.
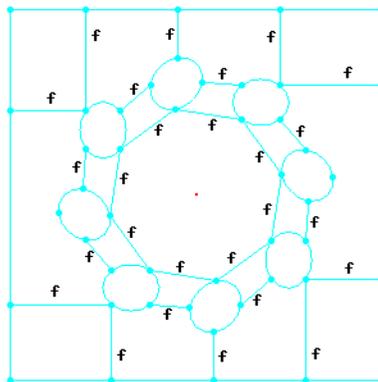
Figure 22.20 Boundary Conditions for Elliptical PDE Solver

You should still be in the SET STRUCTURED SOLVER ATTS menu:

1.   Solver Bound. Conds.

2. Floating

3. Done Setting Attributes

You now will run the elliptic solver.

1. Elliptic Slvr **Run N Iters**

2. 50

This runs the elliptic solver with the attributes that you have previously set. As the elliptic solver iterates, the grid display will be updated and you can monitor the progress of the surface grid. You may stop the solver by pressing the following button:

1. Done - Save

2. Done

This is a small grid, so it will not take long to reach a state of equilibrium. When done the grid looks something like the one in Figure 22.21.
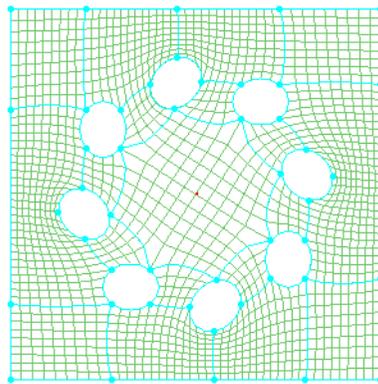
Figure 22.21 Inlet Plane after Running the Elliptic PDE Solver

8.14 Creating Domains for the Passages

You can now proceed with the construction of the domains forming the inlet topology of the passages. The labeling convention used is shown in Figure 22.22.
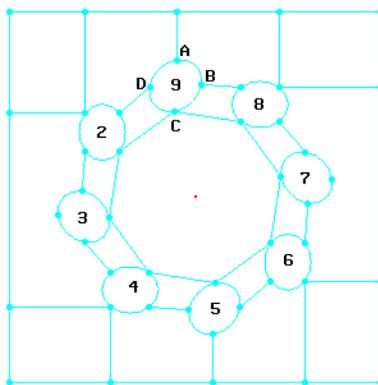
Figure 22.22 Labeling of Inlet Domains to Swirler Passages

From the DOMAIN COMMANDS menu:

1. Domains

2. Create

3. Assemble Edges

4. Turn on the **Auto Next Edge** toggle.

Domain on passage entity 9:

1. Pick connector <u>AB</u>.

2. Pick connector <u>BC</u>.

3. Done

The Display window should now look like Figure 22.23.
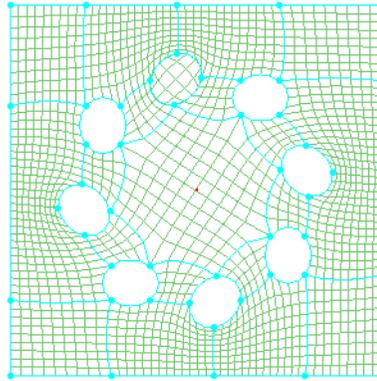
Figure 22.23 Inlet Domain on Passage Entity 9

You now copy and rotate the inlet passage domain to complete the structured mesh on the inlet plane on the Inlet section. From the DOMAIN COMMANDS menu:

1. Copy

2. Pick the passage entity 9 inlet domain.

3. Done

4. Abort - Don't Translate

5. Rotate

6. Enter Axis_1 via Keybrd

7. `0.5, -2.5, 0`

8. Use Z-directed Axis

9. Enter Rotation Angle

10. `45`

11. Done - Rotate

12. Save Domains

Repeat steps 1 - 13 for 90° and 180° angles of rotation, each time selecting the newly copied domains. It is hoped that your Display window should look like Figure 22.24. You want to run the elliptical solver for the passage domains to improve the quality of the surface grid.
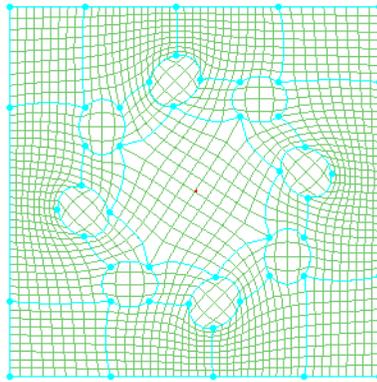
Figure 22.24 Domains on the Inlets to the Swirler Passages

8.15 Extruding Inlet Block

You can now proceed with the construction of the inlet block. You will extrude the inlet plane domains to create a 3D structured block in the negative z-direction. From the MAIN MENU:

1. **Blocks**

2. **Create**

3. Extrude **From Doms**

4. Pick **All**

5. **Done**

6. Type **Tran**

7. **Set Attributes**

8. **Total Translation Dist**

9. **Enter via Keybd**

10. 0.5

11. **Set Translation Vector**

12. **Enter Vector via Keybd**

13. 0, 0, –1

14. **Done**

15. **Run N**

16. 18

17. **Done**

You may want to reorient the Display window as shown in Figure 22.25.
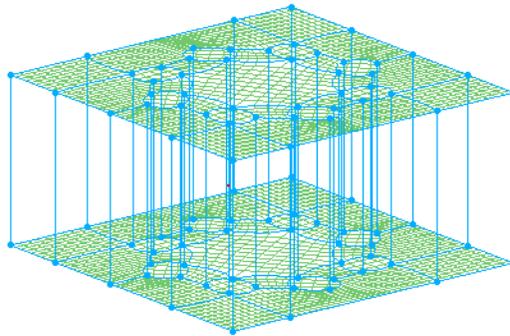
Figure 22.25 Inlet Block of Burner

8.16   Creating Blocks for the Passages

You now need to create the blocks for the eight elliptical passages. This will be done again using the extrusion command.
Before you do this, turn on the swirler passages database entities from the DISPLAY COMMANDS menu, then from the
MAIN MENU:

1. **Blocks**

2. **Create**

3. Extrude **From Doms**

4. Pick the domain of passage entity 9.

5. **Done**

6. Type **Tran**

7. **Set Attributes**

8. **Total Translation Dist**

You are now going to pick the two endpoints which define the distance and vector direction of the extrusion operation as
shown in Figure 22.26 on the following page.
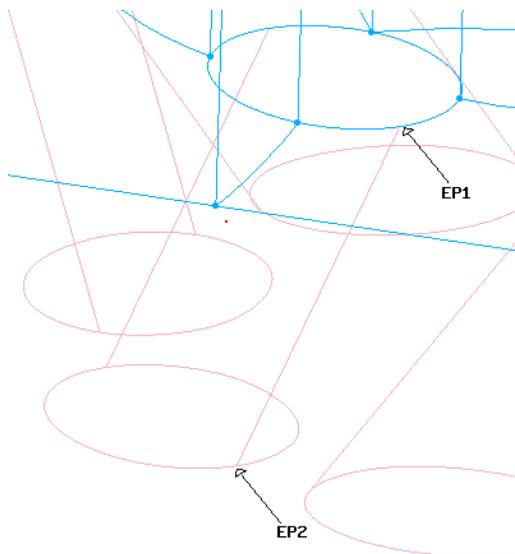
Figure 22.26 Endpoints for Translation Extrusion Operation

From the TRANSLATION DISTANCE menu:

1. **Pick Endpoints**

2. Pick Endpoint EP1.

3. Pick Endpoint EP2.

4. Done Setting Distance

5. **Set Translation Vector**

6. **Pick Endpoints**

7. Pick Endpoint EP1.

8. Pick Endpoint EP2.

9. Done Setting Vector

10. Done

11. Run N

12.10

13. Done

Zoom and reorient the view around passage entity 9, the Display window should now look something like Figure 22.27 on the following page.
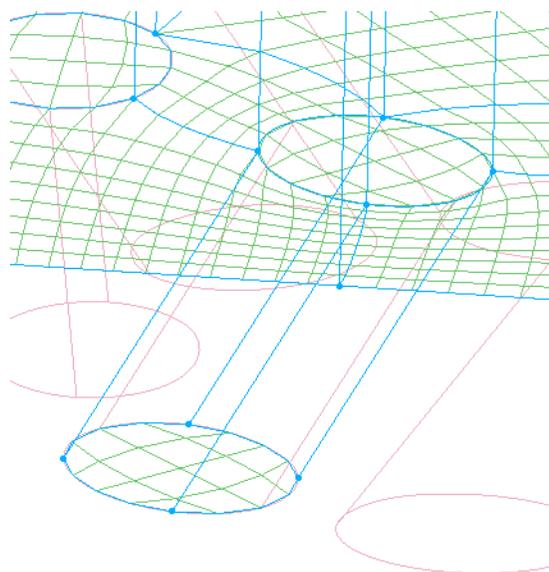
Figure 22.27 Extruded Block for Passage Entity 9

You will now copy and rotate the passage entity 9 block:

1. Copy

2. Pick block of passage 9.

3. Done

4. Abort - Don't Translate

5. Rotate

6. Enter Axis_1 via Keybrd

7. 0.5, -2.5, 0

8. Use Z-directed Axis

9. Enter Rotation Angle

10.45

11. Done - Rotate

12. Done - Save Block

Repeat steps 1 - 13 for 90° and 180° angles of rotation, each time selecting the newly copied domains. You may want to turn on the Hidden Line Removed toggle from the DISPLAY COMMANDS menu and reorient the Display window as shown in Figure 22.28.
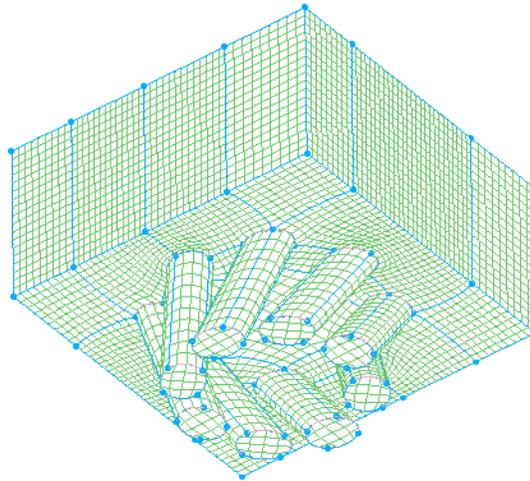
Figure 22.28 Passages Created Using Copy and Rotate

You now need to align the computational axes with the model axes. In all of the blocks you want ξ to run in the x direction, η in y direction, and ζ in z direction. From the BLOCK COMMANDS menu:

1. **Modify**

2. Pick any rectangular block in the Inlet section.

3. **Done**

4. **ReSpecify** ξ, η, ζ

5. Select one of the block edges that run in the positive x direction.

6. ξ **direction**

7. Repeat steps 5 and 6 for the y direction and z direction, pressing the η, and ζ **direction** buttons accordingly.

8. **Done-Replace Blocks**

9. **Modify**

10. Pick **All**

11. **Done**

12. **Align** ξ, η, ζ

13. Pick the rectangular block in step 2.

14. **Done-Replace Blocks**

15. **Done**

8.17 Tutorial Wrap-up

This completes the Gridgen tutorial. You may proceed to generate the grid for burner section in a similar way. If you want to save this grid:

1. Select **I/O** from the Quick Keys.

2. Gridgen **Export**

3. Type In... **Name**

4. Enter the file name `top_swirler.gg`

5. Done

When you are done, use **Quit Gridgen** from the MAIN MENU to exit the program.

9. Pierced Elbow: Basic Unstructured Grid

9.1 Introduction

This tutorial will give you a guided tour of the Gridgen process for creating a simple single tetrahedral volume grid for the flow field inside a 90° degree pierced elbow. We encourage you to complete this tutorial and then begin making your own simple grids.

9.2 Topics Covered

This tutorial covers basic and advanced features of Gridgen. Gridgen skills you will learn during this tutorial session are:

- setting **Defaults** appropriate for an unstructured grid

- creating various database entities using the DB CREATION menu

- creating connectors with the **2 Point Connectors** and DB cnstr **Line** commands

- projecting connectors onto a database surface using **Modify**

- creating unstructured domains using **Assemble Edges**

- creating an unstructured block using the Pick **All** feature

- initializing the tetrahedral volume grid with the Run Solver **Unstrctrd**

- using **Examine** to visualize the interior of the volume grid

- setting boundary conditions for flow solver export with **Set BCs**

9.3 Background

You should work through the 2D Bump on a Wall tutorial (Section 2) to get a feel for Gridgen basics. You should also read Section 2 of the User Manual to familiarize yourself with the operation of the Gridgen user interface before attempting to run this tutorial.

9.4 Geometry

The pierced elbow configuration you will use for this tutorial is shown in Figure 22.1. Presumably this grid will be used with a computational fluid dynamics code to compute the flowfield inside the elbow. The CFD solver is not part of Gridgen.
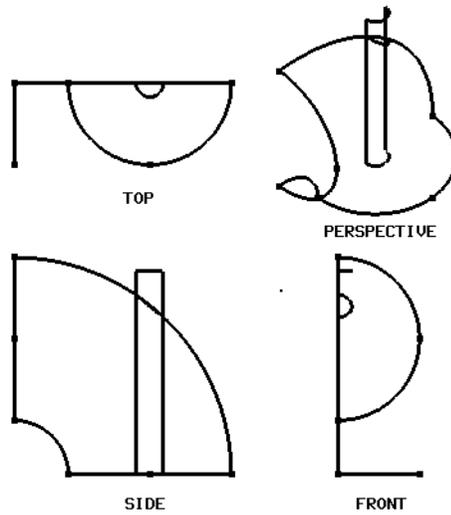
Figure 22.1 The Pierced Elbow Configuration To Be Gridded

The grid you will generate consists of one block filling the entire elbow excluding the region of the piercing shaft as shown in Figure 22.2. For clarity the image only shows a surface grid for the elbow itself, and the shaft surface grid has been shaded. Symmetry plane surface grids and inflow/outflow surface grids are not displayed. The choices of blocking system and grid topology are up to you; this particular blocking system is only one of many that could be used for this elbow geometry. The requirements of the particular analysis software you are using should be considered when planning any grid.
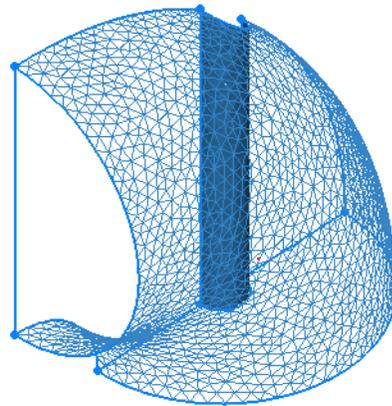
Figure 22.2 The Single Block System To Be Used In the Elbow

Now that a topology for the blocking system has been chosen we can begin creating the grid. For convenience we have assigned a naming convention to important points in the grid. The points' labels and coordinate values are shown in Figure 22.3 and the following table, respectively. Note that the layout of this geometry has not been selected for any other reason than to produce the desired pierced elbow shapes of our eventual grid system.
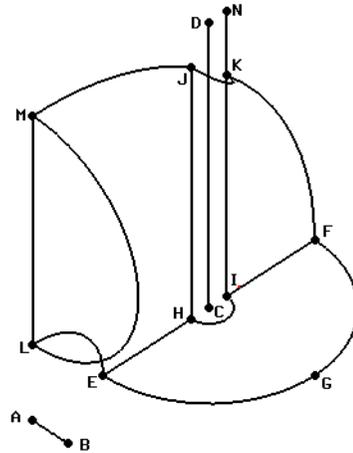
Figure 22.3 Labeled Points For the Elbow

Elbow Coordinates

Label	X	Y	Z
A	0	0	0
B	0	0	10
C	50	0	0
D	50	75	0
E	20	0	0
F	80	0	0
G	50	0	30
I	55	0	0

9.5   Getting Started

Begin by starting Gridgen, either by typing the string `gridgen` on Unix workstations or simply double-clicking the Gridgen shortcut on your PC's desktop. The first step in running Gridgen is to select the software (of the User Manual) to be used in the analysis. This will ensure that the grid generated by Gridgen is consistent with the capabilities of the analysis software. For this analysis we will be using the **Fluent** flow solver (by default Gridgen selects the **generic** solver). To select **Fluent** do the following starting from the MAIN MENU:

1. **Analysis S/W**

2. **Select Analysis S/W**

3. **3D**

4. **Fluent**

5. **Next Page**

6. **Done**

7. **Done**

Use of the **3D** button above tells Gridgen that you will be creating a 3D grid. Verify that you have selected the **3D Fluent** flow solver by checking the display in the Status window.

9.6   *Creating Database Entities*

You are now ready to begin creating the database geometry elements that will comprise the foundation of the grid. The process begins with the creation of simple database lines. Then you will create a database arc and follow that with a curve offset. Finally you will create two surfaces of revolution and intersect them.

For the first two database entities you will use the Curve **Line** command, and these lines will later be used as the axes of rotation for two surfaces of revolution. To create the first line A̲B̲ do the following:

1. **Database**

2. **Create**

3. Curve **Line**

4. Add CP □ **via Keybrd**

5. `0, 0, 0`

6. Add CP □ **via Keybrd**

7. `0, 0, 10`

8. **Save DB curve**

Now create the second line, C̲D̲, by the same process:

1. Curve **Line**

2. Add CP □ **via Keybrd**

3. `50, 0, 0`

4. Add CP □ **via Keybrd**

5. `50, 75, 0`

6. **Save DB curve**

You may not be able to see the two lines in the Display window. Go ahead and zoom the model out and rotate to a more isometric view. Use the pad *7*, *8*, and *9* keys to rotate the model in x, y, and z screen coordinates respectively (Section 2.7.4). The *r* hot key will always restore the model to an orientation where the body and screen coordinate systems are aligned if you are not happy with the results of your manipulations. Since the first line A̲B̲ is so small relative to the red rotation axes, you may wish to also move the rotation point to the y-min end of the line C̲D̲. Do this by holding down the ***Ctrl*** key and clicking on the y-min end of line C̲D̲ with the ***right mouse*** button. Now you will create the two entities that will later be revolved about the lines (axes of revolution) just completed. The first entity will form the cross-sectional shape of the elbow, the 180° circular arc E̲G̲F̲:

1. Curve **Circle**

2. Add CP □ **via Keybrd**

3. `20, 0, 0`

4. Add CP □ **via Keybrd**

5. `80, 0, 0`

6. Store CP □ **via Keybrd**

7. `50, 0, 30`

8. **Done – Save DB Curve**

The second entity to be revolved will be a line which you will create using the Curve **Offset** command. Again, from within the DB CREATION menu:

1. Curve **Offset**

After pressing this button you will now see in the Message window the following prompt:

```
PICK GENERATRIX:
Pick the curve to be
offset.
```

For this particular operation, the generatrix refers to the entity from which the new curve will be offset.

2. Select line CD.

The Message window should now display:

```
OFFSET CURVE:
Define how the generatrix
is to be offset.
```

You should keep in mind when doing offsets that the operation is performed in a constant z-plane in screen coordinates (Section 5.3.6 of the User Manual) or more simply stated, in the plane of the screen. If necessary, you can now revert to a view perpendicular to z=0 in body coordinates by simply using the *r* hot key (Section 2.7.4). Now you are ready to continue the offset operation:

3. **Enter Offset Distance**

4. `5`

5. **Save**

Next we will create the two surfaces of revolution. The first surface we will create will be the elbow itself. This will be done by rotating the arc, EGF, about the axis, AB. You may wish to rotate the model to a more isometric view again to make the remaining operations easier. Use the following procedure from within the DB CREATION menu to create the first surface:

1. Surface **Revolution**

2. Select arc EGF.

3. Select line AB.

4. **(Re)Set the Angle**

5. `-90`

6. **Done – Save Surf of Revol**

And now for the piercing shaft use the same procedure:

1. Surface **Revolution**

2. Select line <u>IN</u>.

3. Select line <u>CD</u>.

4. **(Re)Set the Angle**

5. `-180`

6. **Done – Save Surf of Revol**

7. **Done**

The following figure shows all of the entities you have already created, plus the intersection curve which you will compute next.
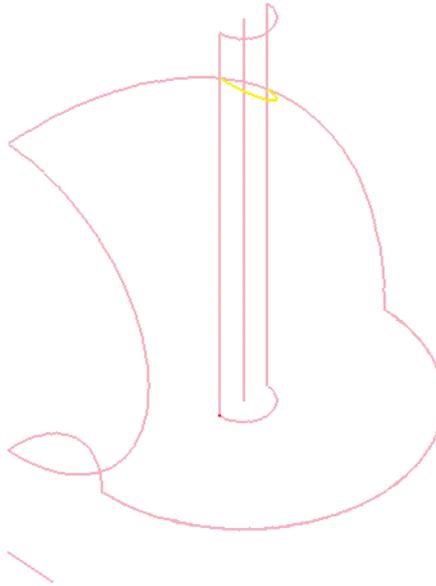
Figure 22.4 A Complete Database Model For the Elbow

To complete the database you will now create the intersection of the two surfaces of revolution. For this procedure Gridgen will put you into Browser mode twice. You will do this to select two groups of entities to intersect with each other. For this tutorial you will only have one entity in each group:

1. **Intersect**

2. **Select Group A**

3. Select the elbow surface of revolution.

4. **Done**

5. **Select Group B**

6. Select the piercing shaft surface of revolution.

7. **Done**

8. **Run**

9. **Done**

10. **Done**

You should now see a yellow database curve at the intersection of the elbow and the shaft.

It is a good idea to save your work frequently so export your database now in Gridgen's composite database file format.

1. **Input/Output**

2. Database **Export**

3. **composite**

4. **Done**

5. Type In... **Name**

6. elbow

Note that Gridgen will automatically append the correct file extension (.dba).

7. **Done**

This completes the database creation portion of this tutorial. Next we will begin creating our connectors.

9.7   *Creating Connectors*

Gridgen's one-dimensional grid elements are referred to as connectors. The connectors will determine the shape of your grid and the basis for its dimension and grid point distribution. Before you begin creating the connectors, you should use the **Defaults** command (Section 3.1.2 of the User Manual) to set an average grid spacing which will be applied to all of the new connectors you create.

1. **Defaults**

2. Con Dim **avg Δs**

3. 3

4. **Done**

Now you are ready to begin connector creation. We will begin with Gridgen's easiest connector creation tool, the **On DB Entities** command (Section 6.4 of the User Manual). This command will simply take all of the database entities that you select and outline them with new connectors. Curves will become single connectors, and surfaces will have a new connector representing each edge of the surface. Begin creating connectors now:

1. **Connectors**

2. **Create**

3. **On DB Entities**

4. Select all of the entities except the first two which are the axes used for the surfaces of revolution.

5. **Done**

A little additional connector clean-up is necessary. The circular connector, <u>MF</u>, needs to be broken at points J and K. This can be done automatically using the Merge command:

1. Merge **Auto**

2. `0.5`

With that done, you should **Delete** the unnecessary connectors lying outside the boundary of the pierced elbow volume. These are the three small connectors on the y-max end of the shaft surface, plus the one connector traversing from J to K on the symmetry plane, for a total of four unneeded connectors.

1. **Delete**

2. Select the four unnecessary connectors.

3. **Done**

4. **Done**

Now finish creating the remaining connectors using Gridgen's **2 Point Connector** command. This command allows you to specify point pairs and automatically saves a new connector with each second point entered. Try it now:

1. **Create**

2. **2 Point Connectors**

Gridgen now goes into connector segment creation mode. The 3D cursor ([Section 2.8](#) of the User Manual) should appear in the Display window as a small white cross. The 3D cursor is positioned by holding down the *right mouse* button and then panning the mouse. The 3D cursor will follow the motion of the mouse. For each connector end point, place the 3D cursor over the desired end point location and use the Add CP □ **by Picking** option in the menu.

3. Place the 3D cursor over point E.

4. Add CP □ **by Picking**

5. Place the 3D cursor over point H.

6. Add CP □ **by Picking**

Gridgen should have automatically saved connector <u>EH</u>. Now continue creating connectors on your own using this procedure for connectors <u>IF</u> and <u>LM</u>. Then return to the CONNECTOR COMMANDS menu:

7. **Done**

8. **Done Creating Conns**

You have successfully created all of the connectors necessary for this tutorial! Now you can go into the DISPLAY COMMANDS menu ([Section 2.9](#) of the User Manual) using **Disp** and toggle on Show **Con GPs** so all of your connector grid points will be displayed. Press **Done** to exit from the DISPLAY COMMANDS menu.

Your model in the Display window should now look like the one shown in Figure 22.5. The remainder of your grid construction will be even easier than what you have done so far. Next you will create your six domains before moving on to creating the block.

At this point you might use Gridgen's Quick Save command to save your grid to a file. Simply press the Quick Key ([Section 2.3.6](#)) marked **Save**; it is one of the twelve small buttons in the three rows at the bottom of the menu. This command exports a Gridgen restart file called "`quicksave.gg`" in the directory in which Gridgen was started by default.
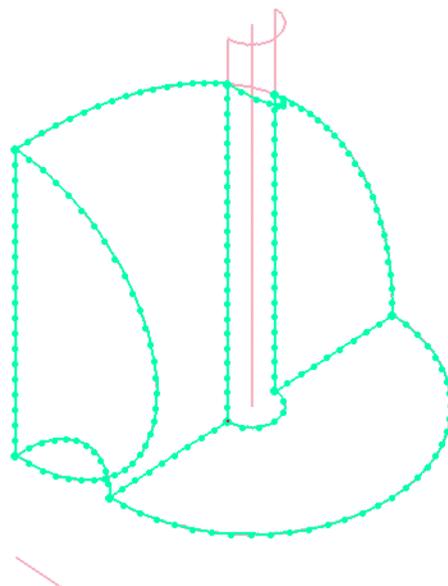
Figure 22.5 All of the Connectors With Grid Points Displayed

9.8   Creating Domains

An important point about domains in Gridgen is that each domain will represent a single boundary condition for your flow solver, which we chose to be **Fluent**. This is important to keep in mind for preparing your grid later to export to your flow solver.   For this tutorial we will be creating triangular mesh domains since our goal is to build an entirely unstructured grid. Building triangulated unstructured domains is much simpler than building structured quadrilateral domains since there are few restrictions on the topology of the domain: it can have any number of edges.

Begin by constructing one of the simplest domains for this grid, the triangular shaped symmetry plane domain, <u>IKF</u>**:**

1. **Doms**
2. **Create**
3. Cell Type **unstructured**
4. **Assemble Edges**
5. Toggle on **Auto Save**
6. Pick in order edge connectors <u>IF</u>**,** <u>FK</u>**,** <u>KI</u>.

Since you turned on the **Auto Save** feature, Gridgen detected automatically that you had completed the loop with the three connectors and paused to save and initialize the domain immediately. Your model should now look like the one shown in Figure 22.6.
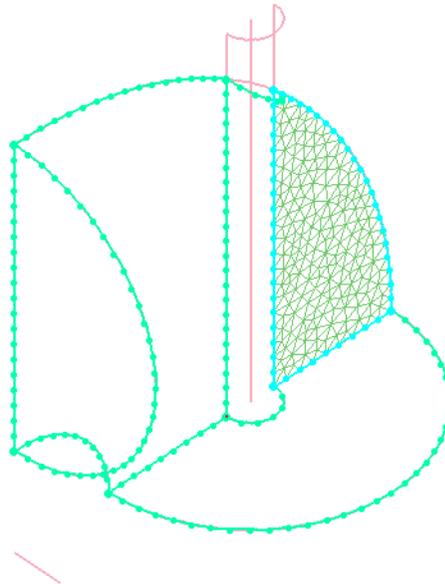
Figure 22.6 The First Domain

Continue creating the remaining five domains while still in the CREATE A DOMAIN menu by simply selecting the connectors as directed below and pausing to allow Gridgen to initialize the completed domains:

1. Pick IF.

2. Pick FE.

3. Pick EH.

4. Pick HI.

5. Pause for initialization.

6. Pick HI.

7. Pick IK.

8. Pick KJ.

9. Pick JH.

10. Pause for initialization.

11. Pick JH.

12. Pick HE.

13. Pick EL.

14. Pick LM (straight line).

15. Pick MJ.

16. Pause.

17. Pick LM (straight line).

18. Pick ML (arc).

19. Pause.

20. Pick <u>ML</u> (arc).

21. Pick <u>LE</u>.

22. Pick <u>EF</u>.

23. Pick <u>FK</u>.

24. Pick <u>KJ</u>.

25. Pick <u>JM</u>.

26. Pause.

27. **Done**

28. **Done**

The domains rendered in purple have been initialized parametrically ([Section 11](#) of the User Manual). This means the domains' grids have been initialized in the parametric space of the corresponding underlying data-base surface and will not require projection in order to make them assume the shape of the surface. The purple color is used as a visual cue to let you know this has taken place. Your model should now look like the one shown in Figure 22.7.
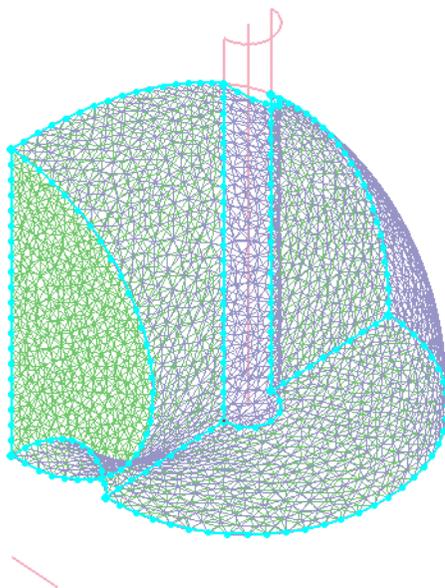
Figure 22.7 All of the Domains Created

Your rendered model in the Display window is quite a cluttered mess with all of the triangular surface grids being displayed. In order to remove some of the clutter in the Display window to make later work a little easier, again return to the DISPLAY COMMANDS menu ([Section 2.9](#) of the User Manual) to change some display attributes of the domains and turn off display of the connector grid points:

1. **Disp**

2. Toggle off Show **Con GPs**

3. Edit Dspla **Dom**

4. Pick **All**

5. **Done**

6. **Off**

7. **Done – Apply Atts**

8. **Abort**

You also really do not need to use the database entities any more for the remainder of the tutorial. Disabling the database entities will remove more clutter from your display. The difference between changing the display attribute to off, as you did for the domains, and disabling, as you are about to do for the database entities, is disabled objects can no longer be selected for other operations such as modification or deletion. While still in the DISPLAY COMMANDS menu, disable your database entities as follows:

1. En/Disable **DB**

2. Pick **None**

3. **Done**

4. **Done**

Your model should now only have the edges of the domains rendered in the Display window as shown in Figure 22.8. Note that the domain edge connectors are now rendered in a light blue color to give you a visual cue as to how those connectors are being used. Connectors that are a part of a domain are always displayed in this color. You are now ready to move on to creating your block in the next section.
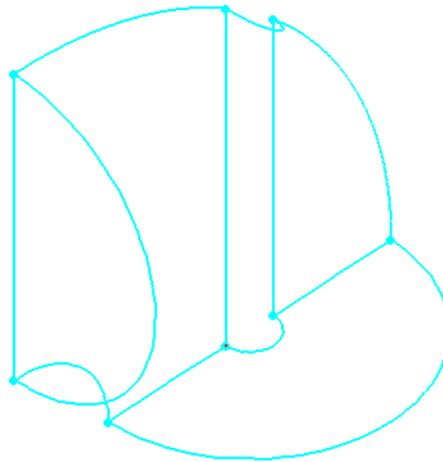
Figure 22.8 Turning Off the Domains' Interiors Reduces Visual Clutter

9.9   *Creating the Block*

For an unstructured topology you only need a single face to define a block, but it will have to form a watertight enclosed shell. Building a simple unstructured block with a single face proceeds very quickly. Create your block now:

1. **Blocks**

2. **Create**

3. Cell Type **unstructured**

4. **Assemble Faces**

5. **Add 1st Face**

6. Pick **All Domains**

7. **Save the Face**

At this point you have already made a complete block which you could save and then move on. But take note of the orange arrows emanating from the surfaces of the block. These represent the face normal vectors. Gridgen will always attempt to orient these properly, but you should make sure that they are pointing into the volume to be gridded. They might in unusual cases be pointing out of the block as shown in the side view of the model in Figure 22.9.
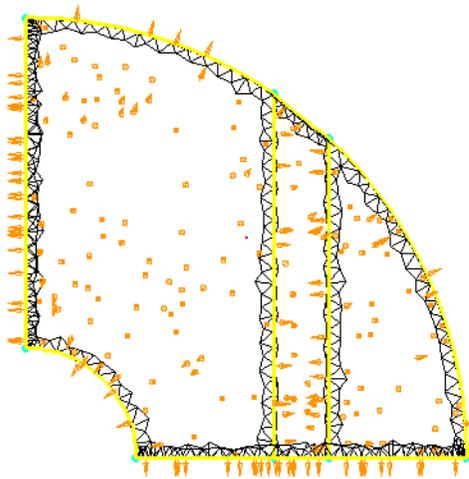
Figure 22.9 Block Face Normals May Point Outward

You want to have the normals pointing into the interior of this block since that represents the volume we intend to discretize with tetrahedra. If your normals look like those in Figure 22.9, reorient the block face normals using **Reorient Faces** while still within the CREATE AN UNSTRUCTRD BLOCK menu.

Now you may save your block.

8. **Done – Save Blocks**

You have now successfully created and saved your block with surface normals pointing into the interior. Note that the color of the block edge connectors is now dark blue giving a color cue that the connectors are used in a block. The block is now ready to be initialized in the unstructured solver.

9.10 *Initializing and Examining the Tetrahedra*

Now that the block has been created, you can initialize the interior tetrahedral volume. In order to minimize memory usage, Gridgen does not compute interior volume grids by default. Once a tetrahedral volume has been generated, it will be maintained in memory by Gridgen until you release it intentionally or you take some action that requires re-initialization, such as changing the number of grid points on one of the blocks edge connectors. Initialize the tetrahedral volume now by:

1. Run Solver **Unstrctrd**

2. Select the block.

3. **Done**

4. **Initialize**

5. **Done - Save**

In rare cases the solver may have trouble creating an interior in a fashion that preserves the surface grid elements on the boundary of the block. When this happens Gridgen will highlight the problem cell edges or entire cells while displaying an error message in the Message window. These issues typically can be resolved by changing the resolution and/or smoothness of the domain in the problem area.

There is now a complete tetrahedral volume for the block although there has been no visible change in how the model is shown in the Display window. **Examine** (Section 16 of the User Manual) the block now to see some sample crinkle surfaces on the interior of the volume mesh. Proceed as follows:

1. **Examine**

2. Select the block.

3. **Done**

4. Phy **X**

At this point Gridgen calculates a Cartesian extent box around the block and displays an x-constant scan plane. You may specify an exact location for the scan plane if you wish:

1. **X kbrd**

2. 50

3. Ref Surf **Crinkle**

Pressing the **Crinkle** button instructs Gridgen to generate a crinkle sheet made up of the faces of tetrahedra that are intersected by the scan plane. Your Display window should now show the model with the extent box, scan plane, and crinkle surface as shown in Figure 22.10.
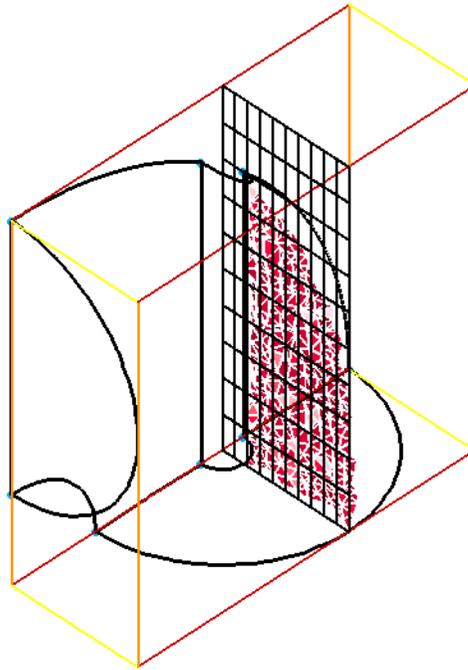
Figure 22.10 A Crinkle Surface Helps to Visualize the Volume Grid

You could also have selected Ref Surf **Flat**, which would generate a true cut through the tetrahedral mesh. This results in a flat sheet of polygons, many of which are going to be slivers. While this can yield useful grid diagnostic information, it is likely not as useful as the Ref Surf **Crinkle** option. The Ref Surf **Cells** option will instruct Gridgen to display entire tetrahedra that are intersected by the scan plane.

By default these reference surfaces continue to be displayed until you either **Erase Cur.,** which erases the reference surface currently under the hiliter, or until you **Erase All**. Exiting from **Examine** also turns off the display of any reference surfaces, but their locations are stored in memory in case you return to the **Examine** utility. At that time, your previous reference surfaces are restored and updated to reflect any changes in the surface or volume grids.

While in examine, feel free to try other Display styles, as well as rendering the reference surfaces colored by some Function. When you have finished using **Examine**, press the **Abort** button to exit, and then **Done** to return to the MAIN MENU.

9.11   *Saving Your Model and Export to Solver*

Hopefully you have been saving your model along the way, but this is definitely a good point at which to make sure you save a good copy of your database and grid models. Since you have created a database model within Gridgen, you also need to be sure to save this data. It will not be included in the Gridgen file you will save as well, however references to this database and database file will be included in the Gridgen restart file. Note that it is important to always save your database The database must be saved using Gridgen's native Composite database file format. Any database entity from any source can be stored in a composite file. You can also mix entities from different sources (IGES, PATRAN, Gridgen) in the same composite file. In case you did not save your database model earlier, do the following while still in the INPUT/OUTPUT COMMANDS menu:

1. Database **Export**

2. **Composite**

3. **Done**

4. Type In... **Name**

5. `elbow`

Note that Gridgen will automatically append the default file extension (.dba).

6. **Done**

Now, to save your Gridgen restart file, proceed as follows:

1. **Input/Output**

2. Gridgen **Export**

3. Type In... **Name**

4. `elbow`

Note that Gridgen will automatically append the default file extension (.gg).

Your restart file is saved immediately. If you had typed in the name of an existing file, you would have been prompted on whether to overwrite that file or not.

Now before you export your flow solver files for Fluent, you will need to set some boundary conditions. Proceed as follows:

1. **Analysis S/W**

2. **Set BCs**

Your Blackboard window should now look like the one shown below. The list contains an entry for each domain. The first column (which contains only the number "1" for this problem) is the block number. The second column of information shows the face number for that block. After the face number in parentheses is the domain number associated with that boundary patch. The blank space after the parentheses is where the name of the boundary condition you apply will be displayed. Set the boundary conditions as outlined in the following steps:

```
 1,fac1(  1)
 1,fac1(  2)
 1,fac1(  3)
 1,fac1(  4)
 1,fac1(  5)
 1,fac1(  6)
```

```
═══ SELECT DOMAINS FOR BC ═══
Scroll  ⇓  ⇘  ⌄  ⌃  ⇗  ⇑
Pick   All   a│None^n│Tog ^t
Page   in Box│on Box│ByText
      🔥get ◆  1st ◇ last
Set to ◇  off ◇  ^on ◆  tog
SortBy ◆ num2 ◇ BC 3 □ rev4
Pick by expression        ^x
Done  ent│Help    ?│Abort esc
```
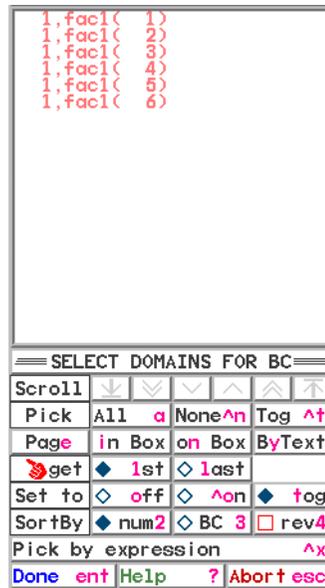
3. Select the shaft and elbow domains.

4. **Done**

5. **Wall**

6. Select the two symmetry plane domains.

7. **Done**

8. **Symmetry**

9. Select the elbow opening which lies in an **x**-constant body coordinate plane.

10. **Done**

11. **Velocity Inlet**

12. Select the elbow opening which lies in a **y**-constant body coordinate plane.

13. **Done**

14. **Outlet Vent**

15. **Abort**

Your boundary conditions are now set and you are ready to export your flow solver file. While still inside the ANALYSIS S/W COMMANDS menu proceed to export the file:

1. **Export Analysis Data**

2. Type In... **Name**

3. `elbow`

Note that Gridgen will automatically append the default file extension (.cas).

4. **Done**

Now export your Gridgen restart file once more to preserve your boundary condition settings in the restart

file. Congratulations! You have successfully exported all of your necessary files. And, of course, this tutorial is now complete.

10. Droplet in a Cylinder: Hybrid Grid

10.1 Introduction

This tutorial will give you a guided tour of the Gridgen process for creating a hybrid volume grid for a flow field through and around a droplet within a cylinder. The grid will consist of tetrahedral, hexahedral, prism and pyramid cells that form the volume of the flow field.

10.2 Topics Covered

This sample session is a short introduction to using Gridgen. Basic Gridgen skills you will learn during this tutorial session are:

- creating database curves using Curve **Offset**
- creating database surfaces using the Surface **Revolution** command
- using **On DB Entities** to automatically create connectors and unstructured domains
- extruding prisms using the **Nrml** extrusion command
- using the **Rot** extrusion command to rotationally extrude structured grids
- using the **Hyp** extrusion command to extrude hexahedral cells
- initializing the tetrahedral volume grid using the Run Solver **Unstrctrd** command
- exporting the hybrid grid using the **Export Analysis Data** command

10.3 Background

You should complete all previous tutorials before attempting this session, since it assumes you already have the skills covered. You should also familiarize yourself with Section 7 , Section 11 and Section 12 for further information on unstructured meshing and extrusion methods used in this tutorial

10.4 Geometry

The droplet in a cylinder configuration you will use for this tutorial is shown in Figure 24.1.
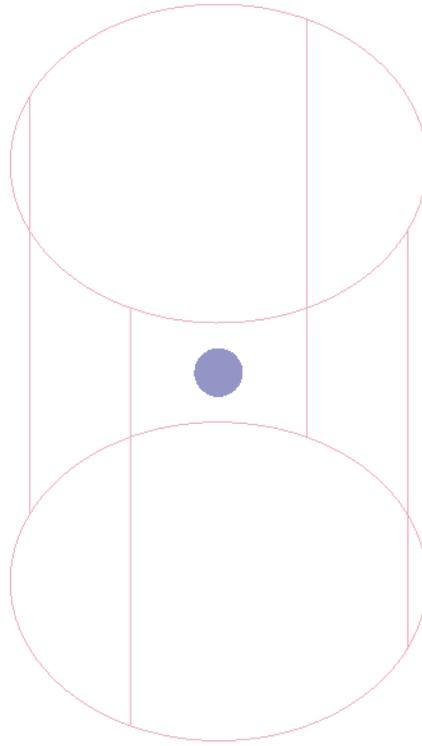
Figure 24.1 Droplet in a Cylinder Geometry

The geometry you will use to create the hybrid grid topology consists of a spherical droplet of diameter 15.0mm suspended in a cylinder of diameter 128.0mm and a height of 200.0mm.

The block topology is shown in Figure 24.2. The topology consists of both structured and unstructured blocks with pyramid transition cells between the structured and unstructured volumes.
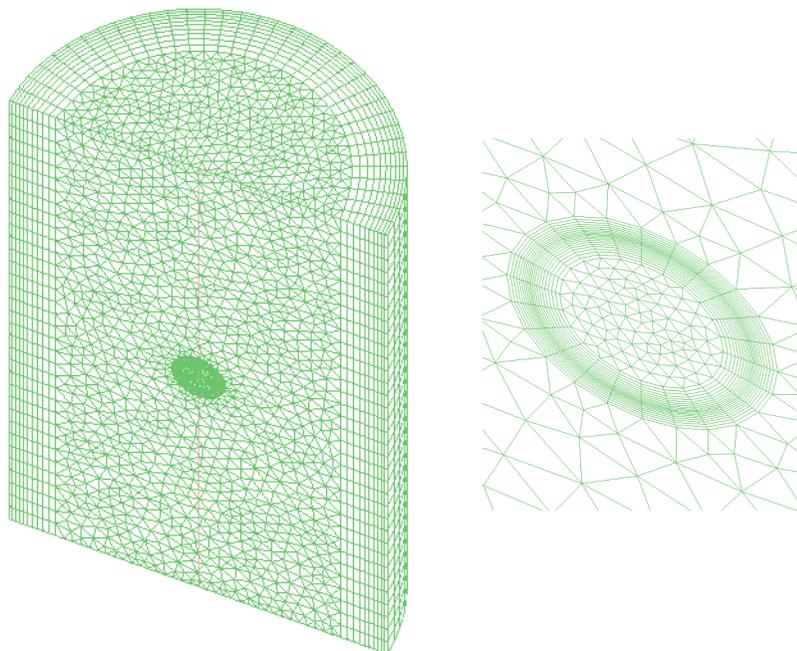
Figure 24.2 Hybrid Grid Topology for Cylinder and Droplet

The droplet has an unstructured surface grid from which prism cell layers are extruded into the interior of the droplet and outward into the cylinder, with clustering of the grid points close to the droplet surface. The remaining core volume is filled in with tetrahedral cells.

The cylinder has a structured surface grid from which hexahedral cell layers are extruded inward into the cylinder. The ends of the cylinder are capped off with an unstructured surface grid, and finally the volume between the cylinder and droplet is filled with tetrahedral cells.

For convenience, only half of the geometry will be modeled to take advantage of symmetry.

10.5 Getting Started

Start Gridgen by either typing the string `gridgen` on Unix workstations or simply double-clicking the Gridgen shortcut key on your PC's desktop.

The first step in running Gridgen is to select the software (Section 17.1 of the User Manual) to be used in the analysis. This will ensure the grid generated by Gridgen is consistent with the capabilities of the analysis software. For this tutorial, you will be using the latest version of the FLUENT flow solver (by default Gridgen selects the Generic flow solver). To select the current FLUENT flow solver, start in the MAIN MENU:

1. **Analysis S/W**

2. **Select Analysis S/W**

3. Turn on the **3D** and **FLUENT** toggles.

4. **Next Page**

5. **Done**

6. **Done**

Use of the **3D** button above tells Gridgen that you will create a 3D grid containing volume cells, as opposed

to a 2D grid with only surface elements. Verify that you have selected the 3D FLUENT flow solver by checking the Status window.

You also need to set up the default average grid spacing parameter for the unstructured surface mesh. From the MAIN MENU:

1. **Defaults**

2. Con Dim **avg** Δ**s**

3. 2

4. **Done**

You have set up the grid attributes that will allow you to create the hybrid grid for this tutorial.

10.6 *Creating Database Entities*

You are now ready to begin creating the database geometry elements that will form the foundation of the grid. The process begins with the creation of simple arcs and lines. Then you will create surfaces of revolution to form the droplet surface.

The first entity you will create is the circular arc section defining the half circumference of the droplet. The radius of the arc is at point 7.5, 0, 0. From Gridgen's MAIN MENU:

1. **Database**

2. **Create**

3. Curve **Circle**

4. Add CP □ **via Keybrd**

5. 0,-7.5, 0

6. Add CP □ **via Keybrd**

7. 0, 7.5, 0

8. Store CP □ **via Keybrd**

9. 7.5, 0, 0

10. **Done - Save DB curve**

That completes the definition of the circular arc shape. Next, create the line that defines the center line of the cylinder and the radius of the cylinder. You should still be at the DB CREATION menu:

1. Curve **Line**

2. Add CP □ **via Keybrd**

3. 0, -100, 0

4. Add CP □ **via Keybrd**

5. 0, 100, 0

6. **Save DB curve**

Next, you will need to create an edge curve defining the radius of the cylinder by offsetting the center line curve by 64.0mm. From the DB CREATION menu:

1. Curve **Offset**

2. Pick the center line curve.

3. **Enter Offset Distance**

4. `64.0`

5. **Save**

Press the keyboard hot key *r* to reset the view. The Display window should look like Figure 24.3.
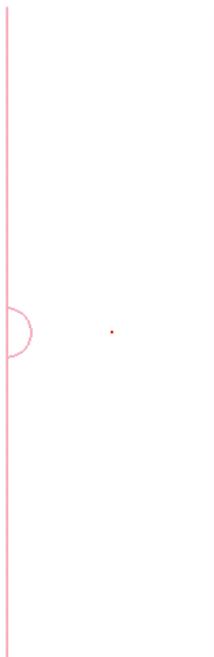
Figure 24.3 Database Curve for Droplet and Cylinder

You will create the surface of the droplet as a surface of revolution. From the DB CREATION menu:

1. Surface **Revolution**

2. Pick the circular arc, the generatrix, of the droplet.

3. Pick the center line curve, the axis curve.

4. **(Re)Set the Angle**

5. `180`

6. **Done-Save Surf of Revol**

7. **Done**

You can change the surface rendering.

1. **Disp**

2. Edit Dspla **DB**

3. Select the surface of the droplet (Entity #4 from the Browser).

4. **Done**

5. render mode **Isolines**

6. `5 5`

7. **Done - Apply Atts**

8. **Abort**

9. **Done**

You can zoom into the area around the droplet for the next step of the grid generation process. If everything is going well, your Display window should look like Figure 24.4.

It is a good idea to save the database as a Gridgen Composite file before proceeding further.

1. **I/O**

2. Database **Export**

3. Type **composite**

4. **Done**

5. Type In... **Name**

6. `hybrid`

7. **Done**
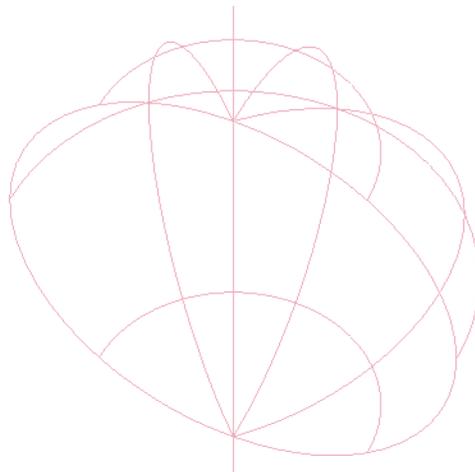
Figure 24.4 Surface of Revolution for the Droplet

10.7   Unstructured Surface Grid on Droplet

You are now ready to create an unstructured surface grid on the surface of the droplet. The number of grid points on the droplet surface is based on the avg Δs grid point spacing parameter defined in Section 10.5. From the MAIN MENU:

1. **Domains**

2. **Create**

3. Cell Type **unstructured**

4. **On DB Entities**

5. Pick the surface of the droplet.

6. **Done**

7. **Done**

You have now created an unstructured surface grid on the droplet surface. For the purposes of the tutorial we will only create half of the droplet and utilize the symmetry of the geometry. The surface grid should look like Figure 24.5 on the following page.
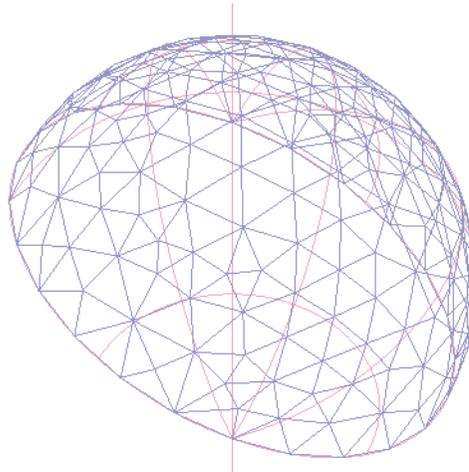
Figure 24.5 Surface Grid around Droplet Surface

10.8   *Droplet Interior Prism Extrusion*

You are now ready to extrude a layer of prisms into the interior of the droplet. You should read Section 12 for explanations of the various attributes used in this section. These attributes are shown in the menus below:

From the MAIN MENU:

1. **Blocks**

2. **Create**

3. Extrude **From Doms**

4. Pick **All**

5. **Done**

You are going to extrude the layer of prisms using the *normal* extrusion method. The normal extrusion attributes menu is shown below.

```
═══NRML EXTRUSION ATTRIBS═══
 Focus   Next    n Prev    ^n
 Apply to ◇   Focus ◆ All  ^s
          Smoothing
 Normal  Count   1 Relax  ^1
  Step   Count   2 Relax  ^2
  Grid   Count   3 Angle  ^3
 Edges   ◇ 1D   ◆ 2D  ◇ Cnt
     Corner Suppression
 Stretching Factor  Set      4
 Turning Angle  Corner  Flat
 Stp Reduction  Primry  Scndry
 Aspect Ratio Factor       ^4
   Δs    Init 5 Min   6 Max   7
  Growth Rate  ◆ Geometric  a
         Set   8 ◇ Subcon    b
 Boundary Conditions         c
  March   ◆ Plane   ◇ Vector
          Set    e Flip    d
 Done                      ent

 Help                        ?
```

1. Type **Nrml**

2. **Set Attributes**

This should take you to the NRML EXTRUSION ATTRIBS menu:

3. Δs **Init**

4. `0.1`

5. **Boundary Conditions**

6. Turn on the **Constant Z Plane** toggle.

7. **Done Setting Attributes**

Now create the prism layer. From the BLOCK EXTRUSION menu:

1. **Run N**

2. `10`

3. **Done**

4. **Done**

You have now created a layer of prism cells from the triangular surface grid into the interior of the droplet.

By default, the rendering of the domains of the extruded prism layers has been turned off. To display the domains on the symmetry plane and interior surface of the extruded prism layers, do the following:

1. **Disp**
2. Edit Dspla **Fac**
3. Pick **All**
4. **Done**

You are now in the DISPLAY ATTRIBUTES menu:

1. display style **Hidden Lines Removed**
2. **Done -Apply Atts**
3. **Abort**
4. **Done**

The rendered domains are shown in Figure 24.6.
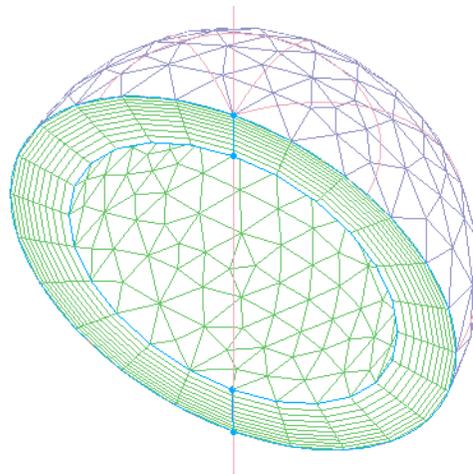
Figure 24.6 Extruded Prism Layers Inwards into the Droplet

10.9   Droplet Core Volume

You now have to create the interior tetrahedral mesh forming the core of the droplet. This will complete the hybrid mesh on the interior of the droplet consisting of prism and tetrahedral cells.

10.9.1   Close the Core of the Droplet

You must first create an unstructured domain at the symmetry plane of the droplet core. This will close the core to form an enclosed shell within which a tetrahedral grid can be generated. From the MAIN MENU:

1. **Domains**
2. **Create**
3. Cell Type **unstructured**
4. **Assemble Edges**

5. Turn on the **Auto Save** toggle.

6. Pick the two connectors (Entities #3 & #4) on the interior perimeter of the extruded prism layer.

7. **Done**

8. **Done**

An unstructured domain on the symmetry plane forming the core of the droplet is created by the above operation. To display the symmetry plane core domain of the droplet:

1. **Disp**

2. Edit Dspla **Dom**

3. Select the domain (Entity #5 from the Browser) on the symmetry plane of the droplet.

4. **Done**

5. display style **Hidden Lines Removed**

6. **Done -Apply Atts**

7. **Abort**

8. **Done**

The Display window should now look like Figure 24.7 on the following page.
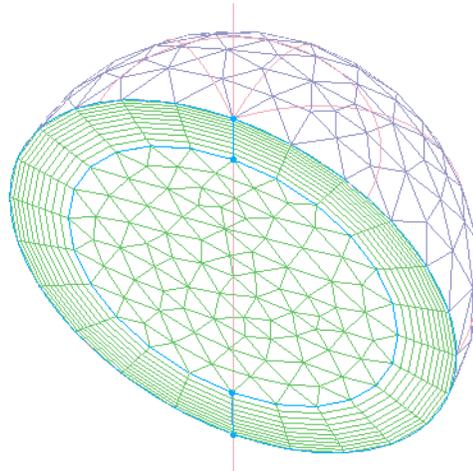
Figure 24.7 Closed Core of the Droplet

10.9.2   Droplet Core Volume Grid

To finish the grid within the droplet, you will now need to create a 3D volume grid enclosed by the core surface grid. The core block has one face defined by two domains: the inner domain of the prism extrusion block and the symmetry plane domain just created. From the MAIN MENU:

1. **Blocks**

2. **Create**

3. **Assemble Faces**

4. **Add 1st Face**

5. Pick **Multiple Doms**

6. Pick the domain on the symmetry plane of the core (Entity #5 from the Browser).

7. Pick the inner surface domain of the extruded prism layer (Entity #2 from the Browser).

8. **Done**

9. **Save the Face**

At this point you have made a complete unstructured block, but take note of the orange arrows emanating from the surfaces of the block. Make sure they are all pointing into the volume of the core; if not, you can use the command:

1. **Reorient Faces**

Otherwise:

1. **Done-Save Blocks**

Now you have successfully created and saved the block with the surface normals pointing into the interior of the core. Tetrahedral cells may now be generated within the core block.

10.9.3   Initializing the Tetrahedral Volume Grid

Now that the core block has been created, you can initialize the interior tetrahedral volume within the core of the droplet. From the BLOCK COMMANDS menu, initialize the tetrahedral volume by:

1. Run Solver **Unstrctrd**

2. Pick the core block from the Browser.

3. **Done**

4. **Initialize**

5. **Done - Save**

6. **Done**

You now have a complete volume grid of the droplet interior, consisting of prism cells extruded from the unstructured surface grid and tetrahedral cells initialized in the core. You may wish to examine the volume grid using the **Examine** command in the BLOCK COMMANDS menu.

10.10 Droplet Exterior Prism Extrusion

You are now going to extrude prism layers from the exterior of the droplet into the cylinder. The method is similar to the one described in Section 10.8 of this tutorial. From the MAIN MENU:

1. **Blocks**

2. **Create**

3. Extrude **From Doms**

4. Pick the spherical unstructured domain (Entity #1 from the Browser) on the surface of the droplet.

5. **Done**

You are going to extrude the prisms normal to the droplet domain. Therefore, you will have to set the normal extrusion attributes.

1. Type **Nrml**
2. **Set Attributes**

This should take you to the NRML EXTRUSION ATTRIBS menu.

3. Δs **Init**
4. `0.1`
5. **Boundary Conditions**
6. Turn on the **Constant Z Plane** toggle.

Since you will be extruding a second prism layer outward into the cylinder, you must reorient the normals so that they face outwards away from the droplet surface.

1. **Set Other Attributes**
2. March **Flip**
3. **Done**

Now create the prism layer. From the BLOCK EXTRUSION menu:

1. **Run N**
2. 10
3. **Done**
4. **Done**

This completes the hybrid mesh on the droplet. Set all faces on the blocks to hidden line; the Display window should then look like Figure 24.8 on the following page. Notice the grid spacing on the prism layers is clustered close to the surface of the droplet. The grid spacing then becomes progressively coarser away from the droplet surface.
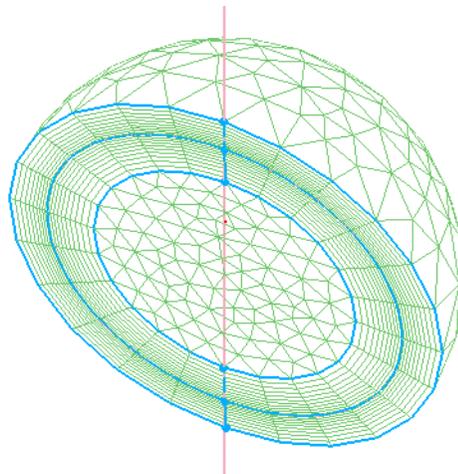
Figure 24.8 Prism Layers Extruded Outward into the Cylinder

10.11 Structured Surface Grid on Cylinder

You are now ready to create a structured (quadrilateral) surface grid on the surface of the cylinder. From this you will extrude a layer of hexahedral cells inwards into the cylinder. To speed up construction you will set defaults values for connector dimensions using the Defaults menu option. To set the defaults from the MAIN MENU:

1. **Defaults**
2. Con Dim **dimen**
3. 35
4. **Done**

Any connectors you create after this point will automatically have 35 grid points assigned to them.

You will now create the connector defining the length of the cylinder at its radius. From the MAIN MENU:

1. **Connectors**
2. **Create**
3. **On DB Entities**
4. Pick the database curve defining the length of the cylinder at its radius (Entity #3 from the Browser).
5. **Done**
6. **Done**

You now extrude from the new connector a structured grid around the center axis. You will be using another type of extrusion command, *rotation,* to extrude the connector around a predefined axis of rotation. From the MAIN MENU:

1. **Domains**
2. **Create**
3. Cell Type **structured**
4. Extrude **From Cons**
5. Pick the connector defining the length of the cylinder at its radius.
6. **Done**

You now need to select the rotation type of extrusion method. Since you are only modeling half of the cylinder, you only need to rotate the extrusion by 180°.

1. Type **Rot**
2. **Set Attributes**
3. **Total Rotation Angle**
4. **Enter via Keybd**
5. 180
6. **Set Rotation Axis**
7. **Use Y-Axis**

8. **Done**

Finally, run the rotation extrusion command. From the DOMAIN EXTRUSION menu:

1. **Run N**

2. 35

3. **Done**

4. **Done**

Your topology should now look like Figure 24.9. Use the keyboard hot key *r* to reset the view and display the topology.
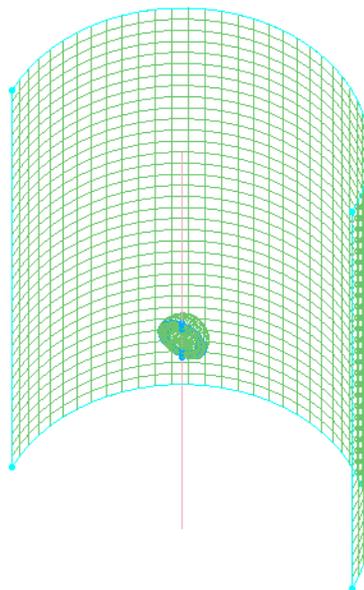
Figure 24.9 Structured Surface Grid on Cylinder

10.12 *Hexahedral Cells from the Cylinder Surface Grid*

You will now extrude hexahedral cell layers from the surface grid into the interior of the cylinder. The process is similar to the extrusion of the prism layer in Section 10.8 of this tutorial. You will be extruding cells inward, normal to the cylinder surface, but using the *hyperbolic* extrusion method. From the MAIN MENU:

1. **Blocks**

2. **Create**

3. Extrude **From Doms**

4. Pick the surface domain of the cylinder.

5. **Done**

You will select the extrusion method and set the extrusion attributes. From the BLOCK EXTRUSION menu:

1. Type **Hyp**

2. **Set Attributes**

3. Δs **Init**

4. `1.0`

5. **Boundary Conditions**

6. Edges **All**

7. Edges **Set**

8. Constant **Auto**

9. March **Flip**

10. **Done**

Run the hyperbolic extrusion.

1. **Run N**

2. `10`

3. **Done**

4. **Done**

The Display window should look like Figure 24.10. Again the faces have been displayed as **Hidden Lines Removed** from the DISPLAY COMMANDS menu.
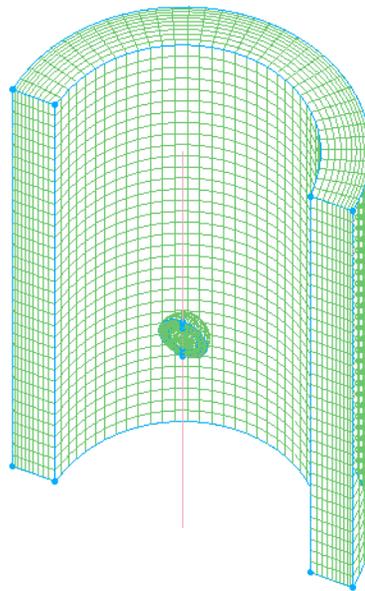
Figure 24.10 Extruded Hexahedral Layer Inwards of the Cylinder Surface

Notice the cells are clustered towards the cylinder surface and become coarser as they move inwards from the cylinder surface.

10.13 Volume Grid in the Core of Cylinder

You now have to close the core of the cylinder to form the 3D volume. You will do this by creating unstructured domains that cap the ends and symmetry plane of the cylinder core, forming a water tight enclosed shell. A tetrahedral volume grid can then be initialized from the faces.

For convenience we have assigned a naming convention to important points in the grid shown in Figure 24.11.
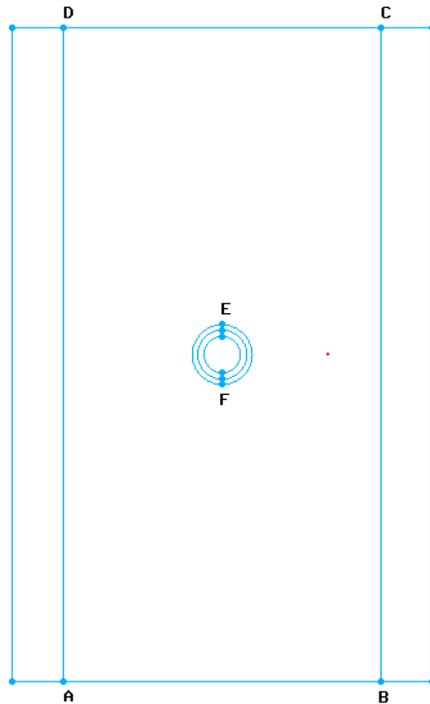
Figure 24.11 Labeled Coordinates on Cylinder Core Symmetry Plane

10.13.1   Close Cylinder Ends

You are now going to create two unstructured domains that will cap the ends of the cylinder core. From the MAIN MENU:

1. **Connectors**

2. **Create**

3. **2 Point Connectors**

Connector <u>AB</u> :

1. Place the 3D cursor over Point A.

2. Add CP □ **by Picking**

3. Place the 3D cursor over Point B.

4. Add CP □ **by Picking**

Connector <u>CD</u> :

1. Place the 3D cursor over Point C.

2. Add CP □ **by Picking**

3. Place the 3D cursor over Point D.

4. Add CP □ **by Picking**

5. **Done**

6. **Done Creating Conns**

7. **Done**

The Display window should now look like Figure 24.12.
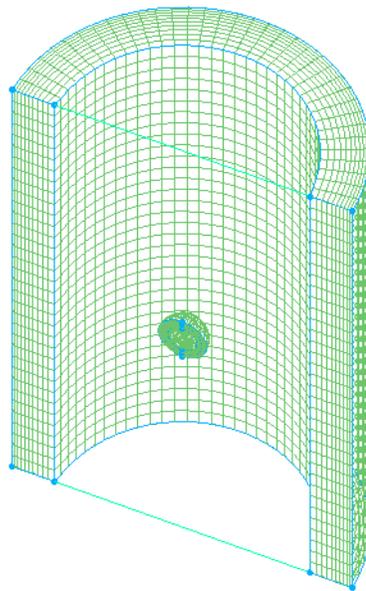
Figure 24.12 Connectors on the Ends of Cylinder Core

You will now create an unstructured domain for each of the end caps. From the MAIN MENU:

1. **Domains**

2. **Create**

3. Cell Type **unstructured**

4. **Assemble Edges**

5. Turn on the **Auto Save** toggle.

6. Pick connector AB then the half circle connector BA.

7. Pick connector CD then the half circle connector DC.

8. **Done**

The unstructured domains are shown in Figure 24.13.
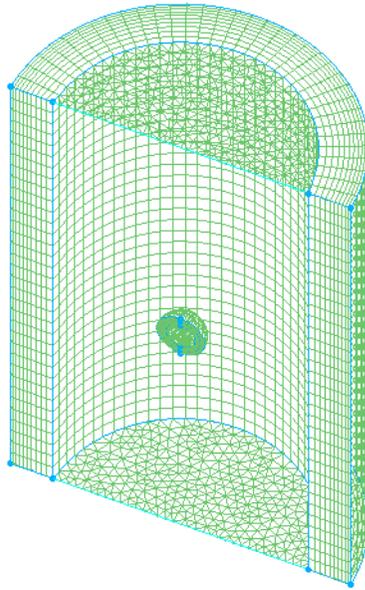
Figure 24.13 Domains on Ends of Cylinder Core

10.13.2   Close Symmetry Plane

Next you need to create the domain that closes the core of the cylinder at the symmetry plane. From the DOMAIN COMMANDS menu:

1. **Create**

2. **Assemble Edges**

3. Turn off the **Auto Save** toggle.

4. Pick the connectors <u>AB</u>, <u>BC</u>, <u>CD</u>, and <u>DA</u> defining the outer loop connectors of the cylinder core symmetry plane.

5. **Next Edge**

6. Pick connectors <u>EF</u> then pick the other connector <u>FE</u> on the outer edge of the exterior prism layer on the droplet. This defines the inner loop connectors of the cylinder core symmetry plane.

When creating a domain that contains a "hole", such as this one, the orientation of the inner loop of connectors must be opposite to the orientation of the outer loop of connectors. If the inner loop of connectors has the same orientation as the outer loop, they must be reoriented through use of the **Reorient Edge** button. Then you may go on to the next step of the domain creation process:

7. **Save Domain**

8. **Done**

9. **Done**

You have now created a completely enclosed shell of the cylinder's interior core. The unstructured domain on the symmetry plane is shown in Figure 24.14.
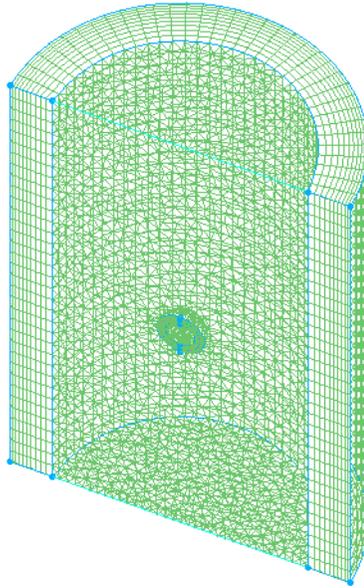
Figure 24.14 Domain on Symmetry Plane

10.13.3   Volume Grid within Core of Cylinder

The final step in the Gridgen process is to generate the volume grid within the core of the cylinder. From the MAIN MENU:

1. **Blocks**

2. **Create**

3. Cell Type **unstructured**

4. **Assemble Faces**

5. **Add 1st Face**

6. Pick **Multiple Doms**

7. Pick the domains forming the face of the cylinder core (Entities #6, #14, #15, #16 & #17 from the Browser).

8. **Done**

9. **Save the Face**

10. **Done-Save Blocks**

11. Run Solver **Unstrctrd**

12. Pick core block of the cylinder.

13. **Done**

14. **Initialize**

15. **Done - Save**

The unstructured solver has created a tetrahedral volume grid. Pyramid elements are automatically created between the structured hexahedral block and the tetrahedral cells in the bulk of the volume. These pyramid cells are the transition cells that allow an unstructured volume grid to be formed from a structured grid.

You can examine the entire volume grid using the **Examine** command in the BLOCK COMMANDS menu. Figure 24.15 shows a cross section through the grid; notice the different types of cells that form the hybrid grid.
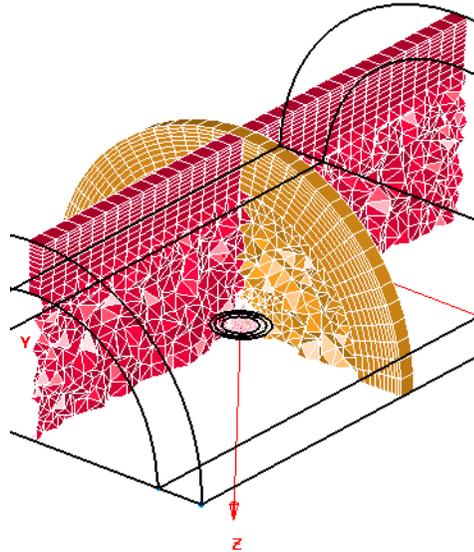
Figure 24.15 Examining the Hybrid Grid

10.14 Saving Your Model and Export to Solver

Now would be a good time to make sure you save a good copy of your database and grid models. We recommend always exporting your database models first, as your Gridgen restart file includes information pertaining to your database file and itsFrom the MAIN MENU:

1. **Input/Output**
2. Database **Export**
3. Type **composite**
4. **Done**
5. Type In... **Name**
6. `hybrid`
7. Gridgen **Export**
8. Type In... **Name**
9. `hybrid`
10. **Done**

You can now export to your flow solver files for Fluent. For the purposes of this tutorial you will not be setting any boundary conditions, but you may do so if you wish. From the MAIN MENU:

1. **Analysis S/W**

2. **Export Analysis Data**

3. Type in... **Name**

4. `hybrid`

5. **Done**

6. **Done**

You have successfully exported all of your necessary files, and you may now exit from Gridgen. This tutorial is now complete.

11. Tension Bar: Feature Extraction

11.1   Introduction

This tutorial will give you a guided tour of the Gridgen process for using the *Feature Extraction* (FE) tool. Feature Extraction is used automatically during shell import or interactively to find geometric features (for example, slope discontinuities) in the shells. Shells represent the faceted equivalent of a continuous B-Spline surface in Gridgen's database.

The user sets a *split angle* that will be the maximum angle allowed between two adjacent triangles in a shell. Feature lines identified by the FE algorithm will then be used to either split a shell into two shells along a feature line or to create a database curve (a line segment). Shells will also be split automatically during import based on point-to-point connectivity.

11.2   Topics Covered

This sample session is a short introduction to using feature extraction. Basic Gridgen skills you will learn are:

- importing STL shell surfaces into Gridgen using the Database **Import** command

- using the **Feature Ext** command to split the shell surfaces into discrete database entities

- automatically create domains using **Con Split** and **Set Angle** commands to ensure that nodes and connectors are created at feature lines

- creating blocks from the domains to form the structured volume grid

11.3   Background

You should complete all previous tutorials before attempting this session, since it assumes you already have the skills covered in them.

11.4   Geometry

The geometry to be meshed is a 3D tension bar, shown in Figure 25.1. The tension bar geometry is provided in STL format and is comprised of triangular faceted shells that form the model. The grids will be generated by first splitting the model into discrete database entities and then applying Gridgen's **On DB Entities** tool to generate the surface grid.
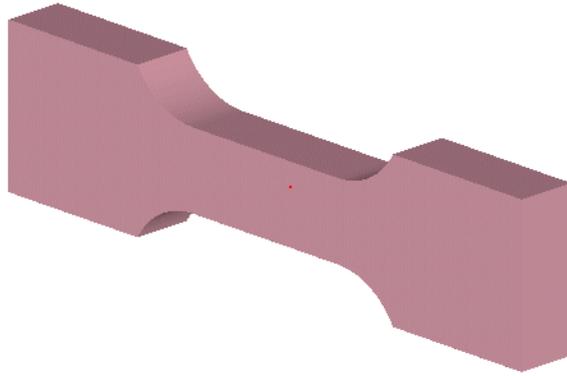
Figure 25.1 Tension Bar Shell Geometry

11.5   Getting Started

Start Gridgen by typing the string `gridgen` on Unix workstations or by simply double-clicking the Gridgen shortcut on your PC's desktop.

11.6   Import Shell (STL) Model

The first step in this tutorial is to import the shell model. The tension bar geometry you will use for this tutorial is an STL file named *tension.stl*. It is located in the *gridgen_home_path*/`examples/tutorial/fe` directory. You may want to copy it to a local directory before trying to import into Gridgen.   Assuming that you have copied the file `tension.stl` into your current working directory, from Gridgen's MAIN MENU:

1.   **Input/Output**

2.   Database **Import**

3.   Select `tension.stl` from the Browser.

4.   **Open**

5.   **Done**

Orient the view in the Display window as shown in Figure 25.2. You will notice that you have imported a single shell database entity made up of triangles defining the tension bar's geometric shape. When feature extraction is applied, the imported model is automatically divided into shells and is much more amenable to meshing.
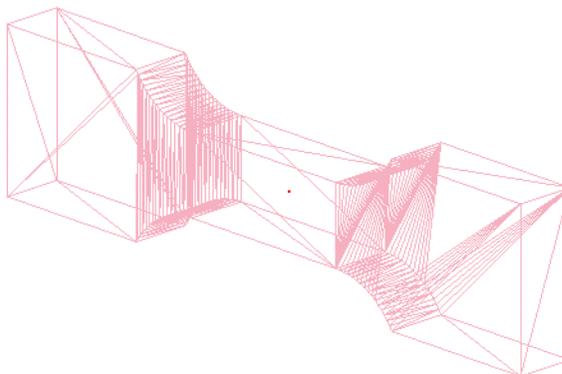
Figure 25.2 Shell Model

11.7   Setting Defaults

For the purposes of this tutorial, you are now going create a structured surface grid. You will need to set a default number of grid points to `15` for all of the connectors.

From the MAIN MENU:

1. **Defaults**

2. Con Dim **dimen**

3. `15`

4. **Done**

Note, setting the **Split Angle** default prior to importing the shell surface will result in the surfaces being

split automatically. This is useful if there are discrete angles in your model that can be split into definite features. However, for more complex models, it is recommended you extract features from the shell surfaces via the **Feature Ext** command in the DATABASE COMMANDS menu.

11.8   Feature Extraction

You are now going to use the *Feature Extraction* tool to extract geometry from the shell surface. From the MAIN MENU:
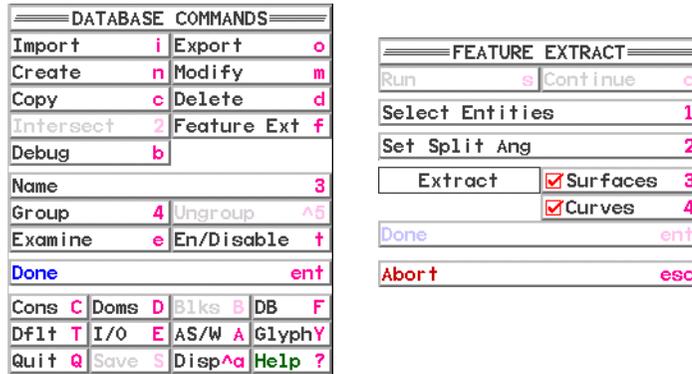
1. **Database**

2. **Feature Ext**

3. **Select Entities**

4. **All**

5. **Done**

6. **Set Split Ang**

7. 45

8. Turn the Extract **Curves** toggle off.

9. **Run**

The original shell surface has now been split into ten discrete surfaces as shown in the Browser on the next page:

```
FEATURE EXTRACT: Paused

Selected:      1 entity
Split Ang =   45.0 deg

100 %


1 shell split into:
   surfaces =  10




════════FEATURE EXTRACT════════
ReRun           s  Continue     c
ReSelect Entities              1
Set Split Ang                  2
    Extract     ☑ Surfaces     3
                □ Curves        4
Done                          ent
Abort                         esc
```

The Display window should look like Figure 25.3. The thick yellow lines in the Display window indicate the boundaries of the new shell surfaces resulting from the splitting operation.
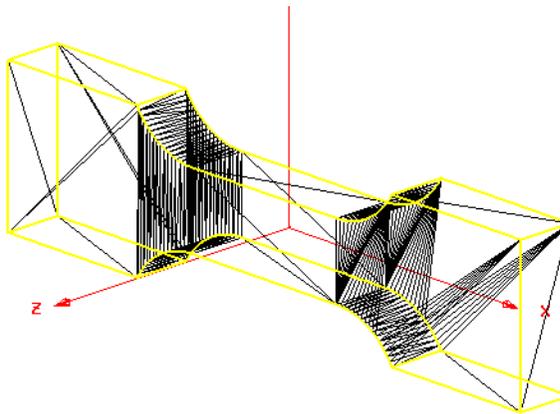
Figure 25.3 Feature Extraction

10. **Done**

11. **Done**

To view the individual shell database entities you can change the display style in the DISPLAY COM-MANDS menu.

11.9   *Creating Structured Surface Grids*

You are now ready to create structured domains on the shell database entities. From the MAIN MENU:

1. **Domains**

2. **Create**

3. Cell Type **structured**

4. Turn the **Con Split** toggle on.

5. **Set Angle**

6. 45

7. **On DB Entities**

8. Pick **All**

9. **Done**

The connector split angle ensures connectors are split at feature edges and nodes are correctly created at the ends of these edges.

Note only eight domains are created automatically. The Display window is shown in Figure 25.4.
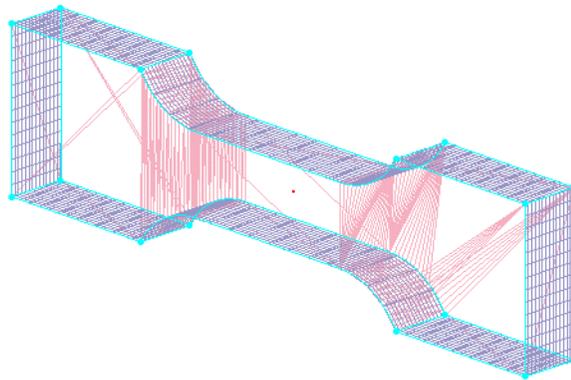
Figure 25.4 Domains on Database Surfaces

You now need to create manually the remaining domains defined in Figure 25.5. These close off the sides of the tension bar to form the faces defining the volume of the block.
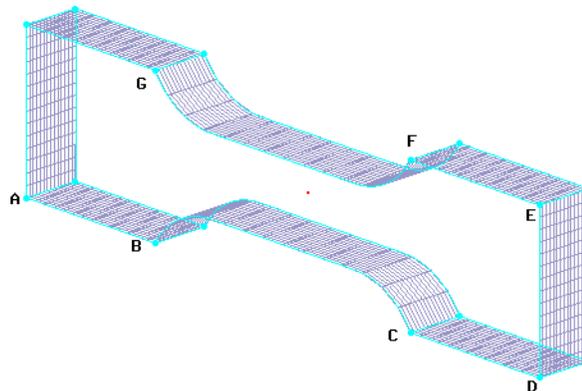
Figure 25.5 Domain Labelling on Tension Bar

You may want to orient your view as shown in Figure 25.5, as you will be creating the domain nearest to you defined by connectors <u>ABCDEFG</u>. From the DOMAIN COMMANDS menu:

1. **Create**

2. **Assemble Edges**

3. Turn the **Auto Next Edge** toggle off and turn the **Auto Complete** toggle on.

4. Pick connectors <u>AB</u>, <u>CB</u> then <u>CD</u>.

5. **Next Edge**

6. Pick connector <u>DE</u>.

7. **Next Edge**

8. Pick connectors <u>EF</u> then <u>FG</u>.

9. Pick connectors that form the opposite face of the tension bar.

10. **Done**

Turn off the database entities to more easily view the surface grid. From the MAIN MENU:

1. **Disp**

2. Turn the Show **EnbldDB** toggle off.

3. **Done**

The surface grid should now look like Figure 25.6 on the next page.
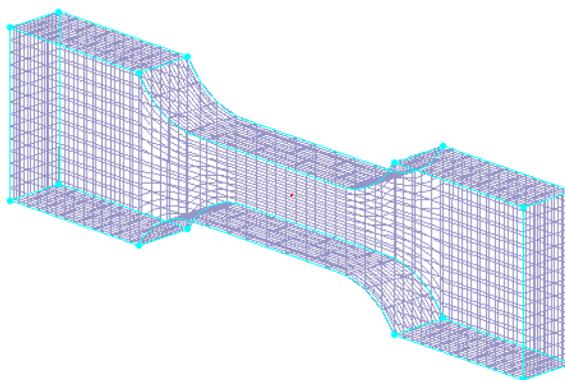
Figure 25.6 Surface Grid on Tension Bar

11.10 Creating Blocks

Finally, you can create the blocks making up the volume of the tension bar. From the MAIN MENU:

1. **Blocks**

2. **Create**

3. **Assemble Faces**

4. **Add 1st Face**

5. Pick the domain defined by the connectors <u>ABCDEFG</u> as the first face of block.

6. **Save the Face**

7. Continue picking the appropriate domains and saving each face in turn to form the block.

8. **Done - Save Blocks**

You have now successfully created the block and you may **Examine** it if you wish.

This completes the tutorial on *Feature Extraction*. The grid including the nodes, connectors, domains, blocks surface and volumes grids can now be saved.

12. Mixer: Using Baffles

12.1 Introduction

This section includes a brief tutorial to help you become familiar with Gridgen's block face creation tools for topologies containing baffles.

12.2 Topics Covered

Gridgen skills you will learn during this tutorial session are:

- setting defaults appropriate for unstructured grids using the **Defaults** command

- using the **On DB Entities** domains command to create unstructured domains

- using the Merge **Auto** command to clean up connector overlap

- building regular and baffle faces automatically using AutoSave **Faces**

- setting wall boundary conditions on the baffles using the **Set BCs** command

12.3 Background

The procedures followed in this tutorial use some of Gridgen's most automated commands. Therefore, this tutorial can be approached by the newest of users. However, as with all of the tutorials in this manual, familiarity with Gridgen's GUI is recommended (Section 2 of the User Manual).

12.4 Geometry

The geometry to be gridded is a 3D pipe with four mixing baffles and is shown in Figure 27.1 in shaded rendering. The straight pipe surfaces are outlined to make the internal baffles visible. The surface representing the inlet is not shown. This model has been saved in Gridgen's native database format, composite format, and is completely closed. This grid will be unstructured, so the topology of the grid will essentially match the topology of the geometry.
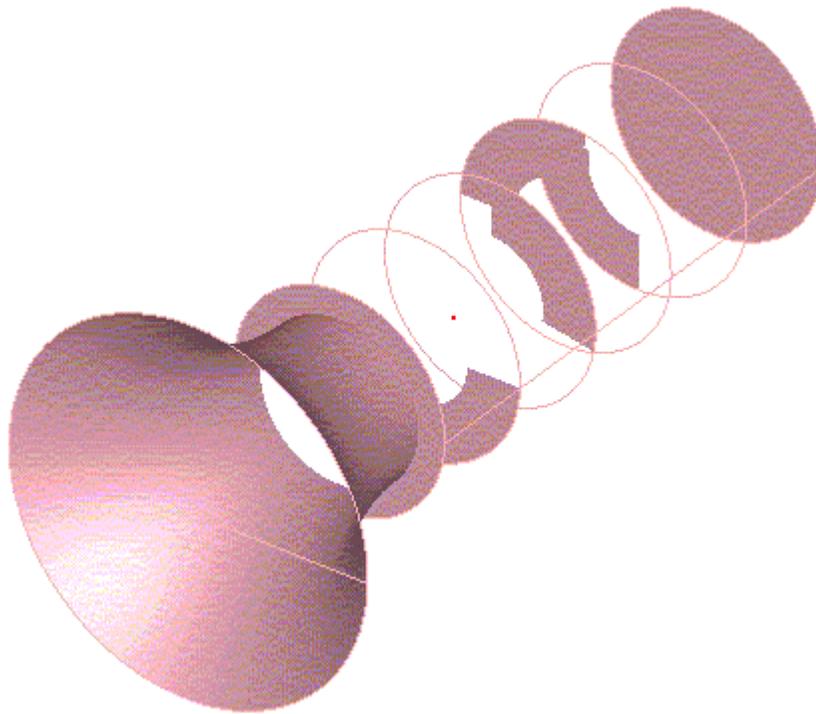
Figure 27.1 Mixer Geometry

12.5   Getting Started

Begin by starting Gridgen, either by typing the string `gridgen` on Unix workstations or simply double-clicking the Gridgen shortcut on your PC's desktop. The first step in running Gridgen is to select the software (Section 17.1 of the User Manual) to be used in the analysis. This will ensure the grid generated by Gridgen is consistent with the capabilities of the analysis software. For this analysis we will use **Fluent**. To make this solver setting, begin at the MAIN MENU:

1. **Analysis S/W**

2. **Select Analysis S/W**

3. **3D**

4. **Fluent**

5. **Next Page**

6. **Done**

7. **Done**

To begin the tutorial, import the composite geometry file. The file should be available in your Gridgen installation directory under *gridgen_home_path*/`examples/tutorial/mixer`. You may wish to copy the file to some other working directory before proceeding. From the MAIN MENU:

1. **Input/Output**

2. Database **Import**

3. Use the Browser to navigate to the directory containing the copy of `mixer.dba` and select it.

4. **Open**

To clean up the display of the model, go to the DISPLAY COMMANDS menu:

1. **Disp**

2. Toggle off Show **EnbldDB**.

3. Toggle off **RotAxs**

4. **Done**

You will see the model essentially disappear while in the DISPLAY COMMANDS menu. However, once this menu is exited, the entities return in an outline (demoted) mode. After some image manipulations ([Section 2.7](#) of the User Manual), your model should appear as shown in Figure 27.2.
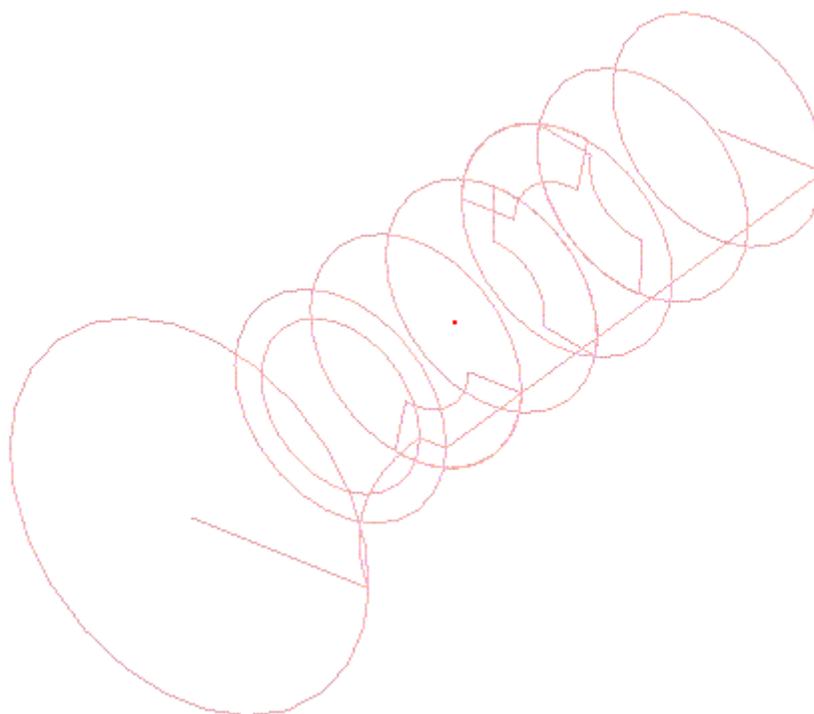
Figure 27.2 After Display Clean-Up

12.6   Setting Defaults

You will need to set one default value to facilitate the use of the **On DB Entities** command for domain creation. For **On DB Entities** to be available at all, you must set either a default dimension (Con Dim **dimen**) or average spacing (Con Dim **avg Δs**).

In the SET DEFAULT VALUES menu:

1. Con Dim **avg Δs**

2. `1.5`

3. **Done**

This setting instructs Gridgen to calculate the number of grid points necessary on each newly created connector in order to achieve an approximate spacing of 1.5 between equally spaced grid points.

12.7   Creating Unstructured Domains

We will use Gridgen's **On DB Entities** command to create essentially all of the topology for this tutorial. Once created, we will look at some of the issues with the resulting connectors.

Beginning on the DOMAIN COMMANDS menu:

1. **Create**

2. Cell Type **unstructured**

3. Turn on the **Auto Merge** toggle to have coincident connectors merged automatically.

4. **Set Toler**

5. `0.5`

Note that Gridgen will use an iterative process to merge connectors during the **On DB Entities** command execution. Merging will start with the default connector tolerance as shown in the SET TOLERANCE VALUES Blackboard and will gradually increase to the tolerance value specified here. Therefore, relatively large values of merge tolerance can be specified without worry of topology destruction.

6. **On DB Entities**

7. Pick **All**

8. **Done**

To make the display of the grid less cluttered, change the render attributes of the domains in the DISPLAY COMMANDS menu.

1. **Disp**

2. Edit Dspla **Dom**

3. Pick **All**

4. **Done**

5. display style **Off**

6. **Done - Apply Atts**

7. **Abort**

8. **Done**

When you are done the grid should look like the one shown in Figure 27.3.
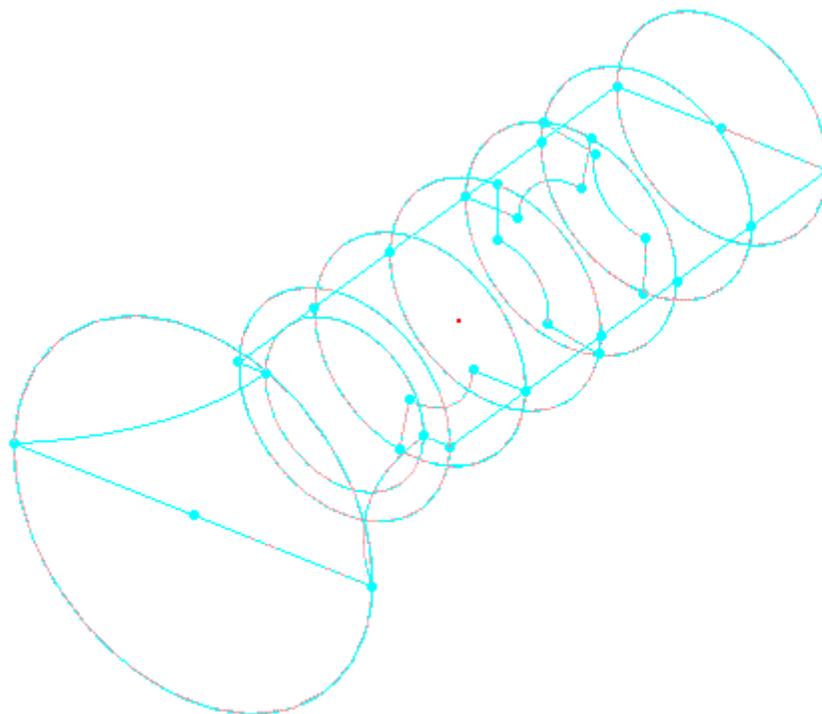
Figure 27.3 All Domains After Changing Display Attributes

12.8   Additional Connector Merging

Proceed to the CONNECTOR COMMANDS menu, set TopoFltr to **NonMnfld+Free**, and use the Merge **Pick** command to note that the baffle domains have connectors overlapping those from the cylinder domains. These were not automatically merged during the **On DB Entities** command since implementation of the merging tools during **On DB Entities** will stop considering connectors once they become part of a manifold edge, effectively disallowing baffles.

Press the **Abort - Done Merging** button to exit from the Merge **Pick** function, set TopoFltr back to **All**, and simply use Merge **Auto** to have Gridgen look for candidates automatically based on your input tolerance and topology filter settings:

1.   TopoFltr **All**

This choice ensures Gridgen will consider all connectors for merging regardless of their use in existing domain connections.

2.   Merge **Auto**

3.   `0.5`

4.   **Update**

Now there are no longer any overlapping connectors and we are ready to proceed to block construction.

12.9   Creating Unstructured Block with Baffle Faces

Completing the grid simply involves creation of a single unstructured block which will consist of one closed face and four baffle faces. Create the block starting from the BLOCK COMMANDS menu.

1. **Create**

2. Cell Type **unstructured**

3. **Assemble Faces**

4. **Add 1st Face**

5. AutoSave **Faces**

This instructs Gridgen to automatically save completely closed faces and baffle faces.

6. Turn off the **NonMnfld Cnnct** toggle so that Gridgen will only allow manifold connections between domains and properly differentiate between the tube face and baffle faces.

7. Pick **All Domains**

Gridgen immediately beeps and indicates in the Message Window that "A total of 1 closed-surface faces and 4 baffle-faces were assembled into the block". After hitting any key to continue, note that the closed face has normals rendered, as shown in Figure 27.4, which correctly point into the interior. Therefore, it is not necessary to use **Reorient Faces**.

8. **Done-Save Blocks**
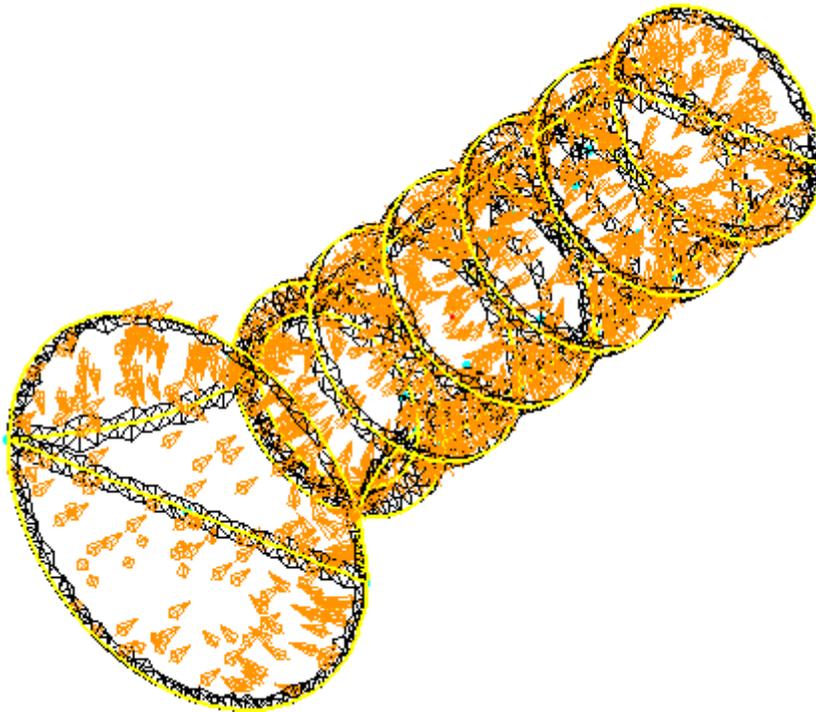
Figure 27.4 Block Face Normals

Also in the BLOCK COMMANDS menu:

1. Run Solver **Unstrctrd**

2. Pick **All**

3. **Done**

4. **Initialize**

5. **Done - Save**

Now use the **Examine** command to have a look at the interior of your new unstructured block. Move a Phy **X**, **Y**, or **Z** Hiliter plane to any position along the model and have Gridgen calculate a Ref Surf **Crinkle** reference surface, for instance. Try to produce reference surfaces that reveal the presence of the interior baffle surfaces.

12.10 Setting Flow Solver Boundary Conditions

To finish this tutorial we will set Fluent boundary conditions.

From the ANALYSIS S/W COMMANDS menu:

1. **Set BCs**

Note there are "Type 1" interfaces specified automatically, as shown in Figure 27.5. These correspond to both sides of each of the baffles, so each corresponding domain number (in parenthesis) is shown twice, once per side. Placing the cursor over each side to hilite it will result in Gridgen hiliting the domain while rendering the normals for that side in the Display Window. Since Gridgen intends to treat these baffles as connections by default, we can use baffles to simply provide interior cluster control and not necessarily use them as physical boundaries. For this tutorial, however, we will set the baffles to solid boundaries.

2. Select all of the domains, including the baffles, except the inlet and exit domains.

3. **Done**

4. **Wall**

5. Select the two inlet patches at the conical end.

6. **Done**

7. **Pressure Inlet**

8. Select the two remaining outlet patches at the other end.

9. **Done**

10. **Outflow**

11. **Done**
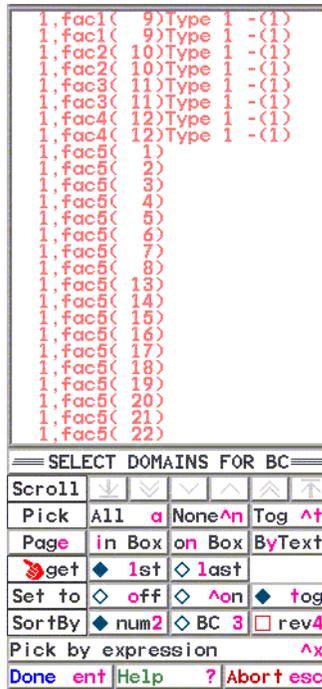
Figure 27.5 Boundary Condition Browser

This concludes our tutorial. You are free to use the **Export Analysis Data** command to export a Fluent case (.cas) file.

13. Converging-Diverging Nozzle: Using Glyph Scripting

13.1 Introduction

This section includes a brief tutorial to help you become familiar with Gridgen's scripting language, Glyph, and the tools available to work with it.

13.2 Topics Covered

Gridgen skills you will learn during this tutorial session are:

• executing and appending to an existing script using **Append to Journal**

• journaling your interactive commands to a script file for later editing and playback

• using **Add Variable** to create variables for easier interaction and easier script playback

• editing and executing a journaled script to generate a new grid

• creating extruded blocks using **Extrude From Doms**

13.3 Background

Familiarity with basic Gridgen usage would be advantageous, so it is recommended you work at least the second tutorial in Section 2. Also, familiarity with Gridgen's GUI is recommended (Section 2 of the User Manual). Finally, a complete volume, the Glyph Reference Manual , is devoted to the Glyph language and will be a valuable resource for this tutorial and all scripting work (Section 1 of the Glyph Reference Manual).

13.4 Geometry

The geometry to be gridded is a 3D, 3-block, structured grid for 1/4 symmetry of an axi-symmetric, converging-diverging nozzle. The grid will be generated by sweeping the symmetry plane surface grids (Figure 28.1) through 90 degrees of rotation about the centerline axis.
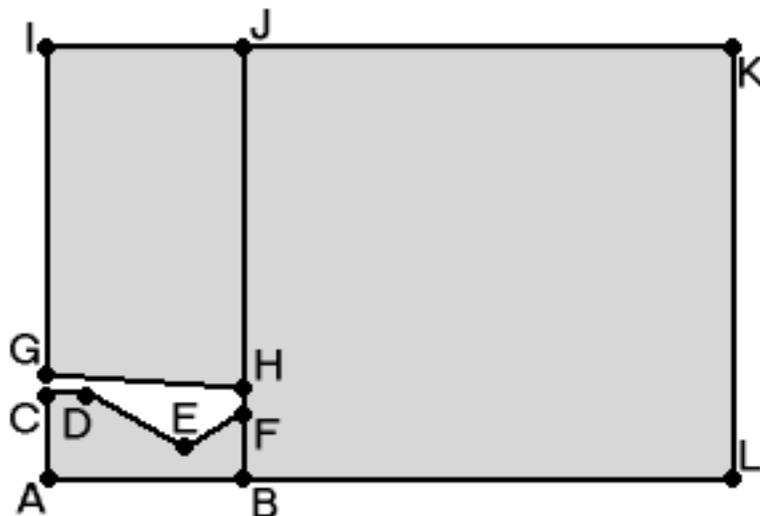
Figure 28.1 Nozzle Geometry

13.5   Getting Started

Because the focus of this tutorial is scripting and not building the grid, a script is provided as your starting point. This script (`nozzle.glf`) was generated by journaling creation of the symmetry plane's connectors and domains to a file. You will use this file to get started and then append your work to it. Begin by starting Gridgen, either by typing the string `gridgen` on Unix workstations or simply double-clicking the Gridgen shortcut on your PC's desktop. We will select the analysis software package to be used at the end of the tutorial.

The Glyph file should be available in your Gridgen installation directory under *gridgen_home_path*/`exam-ples/tutorial/glfnoz`. Since journaling using the **Append to Journal** feature will modify the selected file, you must create a copy of the original Glyph file, `nozzle.glf`, and append to the copy. It is not likely that you would have write permission where the original is installed, and the original should be preserved for other users who wish to work this tutorial. (Another Glyph script file, `noz-all.glf`, is included that generates the entire grid for this tutorial.) From the MAIN MENU:

1. **Glyph**

2. **Append to Journal**

3. Use the Browser to navigate to the directory containing the copy of `nozzle.glf` and select it.

4. **Open**

Watch as Gridgen builds the connectors and domains on the symmetry plane. Note the orange text "`JOUR-NALING TO nozzle.glf`" in the Display window's upper left corner. This is a reminder that the script equivalents of commands you execute will be appended to the file `nozzle.glf` and will continue to be journaled until you use **End Journaling** in the GLYPH COMMANDS menu. There is a small percentage of Gridgen's commands that cannot yet be journaled. Commands not available for journaling will be grayed out in the Gridgen interface.
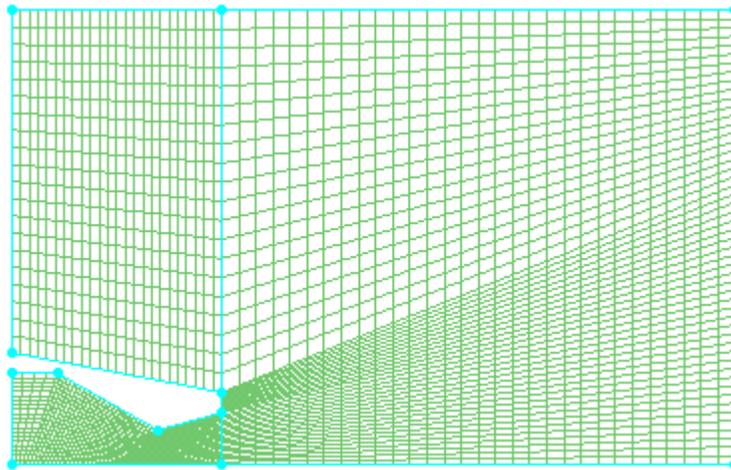
Figure 28.2 Connectors and Domains After Script Execution

As you can see in Figure 28.2, the spacing constraints on the connectors have not been set. This is what you will do by creating and using variables.

13.6   Setting Variables for Spacing Constraints

You are going to set three variables for the values of the grid clustering at various locations in the grid. Variables will make it easy to edit the journaled script and change the value of the clustering for generating another grid. The spacings you will set are:

- wall spacing (`DSw`)

- inlet spacing (`DSi`)

- spacing at node D (`DSd`)

A Gridgen variable can have any value: scalar, vector, integer, floating point, or character. You can define the variable using any valid Tcl expression including arithmetic operations. For this tutorial, the three variables you create will be floating point scalars.

While still in the GLYPH COMMANDS menu:

1. **Add Variable**

Note that the Browser lists variables that have already been defined. These came from the script you started with, `nozzle.glf`.

2. `DSw`

Now the ADD VARIABLE menu shows the seven methods for setting a variable's value.

3. **Enter Tcl Expression**

4. `0.2`

5. **Add Variable**

6. `DSd`

7. **Enter Tcl Expression**

8. `1.0`

9. **Add Variable**

10. `DSi`

11. **Enter Tcl Expression**

12. `1.5`

13.7   Using Variables

You may now go to the CONNECTOR COMMANDS menu and use these variables to set the clustering at the break points according to Figure 28.3. You will use the **Set Δs Vals** command. This command conveniently allows you to select, either in the Browser or graphically, multiple break points at once and set them to the same spacing constraint value. Since you have three spacing variables to apply, you will only need to select and specify three groups of break points.
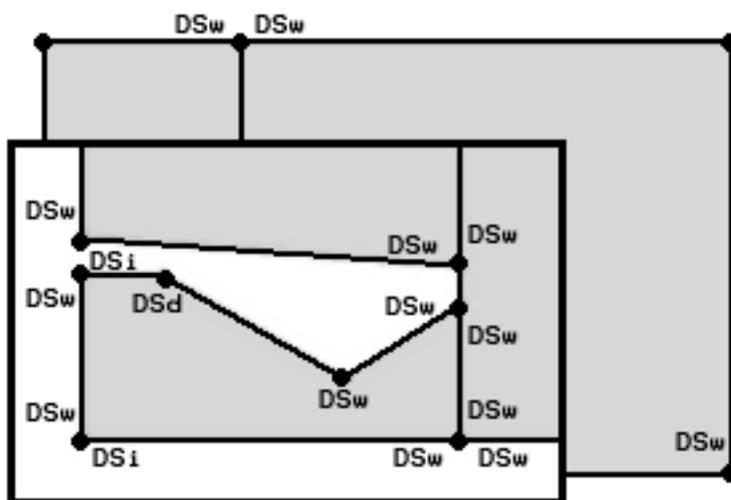
Figure 28.3 Locations to Apply Variables

Start with break points corresponding to `DSi`. Beginning on the CONNECTOR COMMANDS menu:

1. **Set Δs Vals**

2. Select the two break points corresponding to `DSi` in Figure 28.3.

3. **Done**

Note that the text input window is hilighted and expecting you to type in the desired spacing value. Also, note that the Browser contains a list of all variables defined by the journaled script and the three you just created.

4. Select `DSi` from the Browser and enter.

5. Use Figure 28.3 to apply spacing constraints all around the rest of the grid.

6. **Done**

When you are done the grid should look like the one shown in Figure 28.4.
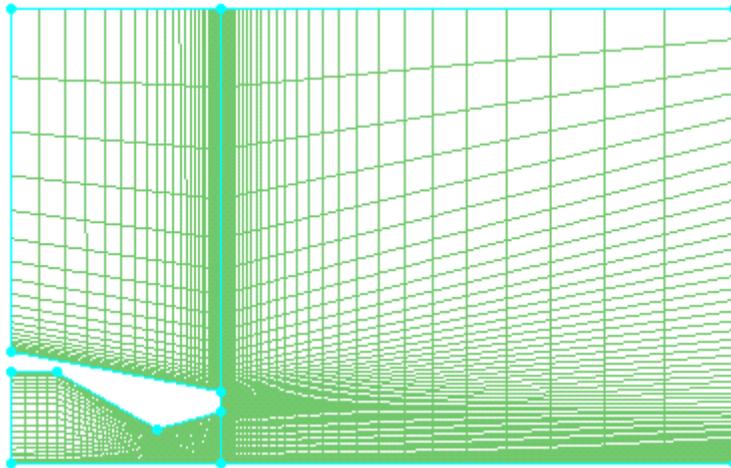
Figure 28.4 Spacing Constraints Applied

To complete the symmetry plane grid you should run the elliptic PDE solver on the domain representing the nozzle's interior. Proceed to the DOMAIN COMMANDS menu.

1. Run Solver **Structured**

2. Select the interior nozzle domain, <u>ABFEDC</u>.

3. **Done**

4. **Set Solver Attribs**

5. **Foreground Cont Funcs**

6. Select edge <u>BF</u> by filling the colored square corresponding to that edge.

7. Edges **Set**

8. αCalc **Interpl**

9. **Done Setting Attributes**

Note that because you are journaling you cannot simply use Elliptic Slvr **Run**, you must enter a number of iterations.

10. Elliptic Slvr **Run N Iters**

11. `100`

12. **Done - Save**

Finally, you may end the journaling of your work by returning to the GLYPH COMMANDS menu and pressing **End Journaling**.

13.8   *Script Editing and Playback*

You can now edit some of the variables in the journaled script you just made (`nozzle.glf`) and execute the script to get a different grid. Edit `nozzle.glf` with your favorite text editor. At the top you will see several initialization commands. These are followed by variables defining various distances, radii, and 3D points throughout the grid. Notice that everything has been non-dimensionalized by the nozzle radius, a variable named `Rth`.

Assume you want to make the nozzle's exit smaller. Therefore, you have to reduce the value of the nozzle exit radius variable `re`. Find where this variable is set by searching for the string "`Gridgen Journal Variable: re`". You will see, as in Figure 28.5, that its value is set to `2.19`. Change its value to `1.5` and save the file `nozzle.glf`.

```
gg::updatePolicy DISPLAY_ONLY

# Gridgen Journal Variable: Rth
set Rth 3.76
# Gridgen Journal Variable: l1
set l1 1.33
# Gridgen Journal Variable: l2
set l2 2.93
# Gridgen Journal Variable: l3
set l3 1.86
# Gridgen Journal Variable: le
set le [expr $l1+$l2+$l3]
# Gridgen Journal Variable: l4
set l4 15
# Gridgen Journal Variable: ri
set ri 2.66
# Gridgen Journal Variable: re
set re 2.19
# Gridgen Journal Variable: ro
```

Figure 28.5 Excerpt From File "`nozzle.glf`"

In order to see the grid being regenerated, you should clear everything from Gridgen. From the MAIN MENU press **Restart Gridgen** followed by **Restart**. Go to the GLYPH COMMANDS menu and press **Execute Script**. Use the Browser to select the file `nozzle.glf` and the script will be rerun. Notice the smaller nozzle exit.

You can experiment with changing other variables in `nozzle.glf` to make changes to the grid. You may press the **Re-execute Last Script** button to bypass having to select the file name using the Browser. You do not have to **Restart** Gridgen each time because of the restart commands that are included at the top of `nozzle.glf`.

13.9   Completing the Grid

Completing the grid for this nozzle involves using rotational extrusion, setting boundary conditions, and exporting the files. Before continuing, you should start journaling again by pressing **Append to Journal** from the GLYPH COMMANDS menu and selecting the file `nozzle.glf`. Create the 3 blocks starting from the BLOCK COMMANDS menu:

1. **Create**

2. Cell Type **structured**

3. Extrude **From Doms**

4. Pick **All**

5. **Done**

6. Type **Rot**

7. **Set Attributes**

8. **Total Rotation Angle**

9. **Enter via Keybd**

10. `90.0`

11. **Set Rotation Axis**

12. **Use X-Axis**

13. **Done**

14. **Run N**

15. `31`

16. **Done**

Next, go to the ANALYSIS S/W COMMANDS menu and choose your solver and set some boundary conditions:

1. **Select Analysis S/W**

2. **FDNS/UNIC**

3. **Next Page**

4. **Done**

5. **Confirm - Make Changes**

Using the **Set BCs** command, apply boundary conditions for the CFD solver.

Finally, export your grid and boundary condition data for FDNS:

1. **Export Analysis BCs**

2. Type In... **Name**

3. `noz-fdns`

Note that Gridgen will automatically append the correct file extension.

4. **Export Analysis Grid**

5. Type In... **Name**

6. `noz-fdns`

The grid is now complete, so go to the GLYPH COMMANDS menu and **End Journaling**. It is worth noting you do not need to save a Gridgen (.gg) restart file since this grid can be re-created entirely from the script.
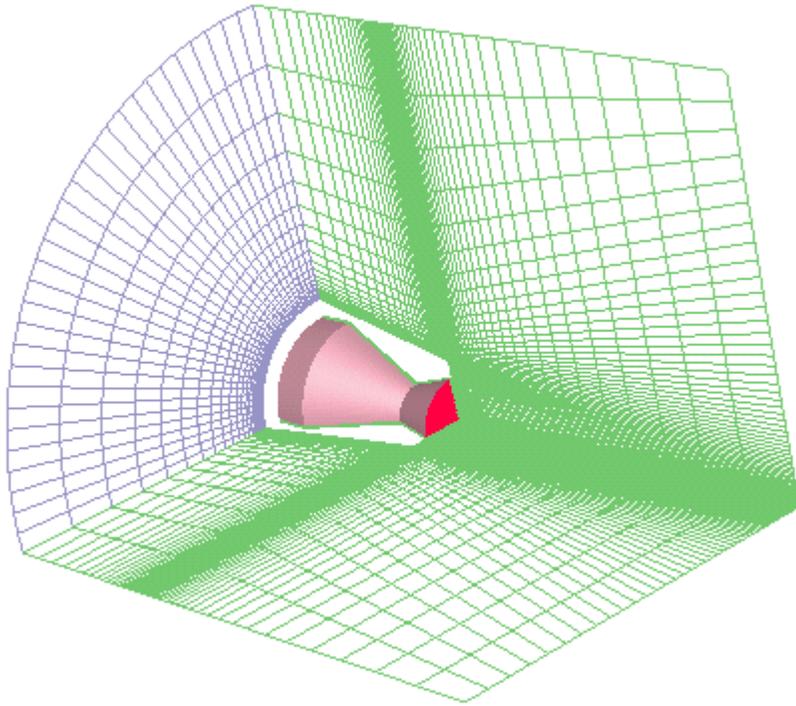
Figure 28.6 Completed Grid

14. Mechanical Part: Basic FEA Mesh

14.1 Introduction

The purpose of this tutorial is to introduce you to Gridgen's **On DB Entities** domains command for unstructured surface grid creation.

14.2 Topics Covered

Gridgen skills you will learn during this tutorial session are:

- setting defaults appropriate for unstructured grids using the **Defaults** command

- using the **On DB Entities** domains command to create unstructured domains

- using **Auto Join** and **Auto Merge** with **On DB Entities** to improve domain topology

- using Run Solver **Unstrctrd** for domain decimation

- building an unstructured block quickly and initializing it with tetrahedra

14.3 Background

The procedures followed in this tutorial use some of Gridgen's most automated commands. Therefore, this tutorial can be approached by the newest of users. However, as with all of the tutorials in this manual, familiarity with Gridgen's GUI is recommended (Section 2 of the User Manual).

14.4 Geometry

The geometry to be meshed is a 3D, mechanical part in IGES file format as shown in Figure 26.1. This grid will be unstructured, so the topology will essentially match the topology of the geometry, with the exception that tools will be used to slightly reduce the number of total domains.
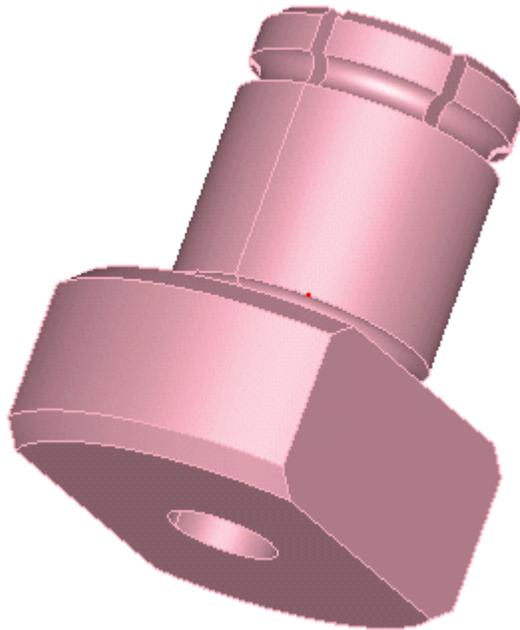
Figure 26.1 Mechanical Part Geometry

14.5   Getting Started

Begin by starting Gridgen, either by typing the string `gridgen` on Unix workstations or simply double-clicking the Gridgen shortcut on your PC's desktop. The first step in running Gridgen is to select the software ([Section 17.1](#) of the User Manual) to be used in the analysis. This will ensure the grid generated by Gridgen is consistent with the capabilities of the analysis software. For this analysis we will simply leave the analysis software at the default setting of generic 3D which should be visible in the status window at the upper left hand corner of the Gridgen interface.

To begin the tutorial, import the IGES geometry file. The file should be available in your Gridgen installation directory under *gridgen_home_path*/`examples/tutorial/dbdom`. You may wish to copy the file to some other working directory before proceeding. From the MAIN MENU:

1. **Input/Output**

2. Database **Import**

3. Use the Browser to navigate to the directory containing `dbdom.igs` and select it.

4. **Open**

After some image manipulations ([Section 2.7](#) of the User Manual), your model should appear as shown in Figure 26.2.
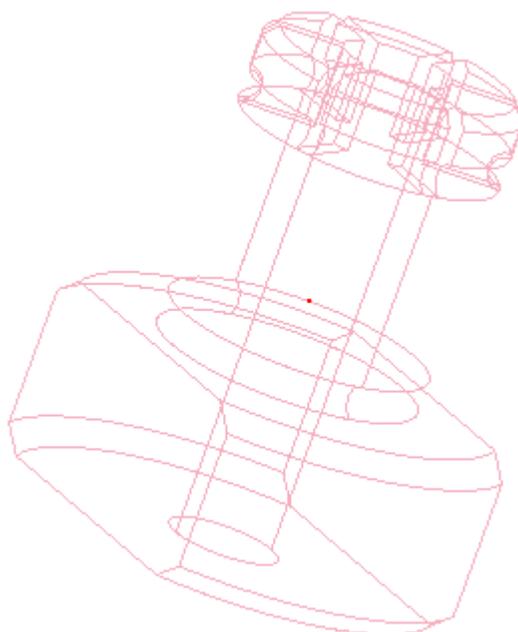
Figure 26.2 After Display Clean-Up

14.6   Setting Defaults

You will need to set three default values to facilitate the use of the **On DB Entities** command. First, in order for the command to be available at all, you must set either a default dimension (Con Dim **dimen**) or average spacing (Con Dim **avg** Δ**s**).

1. **Dflt**

2. Con Dim **avg** Δ**s**

3. `0.05`

This setting instructs Gridgen to calculate the number of grid points necessary on each newly created connector in order to achieve an approximate spacing of 0.05 between grid points. Gridgen can apply more grid points to high curvature areas of connectors to improve geometric resolution.

4. Con Dim **Max Ang**

5. `5.0`

This setting tells Gridgen there can be a turn angle between three consecutive grid points of no more than 5 degrees. If the angle is greater than 5 degrees after applying the above average spacing, Gridgen will insert additional points until the maximum angle criteria is met. Now do the same for new surface grids.

6. UnstrDom **MaxAng**

7. `10.0`

Now Gridgen will calculate the difference in angle between a new unstructured domain's cell normals versus the underlying database normals at the cell vertices. If larger than 10 degrees, additional points will be inserted into the surface grid until the maximum angle criteria is met. These angle criteria will allow us to

better capture the shape of the tight radius surfaces in this model.

8. **Done**

14.7   Creating Unstructured Domains, First Pass

Since the purpose of this tutorial is to illustrate the use of Gridgen's **On DB Entities** command, the domains will be created twice. First, the basic command will be used with no additional clean-up enabled. Once created, we will look at some of the issues in the resulting domains.

From the DOMAIN COMMANDS menu:

1. **Create**
2. Cell Type **unstructured**
3. **On DB Entities**
4. Pick **All**
5. **Done**

To make the new domains more visually accessible, change some of their display attributes in the DISPLAY COMMANDS menu.

1. **Disp**
2. Edit Dspla **Dom**
3. Pick **All**
4. **Done**
5. display style **Shaded/Wireframe**
6. wireframe color "red"
7. solid/shade color "purple"
8. **Done - Apply Atts**
9. **Abort**
10. **Done**

When you are done, the grid should look like the one shown in Figure 26.3.
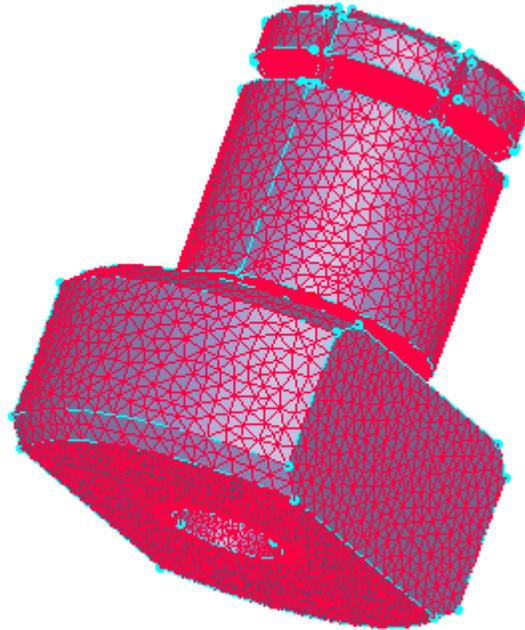
Figure 26.3 All Domains After Changing Display Attributes

Now note a few aspects of these 58 domains. First, there are three areas where Gridgen followed the topology of the underlying database and created two separate domains rather than a single 360 degree domain wrapping around the geometry. In terms of defining the shape properly, a single domain would have been sufficient in each case.

To observe the second, more serious issue, proceed to the CONNECTOR COMMANDS menu:

1. Merge **Conns**

2. `0.01`

Note there are several pairs of coincident connectors in the Browser list. It means that the mesh is not closed due to gaps in the database. Normally this kind of issue only becomes evident or a problem at the block face creation stage of the grid. However, these could have been eliminated at the domain creation step, as you will see in the next section, allowing the problem to be avoided altogether. While in the MERGE CON-NECTORS Browser, note some pairs of connectors are listed with a dash (-) between them and some with an exclamation point (!). Dashes indicate connector pairs with equal number of grid points; exclamations indicate nonequal connector dimension. Simply **Abort** from the MERGE CONNECTORS Browser since we will not be continuing with these domains.

14.8   Creating Unstructured Domains, Second Pass

Now we will restart this Gridgen session and create the domains again using some additional functionality.

From Gridgen's MAIN MENU:

1. **Restart Gridgen**

2. Turn off the **Clear DB** toggle since we wish to keep our database model.

3. Turn off the **Reset Defaults** toggle since we wish to use the same default settings.

4. **Restart**

Now the grid should be cleared. Proceed back to the DOMAIN COMMANDS menu.

1. **Create**

2. Turn on the **Auto Join** toggle to automatically join adjacent domains whose turning angle at their common edge is less than the **Set Angle**.

3. **Set Angle**

4. `40.0`

Note that we do not want to use the default "Any" for our join angle (which is shown in the SET DEFAULT VALUES menu) since this will instruct Gridgen to join domains regardless of turning angle between them and will result in almost all domains in this model being joined.

5. Turn on the **Auto Merge** toggle to have coincident connectors merged automatically.

6. **Set Toler**

7. `0.01`

8. **On DB Entities**

9. Pick **All**

10. **Done**

Use the DISPLAY COMMANDS menu again to set attributes for the new domains.

1. **Disp**

2. Edit Dspla **Dom**

3. Pick **All**

4. **Done**

5. display style **Shaded/Wireframe**

6. wireframe color "red"

7. solid/shade color "purple"

8. **Done - Apply Atts**

9. **Abort**

10. **Done**

Now the grid should look like the one shown in Figure 26.4. Note there are now 54 domains. There are no coincident connectors, and the 360 degree domains are not just simply joined, but also smoothed and decimated. The original database seams are no longer evident.
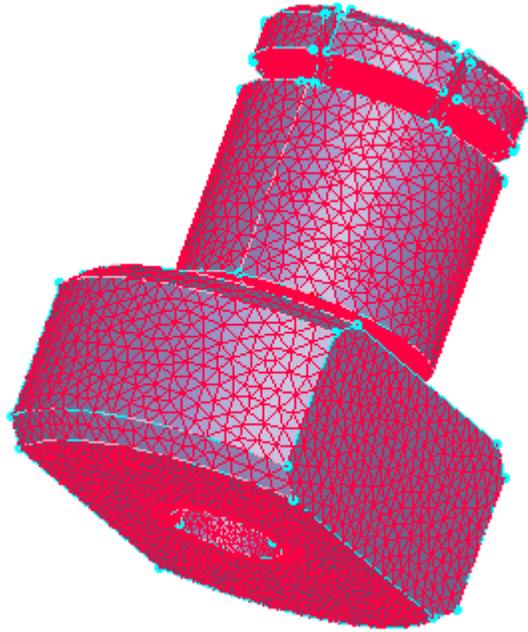
Figure 26.4 All Domains With Additional Clean-up

14.9 Domain Decimation

There are many planar domains in this model, but three particularly large ones on the "base" of the model have significantly denser triangulation than necessary. We will use the domain unstructured solver to decimate these domains.

1. Run Solver **Unstrctrd**

2. Select domains 1, 2, and 52.

3. **Done**

4. **Set Solver Attribs**

5. **Grid Control Params**

6. Triangle **Min Size**

7. `0.01`

8. Triangle **Max Size**

9. `0.1`

10. **Done Setting Attributes**

11. Tri Solver **Decimate**

12. **Done - Save**

The three domains should now be substantially coarser on the interior, which will allow a savings on tetrahedra count in the eventual volume grid. Your grid should now appear similar to the one in Figure 26.5.
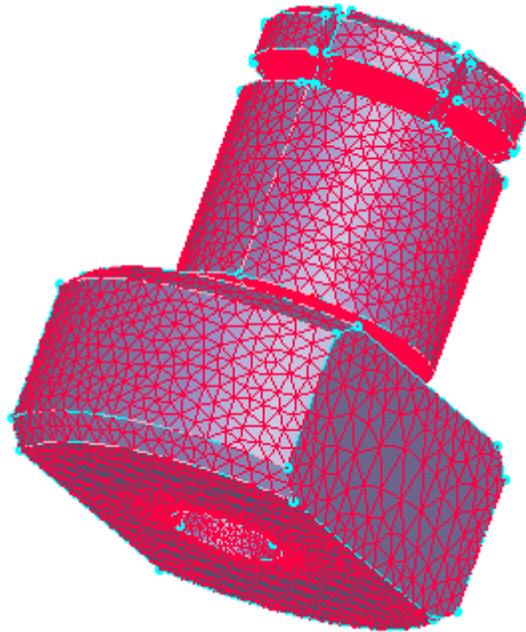
Figure 26.5 Domains After Decimation Pass

14.10 Completing the Grid

Completing the grid for this part simply involves creation of a single unstructured block and using the block unstructured solver to initialize the interior with tetrahedra.

Create the block starting from the BLOCK COMMANDS menu.

1. **Create**
2. Cell Type **unstructured**
3. **Assemble Faces**
4. **Add 1st Face**
5. Pick **All Domains**

Note there are no bold yellow outlines of domain edges which are used to indicate the boundaries of an incomplete face.

6. **Save the Face**

Now note the arrows rendered pointing away from the domains making up the block's face. These represent the face normals and should always point into the interior of the volume or solid you wish to analyze. Gridgen should automatically orient these in most cases, but verify their orientation in the Display window and use **Reorient Faces** to reverse the orientation if necessary.

7. **Done-Save Blocks**

Also in the BLOCK COMMANDS menu.

1. Run Solver **Unstrctrd**

2. Pick **All**

3. **Done**

4. **Initialize**

5. **Done - Save**

Now use the **Examine** command to have a look at the interior of your new unstructured block.

1. **Examine**

2. Pick **All**

3. **Done**

4. Phy **Z**

5. **Z kbrd**

6. `0.0`

7. Ref Surf **Crinkle**

You should now see a reference surface like the one shown in Figure 26.6. You may also wish to examine some particular quality measure using Function **None** to go to the DIAGNOSTIC menu and choosing a specific quality measure for Gridgen to calculate.

This concludes our tutorial. Feel free to try other angles and tolerances for the **On DB Entities** command to see what other results you can obtain on this geometry.
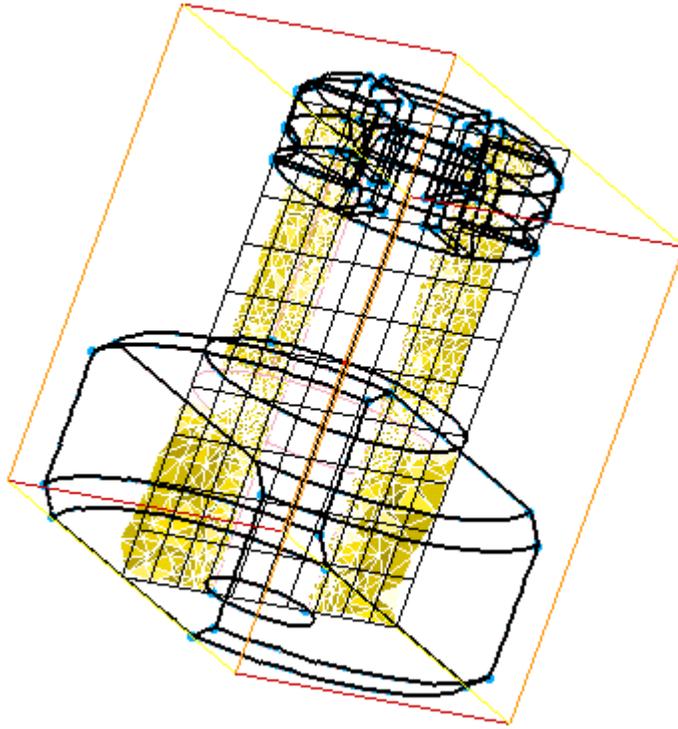
Figure 26.6 Crinkle Reference Surface in Examine

15. Index