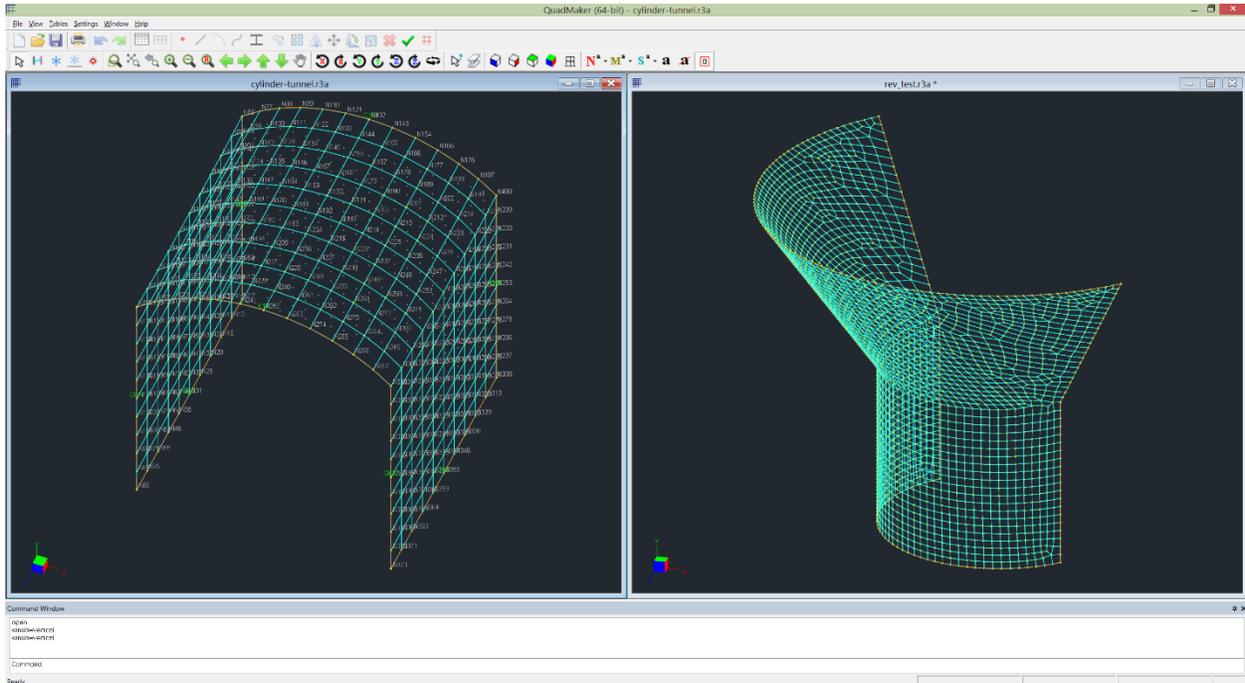


QuadMaker

A 100% Quadrilateral Mesh Generator



User Manual

Computations & Graphics, Inc.
Highlands Ranch, CO 80130, USA

Email: info@cg-inc.com

Web: www.cg-inc.com

End User License Agreement for CGI Software (Non-SDK)

The Software is protected by United States copyright laws and various international treaties. By installing or using the Software, you agree to be bound by the terms of this Agreement. If you do not agree with the terms of this Agreement, do not install or use the Software. This Agreement is governed by the laws of the United States and the State of Colorado. You may not export the Software in violation of applicable export laws.

1. DEFINITIONS

“Software” means all of the contents of the files, disk(s), CD-ROM(s), or other media with which this Agreement is provided.

“Documentation” means all of the contents of the files, printed materials with which this Agreement is provided. “End User” means you.

“CGI” means Computations & Graphics, Inc.

2. GRANT OF LICENSE

a). The following applies if you have purchased a perpetual Software license:

CGI grants you (the End User) a non-exclusive, non-transferable license to use the Software on a single computer. You may not rent, lease, or resell the Software. You may not disassemble, decompile, reverse engineer, or modify the Software in any way. This License starts from the date you receive the Software and will last as long as the End User complies with the terms of this Agreement.

b). The following applies if you have purchased a subscription Software license:

CGI grants you (the End User) a non-exclusive, non-transferable license to use the Software simultaneously via the internet on a certain number of computers for a certain subscription period. You may not rent, lease, or resell the Software. You may not disassemble, decompile, reverse engineer, or modify the Software in any way. This License starts from the date you purchased the subscription license and will last for the subscription period.

3. SUPPORT

CGI offers limited 30 days of free email technical support related to the installation and use of the most recent version of the Software, starting from the start date of this Agreement. CGI has no obligation to provide support in any form if your version of the Software is not the most recent version. CGI, in its sole discretion, will determine what constitutes a support incident. CGI reserves the right to refuse support service to anyone.

4. COPYRIGHT

The Software and Documentation are the intellectual property of and are owned by CGI. You may make at most one copy of the Software and/or the Documentation for backup purposes.

5. COMMERCIAL USES

The Standard and Professional versions of the Software may be used for commercial purposes.

The Evaluation, Educational, and Beta versions of the Software may not be used for commercial purposes.

6. LIMITATION OF LIABILITY

IN NO EVENT WILL CGI OR ITS SUPPLIERS BE LIABLE TO YOU FOR ANY DAMAGES, CLAIMS, OR COSTS WHATSOEVER OR ANY CONSEQUENTIAL, INDIRECT, INCIDENTAL DAMAGES, OR ANY LOST PROFITS OR LOST SAVINGS, EVEN IF CGI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH LOSS, DAMAGES, CLAIMS, OR COSTS.

7. DISCLAIMER

CGI HAS TAKEN EVERY EFFORT TO MAKE THE SOFTWARE RELIABLE AND ACCURATE. HOWEVER, IT IS THE END USER’S RESPONSIBILITY TO INDEPENDENTLY VERIFY THE ACCURACY AND RELIABILITY OF THE SOFTWARE. NO EXPRESS OR IMPLIED WARRANTY IS PROVIDED BY CGI OR ITS DEVELOPERS ON THE ACCURACY OR RELIABILITY OF THE SOFTWARE.

8. REFUND POLICY

All sales are final and no refunds will be given. If you do not agree to and accept this policy, do not purchase the license of this software.

9. TERMINATION OF THIS LICENSE:

This Agreement becomes effective on the date you accept this Agreement and will continue until terminated as provided for in this Agreement. CGI may terminate this license at any time if you are in breach of any of its terms and conditions. Upon termination, you must immediately return to CGI or destroy the Software and all copies thereof.

Copyright

THE SOFTWARE **QUADMAKER** AND ALL ITS DOCUMENTATION ARE THE INTELLECTUAL PROPERTY OF AND ARE OWNED BY COMPUTATIONS & GRAPHICS INC. (CGI). ILLEGAL USE OF THE SOFTWARE OR REPRODUCTION OF ITS DOCUMENTATION IS STRICTLY PROHIBITED.

Disclaimer

CGI HAS TAKEN EVERY EFFORT TO MAKE THE SOFTWARE RELIABLE AND ACCURATE. HOWEVER, IT IS THE END USER'S RESPONSIBILITY TO INDEPENDENTLY VERIFY THE ACCURACY AND RELIABILITY OF THE SOFTWARE. NO EXPRESS OR IMPLIED WARRANTY IS PROVIDED BY CGI OR ITS DEVELOPERS ON THE ACCURACY OR RELIABILITY OF THE SOFTWARE.

Notice

SINCE **QUADMAKER** COMES IN DIFFERENT VERSIONS, SOME FEATURES DESCRIBED IN THIS DOCUMENTATION MAY NOT APPLY TO THE SPECIFIC VERSION OF THE PROGRAM YOU ARE RUNNING.

OpenGL® is a registered trademark of Hewlett Packard Enterprise.

Windows® is a registered trademark of Microsoft Corporation.

QuadMaker is a trademark of Computations & Graphics, Inc.

Copyright 2002-2025 by Computations & Graphics, Inc. All rights reserved.

Last Revised Feb. 2025

Table of Contents

END USER LICENSE AGREEMENT FOR CGI SOFTWARE (NON-SDK)	I
COPYRIGHT	II
DISCLAIMER	II
NOTICE.....	II
INTRODUCTION	7
<i>Graphical User Interface (GUI)</i>	1
<i>Command Window</i>	1
<i>Enter Nodal Coordinates</i>	2
<i>Mouse Use</i>	2
<i>Spreadsheet Navigation</i>	3
<i>System Requirements</i>	4
TECHNICAL BACKGROUND.....	5
<i>Nodes</i>	6
<i>Curves</i>	6
<i>Edges</i>	6
<i>Regions (aka sub-regions)</i>	6
<i>Plane Region</i>	7
<i>Holes</i>	8
<i>Internal Points</i>	8
<i>Trees</i>	9
<i>Cylinder Region</i>	11
<i>Sphere Region</i>	12
<i>Rev Region</i>	14
<i>General Patch Region</i>	15
MENUS.....	16
CHAPTER 1: FILE MENU.....	17
<i>New</i>	17
<i>Open</i>	17
<i>Close</i>	18
<i>Save</i>	18
<i>Save As</i>	18
<i>Save All</i>	18
<i>Export Nodes & Shells</i>	18
<i>Print Setup</i>	19
<i>Print Current View</i>	19
<i>General Information</i>	19
<i>View Log File</i>	19
<i>Open Containing Folder</i>	20
CHAPTER 2: EDIT	21
<i>Undo</i>	21
<i>Redo</i>	21
<i>Duplicate</i>	21
<i>Array</i>	21
<i>Mirror</i>	22
<i>Move</i>	22
<i>Rotate</i>	23
<i>Scale</i>	23
<i>Delete</i>	24
<i>Extrude Extrude Nodes to Curves</i>	24
<i>Merge All Nodes</i>	24
<i>Remove All Orphaned Nodes</i>	25
<i>Remove Unused Curves</i>	25
<i>Switch Coordinates</i>	25

<i>Clear / Clear Undo & Redo</i>	25
<i>Clear / Clear Everything</i>	25
CHAPTER 3: VIEW	26
<i>Redraw</i>	26
<i>Restore Model</i>	26
<i>Preset Views</i>	26
<i>Named Views</i>	26
<i>Named Selections</i>	26
<i>Zoom / Zoom Extent</i>	27
<i>Zoom / Zoom Window</i>	27
<i>Zoom / Zoom Object</i>	27
<i>Zoom / Zoom Previous</i>	27
<i>Zoom / Zoom In</i>	28
<i>Zoom / Zoom Out</i>	28
<i>Pan / Pan Screen</i>	28
<i>Pan / Left</i>	28
<i>Pan / Right</i>	28
<i>Pan / Up</i>	28
<i>Pan / Down</i>	28
<i>Rotate / +X</i>	28
<i>Rotate / -X</i>	29
<i>Rotate / +Y</i>	29
<i>Rotate / -Y</i>	29
<i>Rotate / +Z</i>	29
<i>Rotate / -Z</i>	29
<i>Real Time Motion / Real-Time Pan</i>	29
<i>Real Time Motion / Real-Time Zoom</i>	29
<i>Real Time Motion / Real-Time Rotate</i>	30
<i>Window/Point Select</i>	30
<i>Line Select</i>	30
<i>Select by IDs / Nodes</i>	30
<i>Select by IDs / Curves</i>	31
<i>Select by IDs / Shell4s</i>	31
<i>Select by IDs / Select All</i>	31
<i>Select by IDs / Unselect All</i>	31
<i>Select by Properties / Curve Steps</i>	31
<i>Select by Properties / Orphaned Nodes</i>	32
<i>Select by Properties / Coordinates</i>	32
<i>Select by Properties / Selection Names</i>	32
<i>Select by Properties / Select All</i>	33
<i>Select by Properties / Unselect All</i>	33
<i>Flip Selection</i>	33
<i>Freeze Selected</i>	33
<i>Freeze All Except Selected</i>	33
<i>Thaw</i>	34
<i>Mesh Model</i>	34
<i>Annotate</i>	34
<i>Clear Annotations</i>	34
<i>Query</i>	34
<i>Distance</i>	35
<i>Render / Render Options</i>	35
<i>Render / Quick Render</i>	35
<i>Options / Drawing Grid</i>	36
<i>Options / Global Axes</i>	36
CHAPTER 4: GEOMETRY	37
<i>Curve Steps</i>	37
<i>Drawing Grid</i>	37
<i>Object Snap</i>	38
<i>Draw Node</i>	38
<i>Draw Line</i>	39

<i>Draw Arc</i>	39
<i>Draw Spline</i>	40
<i>Quick Draw Line</i>	40
<i>Quick Draw Arc</i>	40
<i>Quick Draw Spline</i>	41
<i>Generate Lines</i>	41
<i>Generate Lines Rectangular</i>	41
<i>Generate Cylindrical</i>	42
<i>Generate Lines Arc</i>	43
CHAPTER 5: REGIONS	44
<i>Add Plane Region</i>	44
<i>Add Cylinder Region</i>	44
<i>Add Sphere Region</i>	45
<i>Add Rev Region</i>	45
<i>Add General Patch Region</i>	45
<i>Add Hole</i>	46
<i>Add Internal Points</i>	46
<i>Add Tree</i>	47
<i>Edit Regions</i>	47
<i>Delete Region</i>	47
<i>Delete Holes</i>	47
<i>Delete Internal Points</i>	48
<i>Delete Trees</i>	48
<i>Clear Mesh Model</i>	49
<i>Activate Regions</i>	49
<i>View Mesh Model</i>	49
CHAPTER 6: TABLES	50
<i>Curve Steps</i>	50
<i>Nodes</i>	50
<i>Curves</i>	50
<i>Plane Regions</i>	51
<i>Cylinder Regions</i>	53
<i>Sphere Regions</i>	54
<i>Rev Regions</i>	54
<i>General Patch Regions</i>	54
CHAPTER 7: GENERATE	56
<i>Generate Mesh</i>	56
<i>Generate Mesh From File</i>	56
CHAPTER 8: SETTINGS	57
<i>Units & Precisions</i>	57
<i>Data Options</i>	57
<i>New Origin</i>	58
<i>Vertical Axis</i>	58
<i>Graphic Scales</i>	59
<i>Colors</i>	59
<i>Preferences</i>	60
<i>Enable/Disable Hardware Acceleration</i>	61
<i>Toolbars Main Toolbar</i>	61
<i>Toolbars View Toolbar</i>	61
<i>Toolbars Status Bar</i>	61
<i>Toolbars Command Bar</i>	61
<i>Tools Copy Command History</i>	61
<i>Tools Clear Command History</i>	61
<i>New Window</i>	62
<i>Close</i>	62
<i>Close All</i>	62
<i>Tile Horizontal</i>	62
<i>Tile Vertical</i>	62
<i>Tile Cascade</i>	62

TUTORIALS 63

Tutorial 1 – Plane Region..... 65

Tutorial 2 – Plane Region with a Hole 73

Tutorial 3 – Cylinder Region (Scodelis-Lo Roof)..... 79

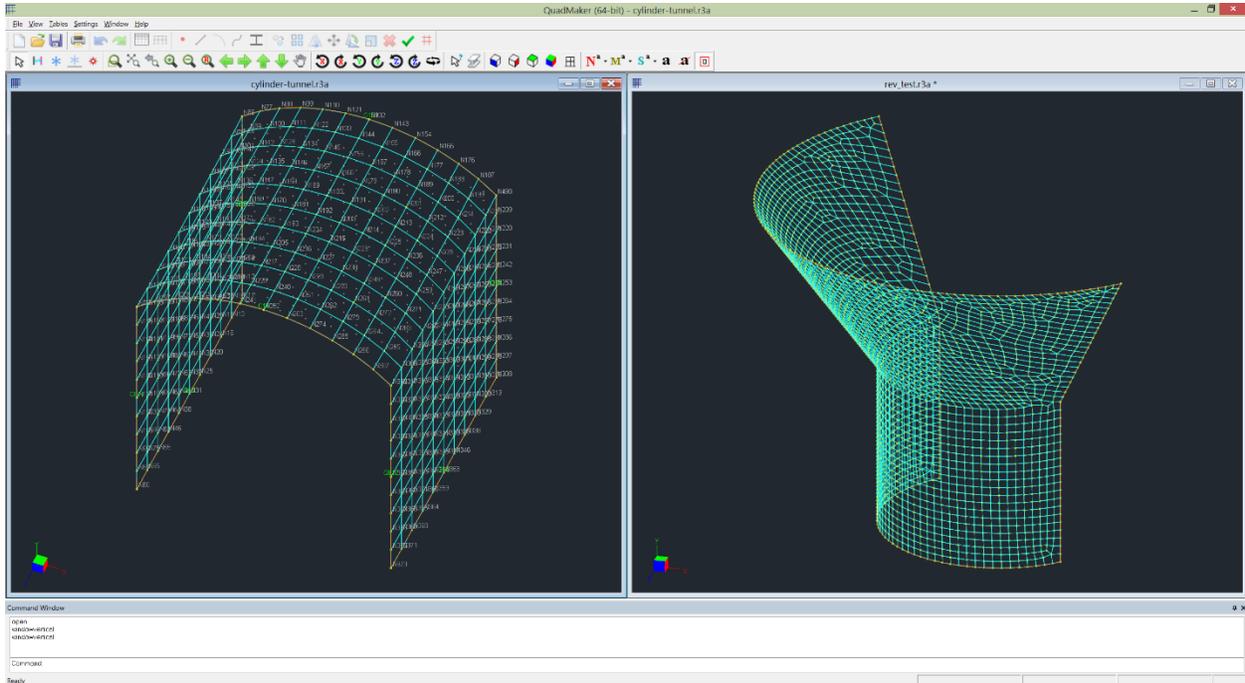
Tutorial 6 – General Patch Region..... 91

Tutorial 7 – Floor with Grid of Beams 94

Introduction

Graphical User Interface (GUI)

QuadMaker has a modern graphical user interface. It includes menus, multiple toolbars, command window, multiple views and a status bar.



Command Window

A command can be entered by using the keyboard. This is especially useful (and efficient) when drawing geometry such as nodes, lines, arc, and splines. For example, you can use “line” command to start draw lines, or “open” command to open a new file. To enter a command using the keyboard, type the command name (e.g.: “line” or “open”) on the command line window and press Enter or Spacebar key. For example, in the “line” command, type in “line” in the command window, then enter the next nodal coordinates directly and press Enter key.



To repeat the last command, press Enter or Spacebar at command prompt. You can also invoke the last command by right-clicking your mouse and select Repeat Last.

To cancel a command in progress, press ESC.

By default, the command window is docked. You can undock the command window by dragging it away. You can then resize its width and height. To dock the command window again by double clicking on its caption.

The command window can be turned on or off by pressing F3.

Enter Nodal Coordinates

Nodal coordinates can be specified by either Cartesian or Polar systems through the command window.

Cartesian coordinates can be specified by either absolute or relative X, Y and/or Z values, separated by comma(s). Z value may be omitted, in which case the program will assign $Z = 0$.

By default, the values entered are absolute coordinates, which are based on the origin (0, 0, 0) of the global coordinate system. Relative coordinates are based on the last nodal coordinates entered. To specify relative coordinates, precede the coordinate values with a @ character.

For example, the following will draw three nodes (1, 1, 0), (3, 4, 2) and (2, 8, 3)

Command: node

Specify nodal coordinates: 1,1

Specify nodal coordinates: @2,3,2

Specify nodal coordinates: @-1,4,1

Polar coordinates can be specified by either absolute or relative distance plus an angle, separated by an < character. The angle is degrees measured from the +X direction and is positive in counterclockwise direction around +Z axis.

By default, the value entered is absolute distance, which is based on the origin (0, 0, 0) of the global coordinate system. Relative distance is based on the last nodal coordinates entered. To specify relative distance, precede the distance values with an @ character.

For example, the following will draw three nodes (1, 1, 0), (4.33013, 2.5, 0) and (6.45145, 4.62132, 0)

Command: node

Specify nodal coordinates: 1,1

Specify nodal coordinates: 5<30

Specify nodal coordinates: @3<45

Mouse Use

You can zoom in or out the model view by rolling middle mouse wheel.

You can pan the model view by holding down and move the middle mouse button.

Spreadsheet Navigation

The program uses spreadsheets (grid control, see Figure 0.2) extensively for data input and output. It offers multiple ways to navigate within a spreadsheet as specified in the following table.

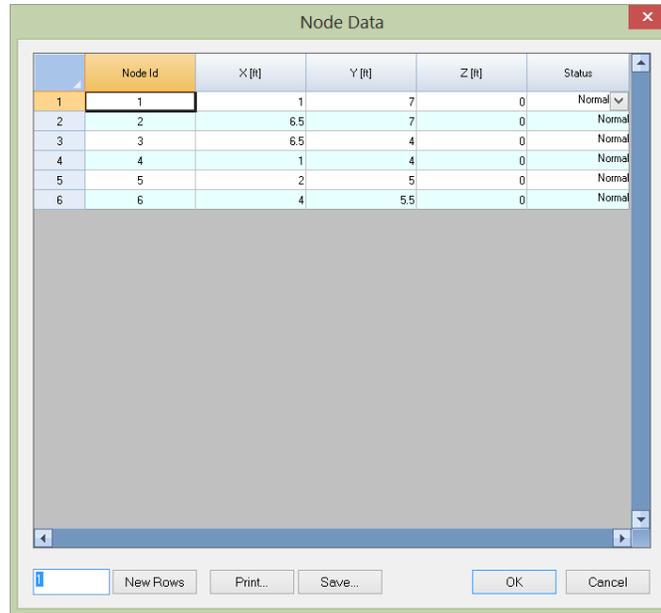


Figure 0.2

Key	Action
up arrow	Moves active cell up one row
down arrow	Moves active cell down one row
right arrow	Moves active cell right one column
left arrow	Moves active cell left one column
Shift+arrow key	Extends selection in direction of arrow key
Page Up	Moves active cell one page up
Page Down	Moves active cell one page down
Ctrl+Page Up	Moves active cell one page left
Ctrl+Page Down	Moves active cell one page right
Home	Moves active cell to first cell in row
End	Moves active cell to last cell in row that contains data
Ctrl+Home	Moves active cell to first row, first column
Ctrl+End	Moves active cell to last row and column that contain data
Tab	Moves active cell to next cell to the right (or at end of row moves to beginning of next row)
Shift+Tab	Moves active cell to next cell to the left (or at beginning of row moves up to end of row above)
Shift+space	Selects current row
Ctrl+space	Selects current column
Shift+Ctrl+space	Selects entire sheet

Ctrl+X or Shift+Del	Cuts current selection or active cell's data to Clipboard
Ctrl+V or Shift+Ins	Pastes Clipboard contents into active cell
Ctrl+C or Ctrl+Ins	Copies current selection or active cell's data to Clipboard
Enter	Active cell moves down
Esc	If sheet is in edit mode, previous cell value replaces new value and edit mode is turned off
F2	If edit mode is on, cell value is cleared

System Requirements

Memory (RAM)	8 GB or more is recommended
Video Card	OpenGL® acceleration is desired
Hard Disk	100 GB free disk space
Operating System	x64-based Windows 7, 8, 10, and 11 ARM64-based Windows 11

Technical Background

QuadMaker is an interactive Windows finite element mesh generator that produces 100% quadrilateral elements (shell4) on planar and curved surfaces in 3D space. A mesh model must be defined prior to generating mesh elements. A mesh model consists of one or more mesh regions (aka sub-regions) of different geometrical shapes. The supported regions includes flat surface, cylindrical surface, spherical surface, surface of revolution (rev), and general patch surface. With these types of regions, arbitrarily complex geometrical shapes can be constructed.

Nodes

A **node** is a 3D point in space. It is identified by an number (node id).

Curves

A **curve** represents a boundary part of a surface. It is identified by an number (curve id). There are three types of curves:

1. QL-type curve which is a line segment defined by two nodes.
2. C-type curve which is an arc defined in 3D space by three consequent nodes not belonging to one and the same line.
3. Q-type curve which is a general splice consists of four or more nodes. The curve must neither self-intersect nor contain cusps;

An important property in each curve is “step size” (STEP), a user-defined distance between neighboring nodes that are to be generated along the curve during the preprocessing stage.

Edges

An edge is an non-intersecting sequence of QL-type, C-type, and/or Q-type curves. Edges are used in sphere, rev, and general patch regions. They are useful when parts of a surface boundary contains one or several cusps.

Regions (aka sub-regions)

A **region** (aka sub-region) is a closed, non-intersecting sequence of curves that defines a 3D surface.

Plane Region

A PLANE-type region is defined by a sequence of curves that belongs to one and the same flat surface. Different plane regions in a mesh model may lie on different planes. It is important to point out that the boundary curves may be specified in arbitrary order. Their actual order will be restored automatically during the preprocessing stage. This is a very convenient feature in QuadMaker.

A region can have one or more dependents such as holes, internal points, and internal lines (aka. trees).

An additional parameter in a region called REF (refinement coefficient) can be specified with a value between [0.0 and 0.35]. This parameter can, to some extent, regulate the density of mesh. Here is how the REF value works:

1. If STEP sizes for all curves comprising the boundary of a region are set to the same or close values, then all parts of the sub-region will be meshed more or less the same density. In such a case, the value of parameter REF is not important and can be set to 0.
2. If there are significant variations in the step sizes along the region, the mesh will have transitional areas between parts of fine mesh (originating near those curves that have smaller step sizes) and coarse mesh (originating near curves with large step sizes). In that case the user may specify how far the parts of the fine mesh can "penetrate" into the interior of the sub-region. As a general rule, the higher the value of REF, the more space will be occupied by fine mesh.

The following illustrates two regions (Figure 0.1) with REF value of 0 at the left and REF value of 0.35 at the right.

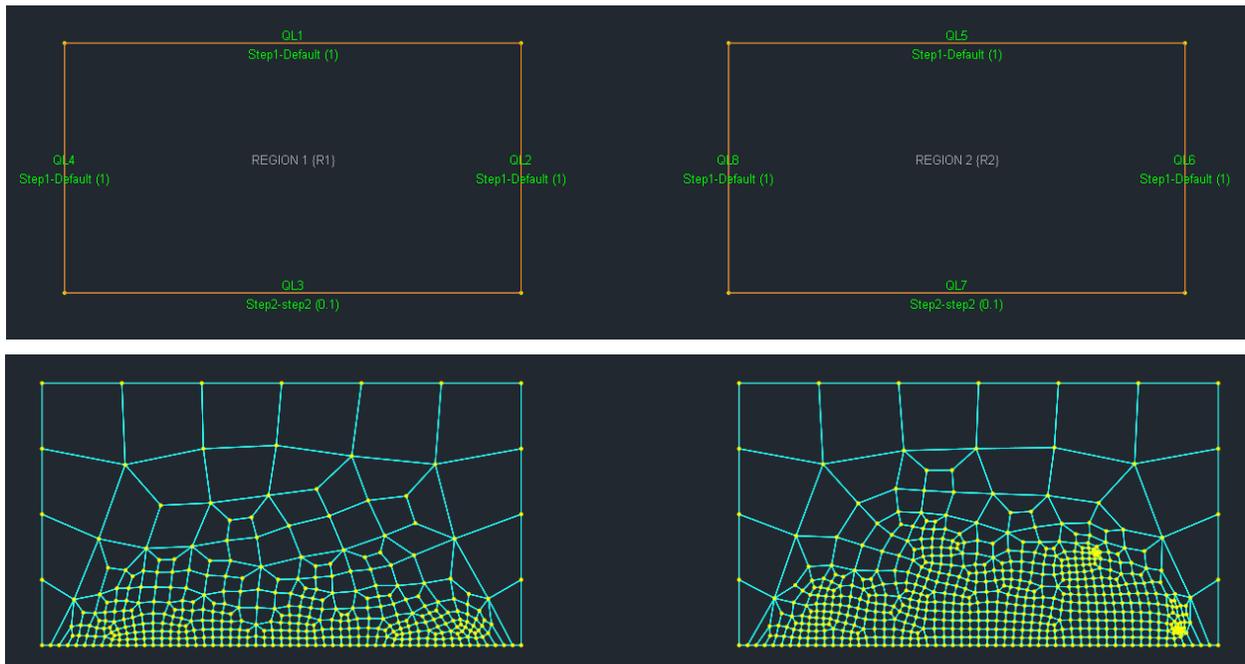


Figure 0.1

Holes

A **hole** in a plane region is defined by a closed, non-intersecting sequence of curves that belong to the same plane as the region. The boundary curves may appear in arbitrary order. Their actual order will be restored automatically during the preprocessing stage.

The following is an example (Figure 0.2) where a triangular hole exists inside a rectangular plane region.

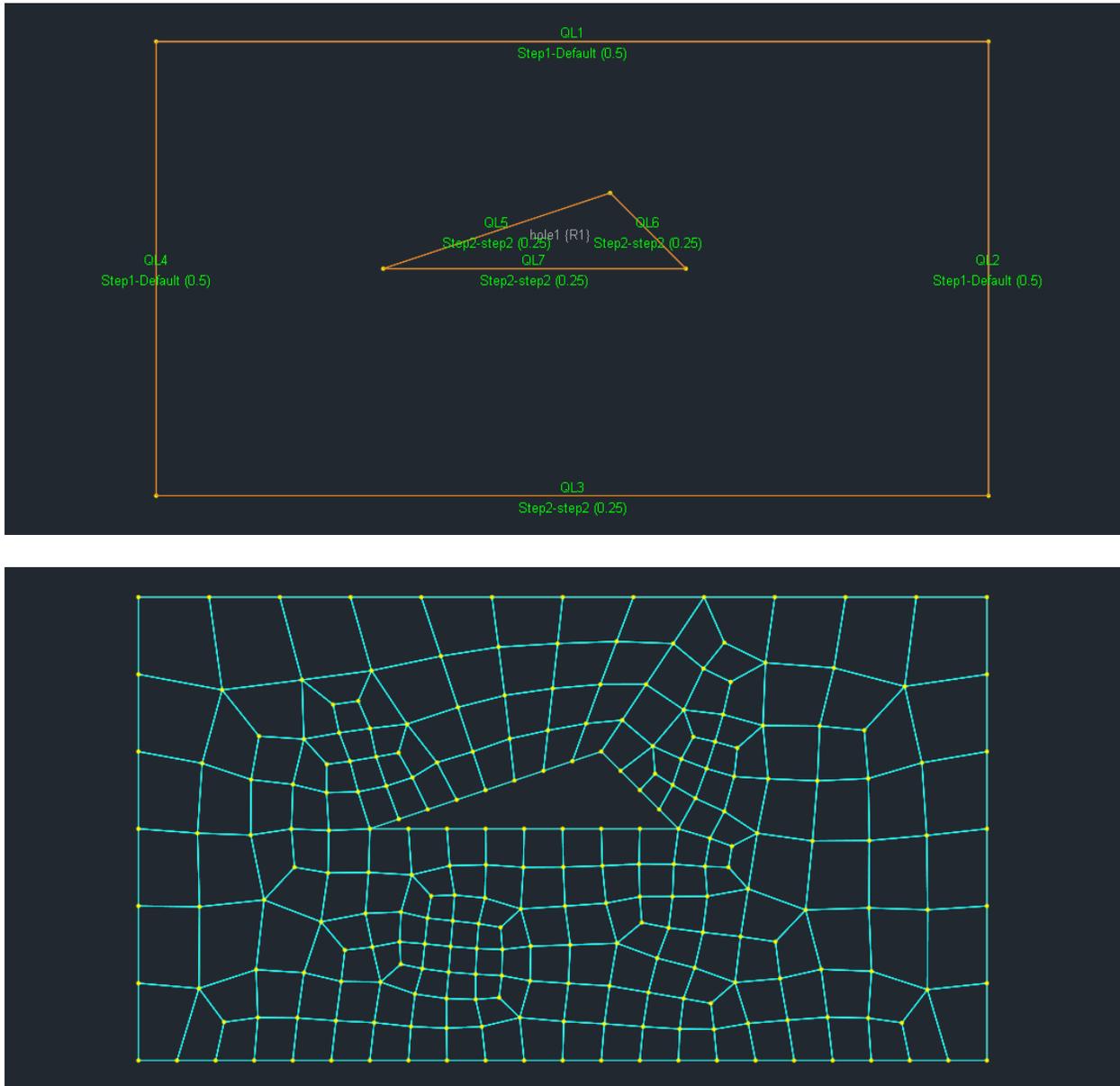


Figure 0.2

Internal Points

An **internal point** (PINT) defines a location that the mesh generation engine is to include a node in the specified location. It is well known that areas of large stress gradients often arise around locations of

boundary constraints and point loads. Accurate FEA modeling requires zones of very fine elements to be generated around these PINT-points. Two optional numerical, STEP and RAD, allow the user to set a surrounding refinement area:

- STEP: a desirable average edge length of the quadrilaterals around the point
- RAD: a desirable radius of the refinement area around the point.

In the following example (Figure 0.3), we have two internal points: one with STEP=0 and RAD=0, and another with STEP=0.1 and RAD=0.25.

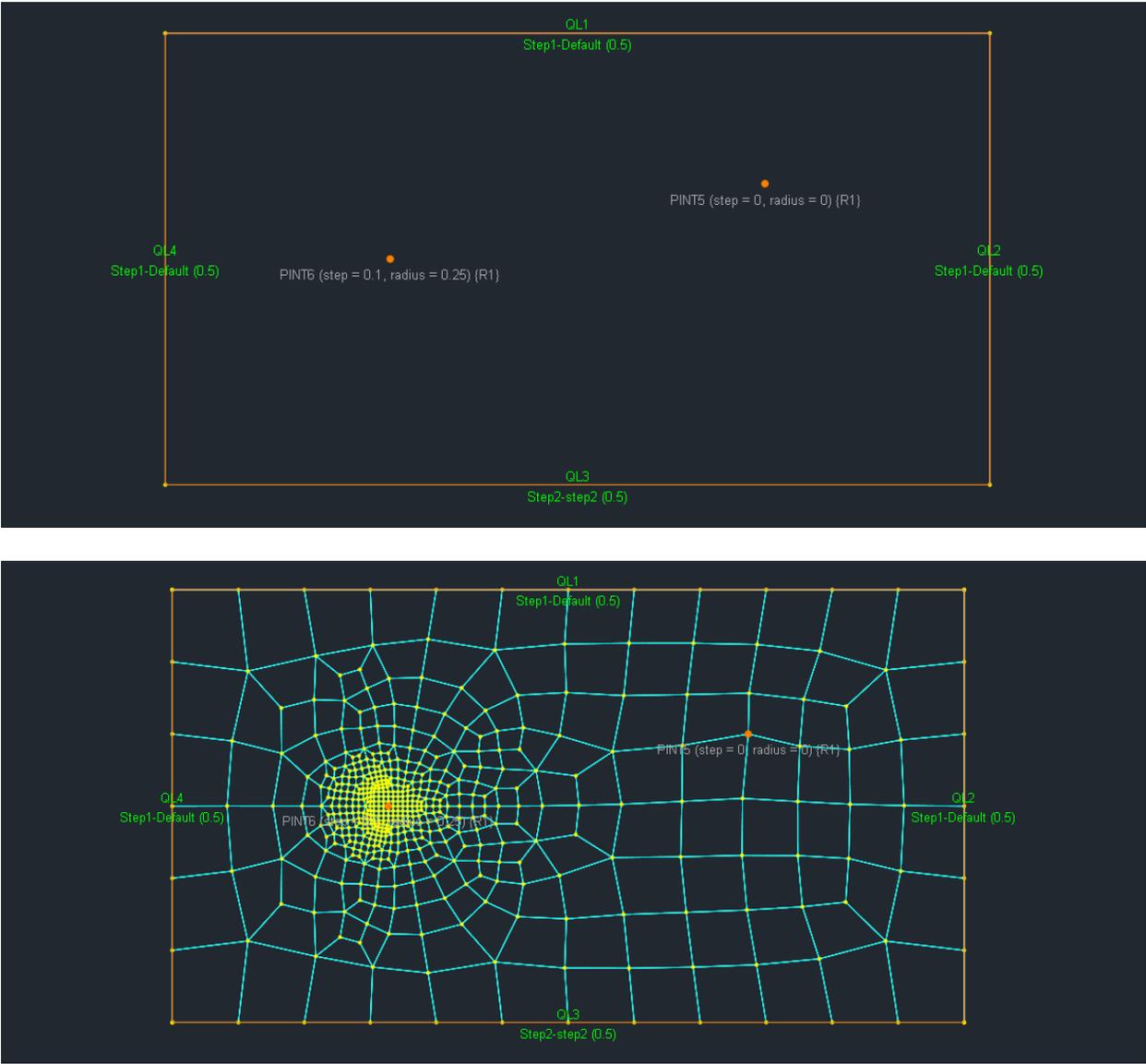


Figure 0.3

Trees

A tree (a combination of internal lines) in a region defines a sequence of curves that the mesh generation engine is to include nodes along these curves. Conceptually, a tree is like a physical tree

which starts from a root and then branch out. A tree may represent curves, walls, or other boundaries in a structural model. It is important to point out that the curves in a tree must be all connected (no isolated curves). Tree curves must not form internal (closed) contours.

In the following example (Figure 0.4), a tree is defined by a single arc originating from the lower left corner of the rectangular plane region.

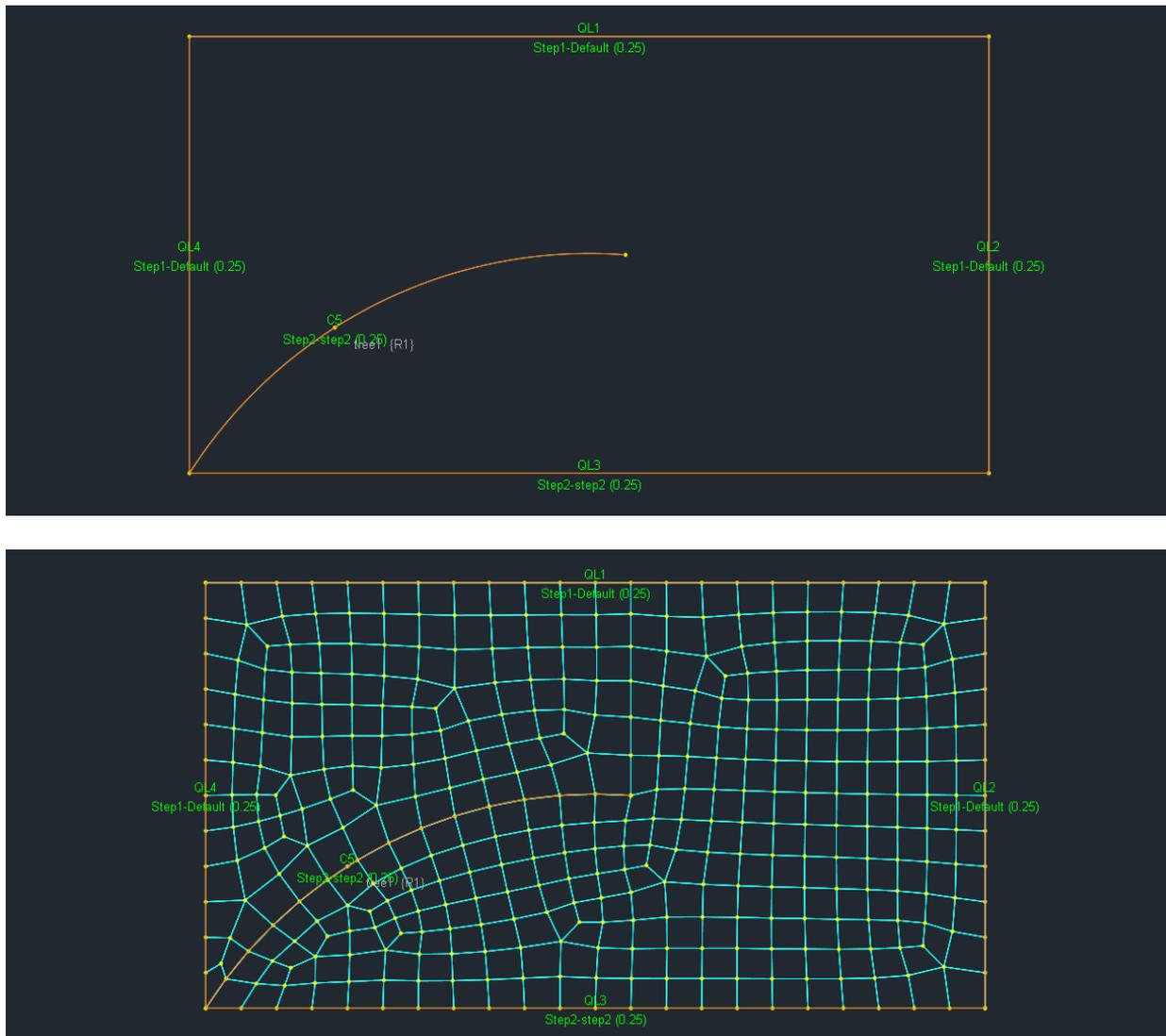


Figure 0.4

Cylinder Region

A CYLINDER-type region is defined by a closed, non-intersecting sequence of curves that belongs to one and the same cylindrical surface formed when a straight line in 3D space travels such that each new position of the line is parallel to its previous position. The cylindrical surface orientation is defined by the directional angles in degrees of the straight line: AngleX, AngleY, and AngleZ. An additional parameter in a region called REF (refinement coefficient) can be specified. For more information, please refer to PLANE region section earlier in this chapter. It is important to point out that the boundary curves may be specified in arbitrary order. Their actual order will be restored automatically during the preprocessing stage. This is a very convenient feature in QuadMaker.

As an example, the following (Figure 0.5) is a mesh model for cylindrical tunnel. All curves have the STEP size of 1 ft. The angleX=90 deg, angleY=90 deg, and angleZ=0 deg are the directional angles (in degrees) of the straight line that defines the cylindrical surface orientation. The mesh refinement coefficient REF is specified as 0.0.

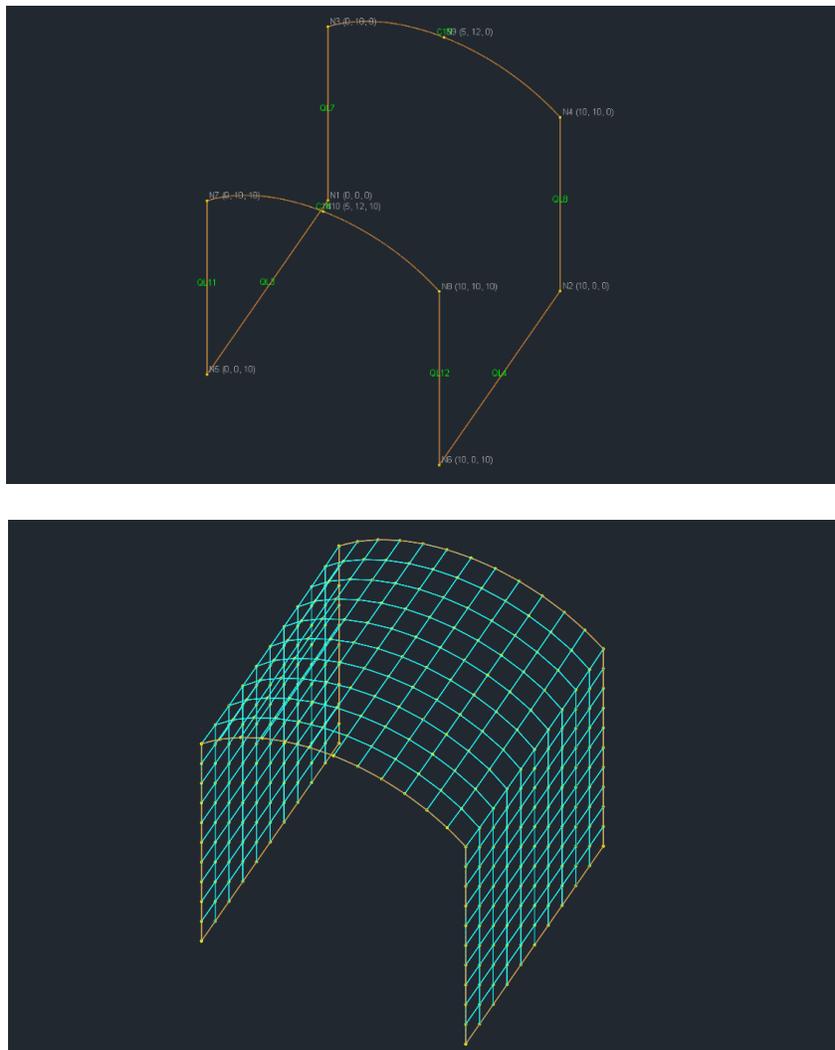


Figure 0.5

Sphere Region

A SPHERE-type region is defined by 2, 3 or 4 edges, all belonging to one and the same spherical surface. The curves in each edge must be arc (C-type) curves. The center coordinates of the sphere are specified as CenterX, CenterY, and CenterZ. An additional parameter in a region called REF (refinement coefficient) can be specified. For more information, please refer to PLANE region section earlier in this chapter. It is important to point out that the boundary edges as well as the curves in each edge may be specified in arbitrary order. Their actual order will be restored automatically during the preprocessing stage. This is a very convenient feature in QuadMaker.

As an example, the following (Figure 0.6) is a mesh model for a quarter of a sphere. The model is defined by two edges (with one arc curve in each edge), each curve has a STEP size of 0.1 ft. The center of the sphere is (0, 0, 0). The mesh refinement coefficient REF is specified as 0.0.

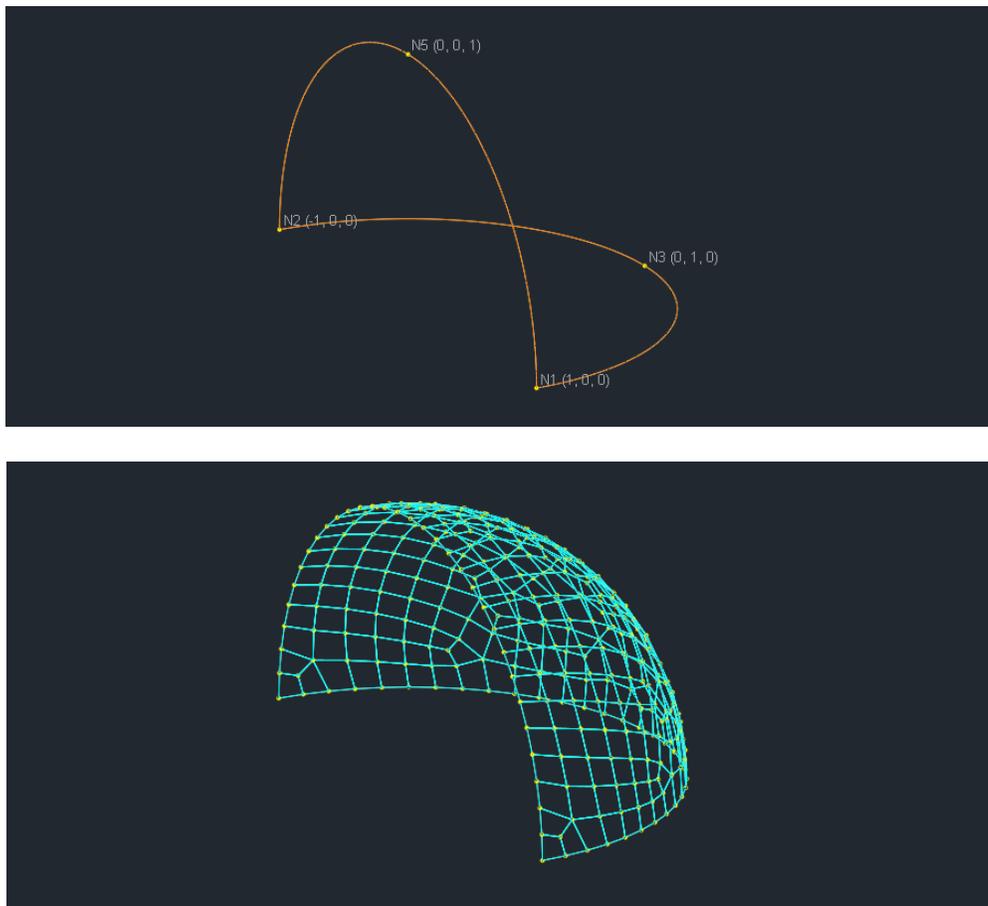


Figure 0.6

As another example, the following (Figure 0.7) is a mesh model for a small patch of a sphere. The model is defined by three edges (with one arc curve in each edge), each curve has a STEP size of 0.1 ft. The center of the sphere is (0, 0, 0). The mesh refinement coefficient REF is specified as 0.0.

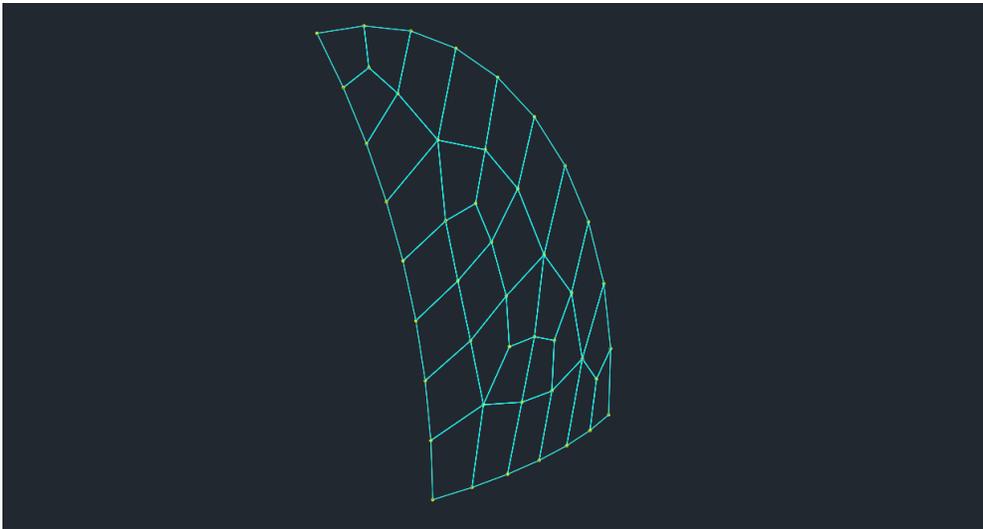
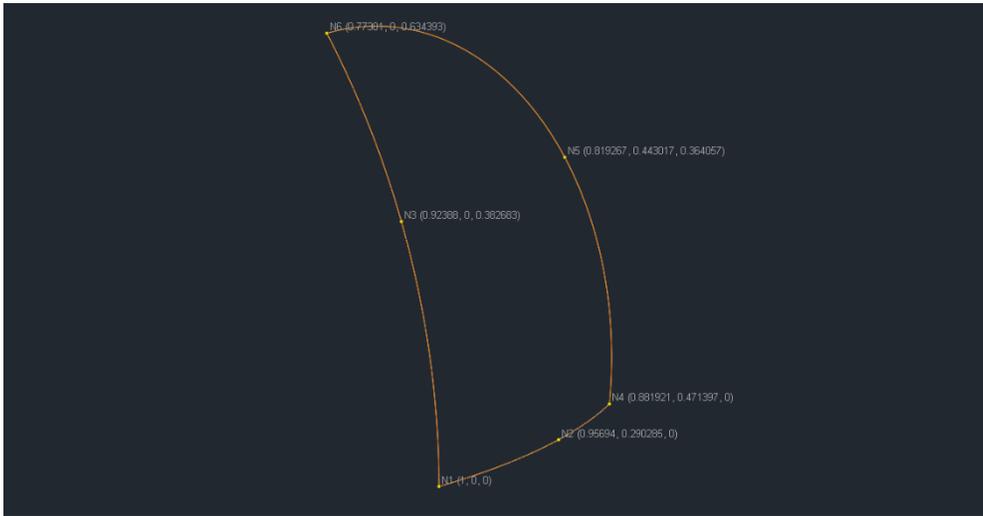


Figure 0.7

Rev Region

A REV-type region is defined by exactly 4 edges, two of which must be C-type curves representing the "upper" and the "lower" arcs of the surface of revolution correspondingly. The upper and lower arcs belong to parallel planes that are orthogonal to the axis of revolution. Two other edges, describing the "left" and the "right" sides of the patch, define the actual shape of the surface of revolution. They may consist of unequal numbers of curves. The corresponding start and end points of the curves must belong to one and the same plane orthogonal to the axis of revolution. An additional parameter in a region called REF (refinement coefficient) can be specified. For more information, please refer to PLANE region section earlier in this chapter. It is important to point out that the boundary edges as well as the curves in each edge may be specified in arbitrary order. Their actual order will be restored automatically during the preprocessing stage. This is a very convenient feature in QuadMaker.

As an example, the following (Figure 0.8) is a mesh model for a half of a cone. The model is defined by four edges: upper and lower arcs, left and right sides. Each of the curves has a STEP size of 0.1 ft. The mesh refinement coefficient REF is specified as 0.0.

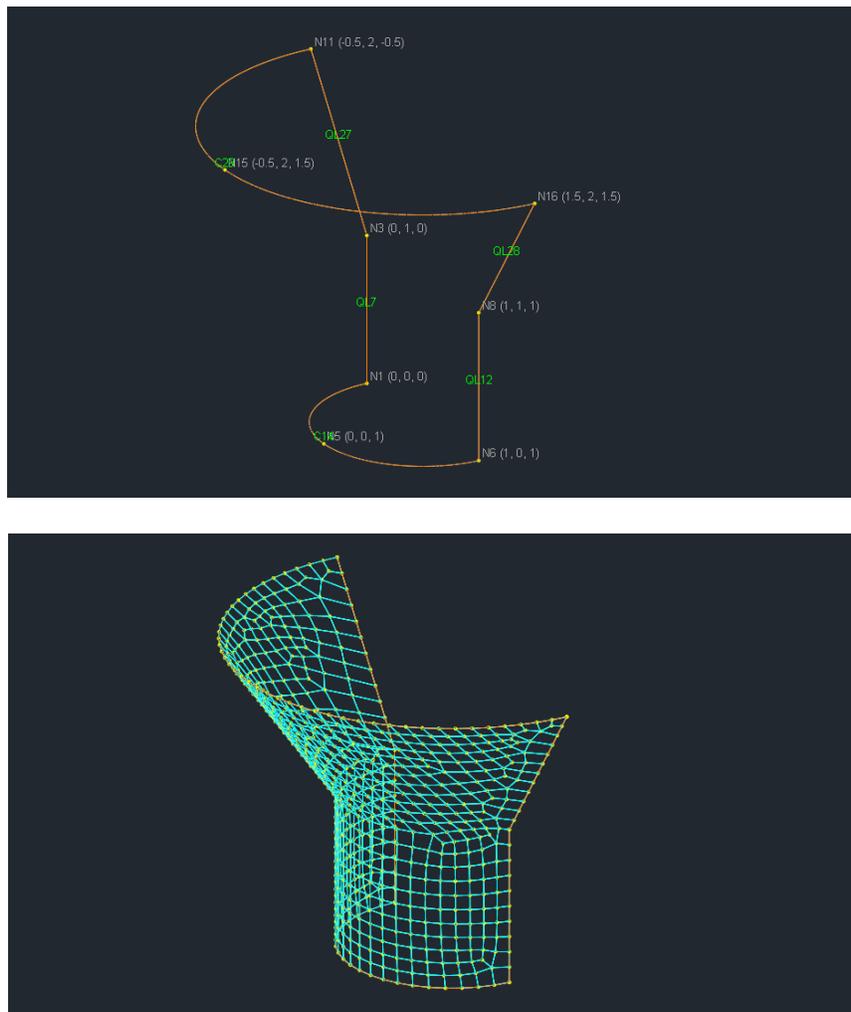


Figure 0.8

General Patch Region

A GENERAL PATCH-type region is defined by 2, 3, or 4 edges. One edge may consist of one or more curves. The latter option is useful in cases when the user wants an edge to be divided to several parts with different step sizes; or when a particular edge contains one or several cusps. An additional parameter in a region called REF (refinement coefficient) can be specified. For more information, please refer to PLANE region section earlier in this chapter. It is important to point out that the boundary edges as well as the curves in each edge may be specified in arbitrary order. Their actual order will be restored automatically during the preprocessing stage. This is a very convenient feature in QuadMaker.

General-type patches have to be used in situations where no other information about the patch (i.e. actual type of surface) is available except of its boundary curves.

As an example, the following (Figure 0.9) is a mesh general model. The model consists of four arc edges. Each of the curves has a STEP size of 0.25 ft. The mesh refinement coefficient REF is specified as 0.0.

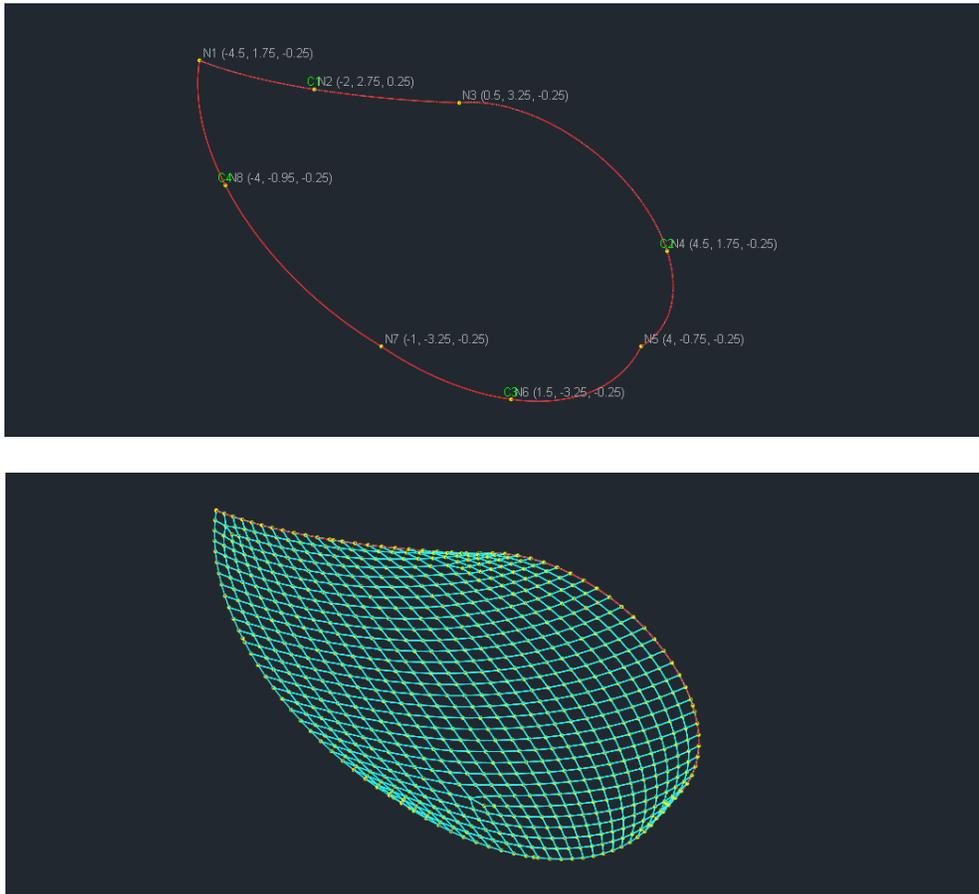


Figure 0.9

Menus

Chapter 1: File Menu

The File menu provides commands that are related to files. It also provides commands related to text and graphical reports. Input data for a mesh model is saved in one single text file (also called a model file in this program) and has the default extension of “qdm”. You may however save or open model files with different extensions. The program provides Multiple Document Interface (MDI), that is, you may open and work on multiple files at the same time. The default result file is a text file with the extension “r3a” which can be directly imported to Real3D finite element analysis program. You can then export the mesh result in a simpler text file that contains only node and element data. In addition, a text log file is created during the solution process of each model. It has the same name as the input file but with the extension of “log”.

New

File | New creates a new QuadMaker model file.

Open

File | Open opens an existing QuadMaker model file (qdm) or result file (r3a). You may open one or multiple files at one time. To open multiple files, simply go to the file list in Open dialog box (Figure 1.1) and click the files while pressing CTRL or SHIFT.

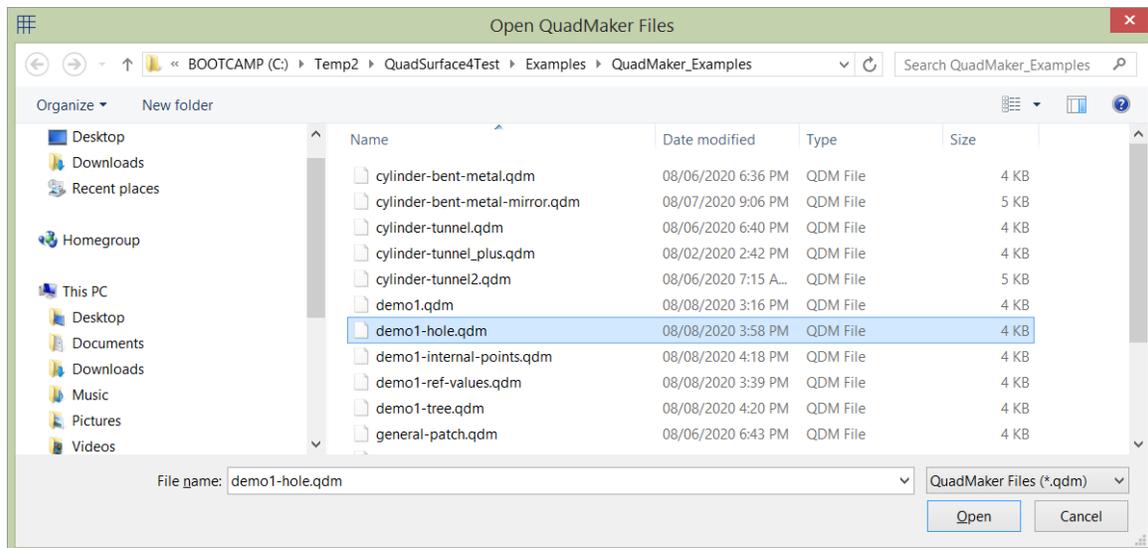


Figure 1.1

Multiple Document Interface (MDI) makes it easy for you to share data and compare results between different models. Multiple windows or views may be associated with a single file. These windows may have different display settings with respect to zooming, panning, displaying or hiding various entities, annotations, etc.

Close

File | Close closes the active QuadMaker model file (qdm) or result file (r3a). You will be prompted to save the document if changes have been made since it was last saved. You may close all opened documents without exiting the program.

Save

File | Save saves the model file (qdm) or result file (r3a). If the model has not been saved before, the Save As dialog box will be displayed prompting you to enter a file name. When the current window is a report view, File | Save saves the report file.

Save As

File | Save As saves the model or result file under a different name.

Save All

File | Save All will save all the documents currently opened. If a model has not been saved before, the Save As dialog box will be displayed prompting you to enter a file name.

Export Nodes & Shells

File | Export Nodes & Shells exports the nodes and shells data from the result file. The following is the file format:

```
Nodes: {Number of Nodes}
{Node Id} {Nodal X coordinate} {Nodal Y coordinate} {Nodal Z coordinate}
{Node Id} {Nodal X coordinate} {Nodal Y coordinate} {Nodal Z coordinate}
...
Shells: {Number of Shells}
{Shell Id} {Shell Node 1} {Shell Node 2} {Shell Node 3} {Shell Node 4}
{Shell Id} {Shell Node 1} {Shell Node 2} {Shell Node 3} {Shell Node 4}
...
```

Here is an actual example of the exported file:

```
Nodes:235
1          6.5          7          0
2          6.041667354  7          0
3          6.5          6.4999995  0
...
Shells:202
1          6          13          22          12
2          13         23          34          22
3          23         35          48          34
4          49         63          48          35
...
```

Print Setup

File | Print Setup allows you to set up the page layout (Figure 1.2) before printing current view.

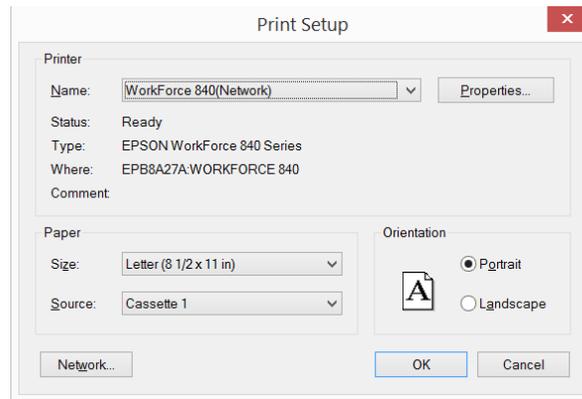


Figure 1.2

Print Current View

File | Print Current View allows you to preview and print the current view.

General Information

File | General Information prompts you with the following dialog box (Figure 1.3). You may enter a model name, your company name, your name and a description about the model (up to 1020 characters long).

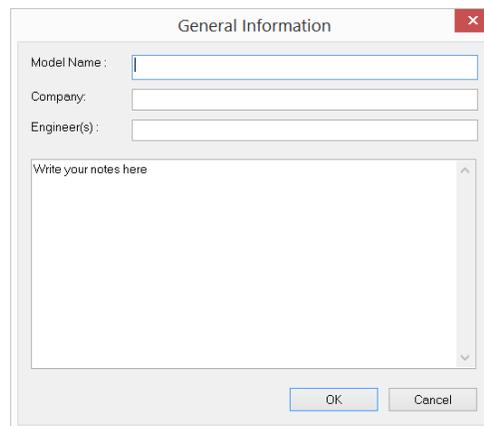


Figure 1.3

View Log File

File | View Log File allows you to view the log file generated during the solution process

Open Containing Folder

File | Open Containing Folder allows you to open the folder that contains the current model file.

Chapter 2: Edit

The Edit main menu provides commands to edit or modify the model.

Undo

Edit | Undo undoes the previous structural command that affects the model geometry (nodes and curves). By default, you may undo up to 200 levels. You may set a different number of undo levels by running Settings | Data Options. Non-structural commands such as zooming or panning may not be undone. More undo levels requires more computer memory.

Redo

Edit | Redo reverses the previous undo command.

Duplicate

Edit | Duplicate prompts you with the following dialog box (Figure 2.1). It allows you make one copy of the selected nodes and curves of a model to a different location. You may specify copy distances along the X, Y or Z directions. You have the option to automatically merge nodes after copying. You should check this option unless duplicate nodes are explicitly permitted.

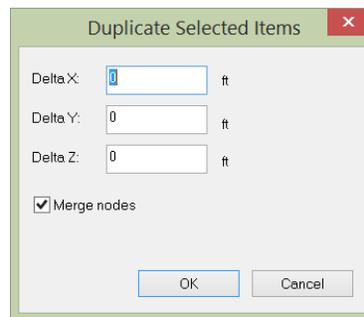


Figure 2.1

Array

Edit | Array prompts you with the following dialog box (Figure 2.2). It allows you make one or more copy of the selected nodes and curves of a model to a different location. You may specify the step size along the X, Y or Z directions. You have the option to automatically merge nodes after copying. You should check this option unless duplicate nodes are explicitly permitted.

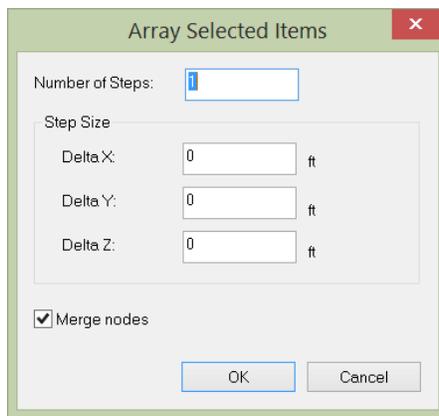


Figure 2.2

Mirror

Edit | Mirror prompts you with the following dialog box (Figure 2.3). It allows you make one mirror copy of the selected nodes and curves of a model about XY, YZ or XZ plane. You have the option to automatically merge nodes after mirroring. You should check this option unless duplicate nodes are explicitly permitted.

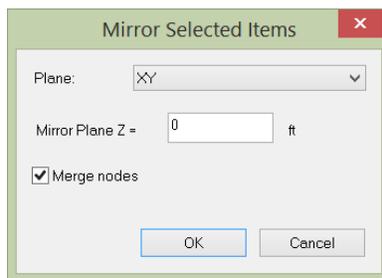


Figure 2.3

Move

Edit | Move prompts you with the above dialog box (Figure 2.4). It lets you move selected nodes and curves of a model to a different location. You may specify move distances along the X, Y or Z directions. You have the option to automatically merge nodes after moving. You should check this option unless duplicate nodes are explicitly permitted.

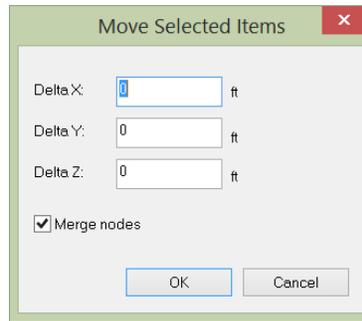


Figure 2.4

Rotate

Edit | Rotate prompts you with the following dialog box (Figure 2.5). It lets you rotate selected nodes and curves of a model by an angle about one of the global axes. You have the option to automatically merge nodes and elements after rotating. You should check this option unless duplicate nodes are explicitly permitted.

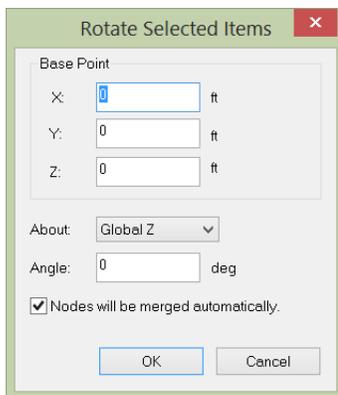


Figure 2.5

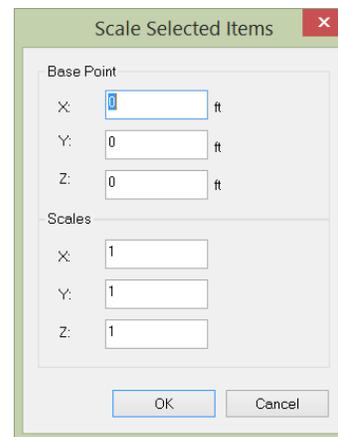


Figure 2.6

Scale

Edit | Scale prompts you with the above dialog box (Figure 2.6). It lets you scale selected nodes and curves of a model in the X, Y or Z directions. You may specify the coordinates of a base point and scales for the three global directions.

The following formula is used to perform the scaling in the program.

$$X_{new} = X_{base} + (X_{old} - X_{base}) * scale$$

Where X_{new} represents the nodal coordinates after scaling, X_{old} represents the nodal coordinates before scaling and X_{base} represents coordinates of the base point.

Delete

Edit | Delete prompts you with the following dialog box (Figure 2.7). It allows you to delete selected nodes or/and curves. Dependents such as regions, holes, internal points, trees will be deleted if their parent nodes or curves are deleted.

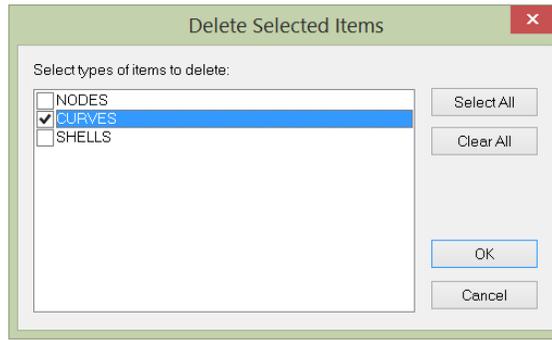


Figure 2.7

Extrude | Extrude Nodes to Curves

Edit | Extrude | Extrude Nodes to Curves prompts you with the following dialog box (Figure 2.8). It generates a series of curves by extruding selected nodes along a global direction.

You may specify a distance list and an extrusion direction for the generation of curves. The distance list is a comma separated list that specifies multiple distances. For example, a distance list of “12,2@14,3@10” will generate distances of 12, 14, 14, 10, 10 and 10 in length units. You have the option to automatically merge nodes after extrusion. You should check this option unless duplicate nodes are explicitly permitted. The following curves in Figure 2.9 are generated by extruding one node (the first node) using the input from Figure 2.8.

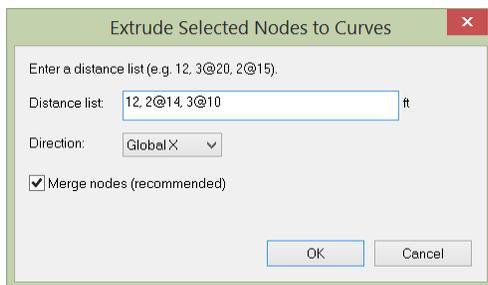


Figure 2.8

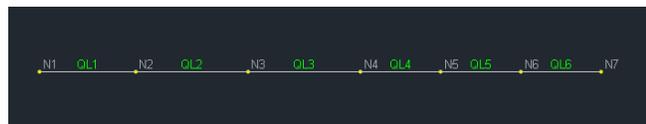


Figure 2.9

Merge All Nodes

Edit | Merge All Nodes merges all nodes that are located within a distance tolerance between two or more nodes. You may set the distance tolerance using the command Settings | Data Options.

Remove All Orphaned Nodes

Edit | Remove All Orphaned Nodes removes all nodes that are not connected to any curves.

Remove Unused Curves

Edit | Remove Unused Curves removes all curves that are not used in any of the regions (including holes and trees).

Switch Coordinates

Edit | Switch Coordinates prompts you with the following dialog box (Figure 2.10). It allows you to switch the X and Z, or the Y and Z coordinates of the selected nodes. For example, you may generate a floor system on the XY (vertical) plane and then switch coordinates to place the floor on the XZ (horizontal) plane.

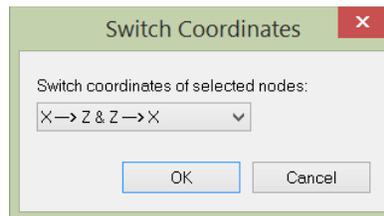


Figure 2.10

Clear | Clear Undo & Redo

Edit | Clear | Clear Undo & Redo clears the undo/redo buffer, thus frees up computer memory.

Clear | Clear Everything

Edit | Clear | Clear Everything clears all input from computer memory. You should think twice before running this command.

Chapter 3: View

The View menu provides commands to graphically view inputs such as geometry; perform selections by various methods.

Redraw

View | Redraw regenerates and redraws all graphics in the model view.

Restore Model

View | Restore restores original settings for the model view. These settings include zooming = 1.0, panning = 0, rotations = 0s.

Preset Views

You may place the model view in a preset orientation by selecting one of these commands (Figure 3.1).

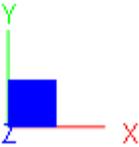
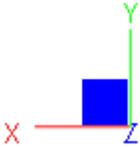
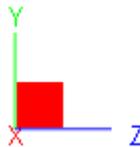
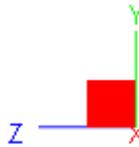
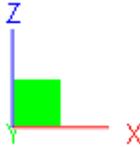
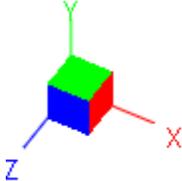
Front	Back	Left	Right
			
Top	Bottom	Isometric	
			

Figure 3.1

Named Views

View | Named Views allows you to save the current view settings such as zooming factor, panning distance or rotation angles so that you may recall this view later on.

Named Selections

View | Named Selections prompts you with the following dialog box (Figure 3.2). It allows you to save the currently selected nodes and curves to a name. You may use the command View | Select by Properties | Selection Names to recall the previously saved named selections. This command is very useful to group related items.

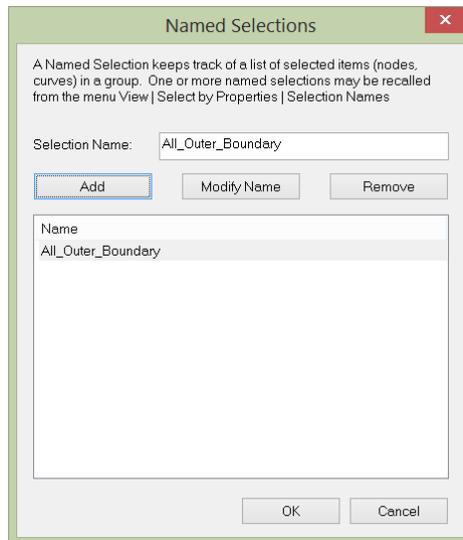


Figure 3.2

Zoom | Zoom Extent

View | Zoom | Zoom Extent displays the entire model in the view.

Zoom | Zoom Window

View | Zoom | Zoom Window zooms in on a specific part of the model by clicking and dragging the left mouse button. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Zoom | Zoom Object

View | Zoom | Zoom Object zooms in on a specific node or curve. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Zoom | Zoom Previous

View | Zoom | Zoom Previous lets you zoom back to the previous view. You may use this command after you zoom in or pan to view a portion of your model in greater detail.

Zoom | Zoom In

View | Zoom | Zoom In zooms in on the model by a preset factor (1.25). You may run this command by pressing CTRL+UP arrow or CTRL+RIGHT arrow.

Zoom | Zoom Out

View | Zoom | Zoom Out zooms out on the model by a preset factor (1.25). You may run this command by pressing CTRL+DOWN arrow or CTRL+LEFT.

Pan | Pan Screen

View | Pan | Pan Screen pans (moves) the model by clicking and dragging the left mouse button. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Pan | Left

View | Pan | Left pans the model to the left by a preset screen distance. You may run this command by pressing CTRL+LEFT arrow.

Pan | Right

View | Pan | Right pans the model to the right by a preset screen distance. You may run this command by pressing CTRL+RIGHT arrow.

Pan | Up

View | Pan | Up pans the model to the top by a preset screen distance. You may run this command by pressing CTRL+UP arrow.

Pan | Down

View | Pan | Down pans the model to the bottom by a preset screen distance. You may run this command by pressing CTRL+DOWN arrow.

Rotate | +X

View | Rotate | +X rotates the model view about X by a preset positive angle (5 degrees). You may run this command by pressing SHIFT+DOWN arrow.

Rotate | -X

View | Rotate | -X rotates the model view about X by a preset negative angle (5 degrees). You may run this command by pressing SHIFT+UP arrow.

Rotate | +Y

View | Rotate | +Y rotates the model view about Y by a preset positive angle (5 degrees). You may run this command by pressing SHIFT+RIGHT arrow.

Rotate | -Y

View | Rotate | -Y rotates the model view about Y by a preset negative angle (5 degrees). You may run this command by pressing SHIFT+LEFT arrow.

Rotate | +Z

View | Rotate | +Z rotates the model view about Z by a preset positive angle (5 degrees). You may run this command by pressing CTRL+SHIFT+UP arrow or CTRL+SHIFT+RIGHT arrow

Rotate | -Z

View | Rotate | -Z rotates the model view about Z by a preset negative angle (5 degrees). You may run this command by pressing CTRL+SHIFT+DOWN arrow or CTRL+SHIFT+LEFT arrow

Real Time Motion | Real-Time Pan

View | Real Time Motion | Real-Time Pan allows you to pan the model view in real time by clicking and dragging the left mouse button. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Please note that you can achieve real-time pan at any time by pressing and dragging the middle mouse button.

Real Time Motion | Real-Time Zoom

View | Real Time Motion | Real-Time Zoom allows you to zoom in or out on the model view in real time by clicking and dragging the left mouse button up or down. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Please note that you can achieve real-time zoom at any time by wheeling the middle mouse button.

Real Time Motion | Real-Time Rotate

View | Real Time Motion | Real-Time Rotate allows you to rotate the model view in real time by clicking and dragging the left mouse button. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Window/Point Select

View | Window/Point Select allows you to window-select or point-select nodes and elements by clicking or clicking-dragging the left mouse button. The point-selection selects or unselects at most one node or curve. The window-selection selects or unselects a group of nodes or curves within a rectangular selection window. If window-selection starts from the right, elements that are crossed by the selection rectangle will also be selected or unselected. By default, the selection mode is REVERSE SELECT, that is, entities picked will be selected if they are currently unselected and will be unselected if they are currently selected. However, the selection mode will be SELECT if CTRL is pressed while selecting, that is, entities picked will always be selected. The Window/ Point Select command is the default command in the program.

The selection of nodes and curves is an important activity in the program. Most of the commands apply only to selected nodes or curves. For this reason, many selection methods are provided in the program and are explained in the following sections.

Line Select

View | Line Select allows you to line-select elements by clicking-dragging the left mouse button. The line-selection selects or unselects a group of curves that intersect with the line.

Select by IDs | Nodes

View | Select by IDs | Nodes prompts you with the following dialog box (Figure 3.3). It allows you to select nodes by specifying a range of node IDs.

Three selection modes are provided: “Select”, “Unselect”, “Reverse Select”. The “Select” mode will select nodes. The “Unselect” mode will unselect nodes. The “Reverse Select” mode will select the unselected nodes and unselect the selected nodes.

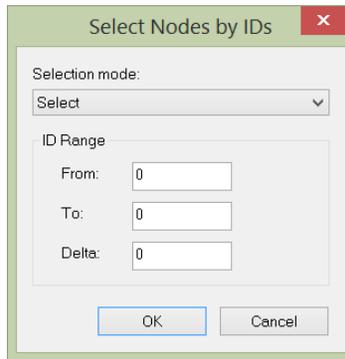


Figure 3.3

Select by IDs | Curves

Select by IDs | Shell4s

View | Select by IDs | Curves, Shell4s prompt you with dialog boxes that are almost identical to the dialog box used in View | Select by IDs | Nodes. They allow you to select curves or shells by specifying a range of IDs. Due to the similarities to the previous section, they are not repeated here.

Select by IDs | Select All

View | Select by IDs | Select All selects all nodes and elements. You may run this command by pressing CTRL+A.

Select by IDs | Unselect All

View | Select by IDs | Unselect All unselects all nodes and elements. You may run this command by pressing ESC. If you are in the middle of another command such as zooming, press ESC twice to unselect all.

Select by Properties | Curve Steps

View | Select by Properties | Curve Steps prompts you with the following dialog box (Figure 3.4). It allows you to select/unselect curves that use the specified steps.

Three selection modes are provided: “Select”, “Unselect”, “Reverse Select”. The “Select” mode will select curves. The “Unselect” mode will unselect curves. The “Reverse Select” mode will select the unselected curves and unselect the selected curves.

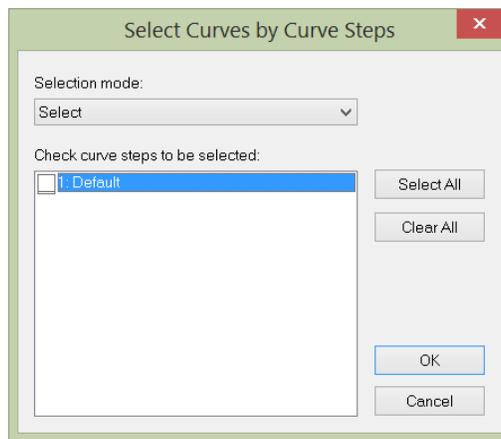


Figure 3.4

Select by Properties | Orphaned Nodes

View | Select by Properties | Orphaned Nodes selects all orphaned nodes.

Select by Properties | Coordinates

View | Select by Properties | Coordinates prompts you with the above dialog box (Figure 3.5). It allows you to select/unselect nodes and curves based on nodal coordinates. Nodes are selected/unselected if their coordinates are within the boundary of the minimum and maximum coordinates. Curves are selected/unselected if coordinates of their nodes are within the boundary of minimum and maximum coordinates. The selection modes are similar to the ones used in previous sections and are not repeated here.

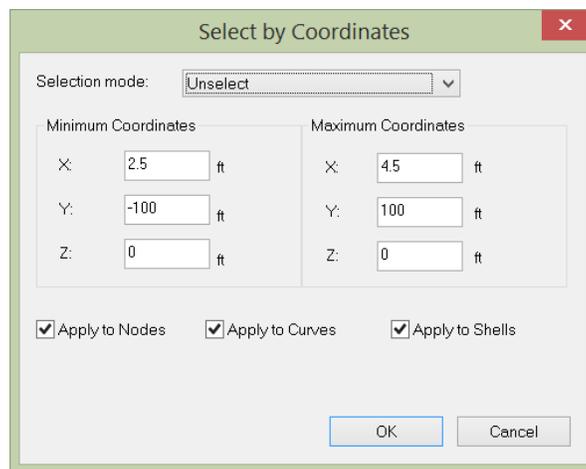


Figure 3.5

Select by Properties | Selection Names

View | Select by Properties | Selection Names allows you to select/unselect nodes and curves based on saved named selections (Figure 3.6). You may assign (or save) named selections from the Geometry menu.

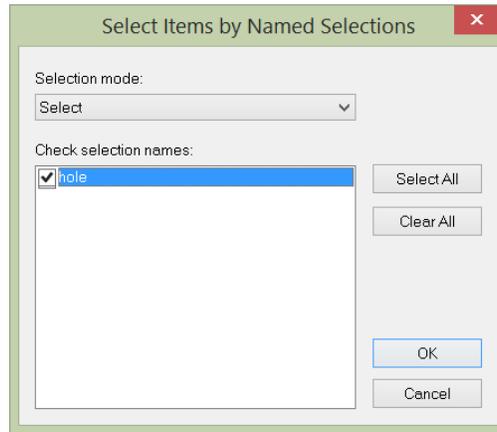


Figure 3.6

Select by Properties | Select All

Select by Properties | Unselect All

View | Select by Properties | Select All, Unselect All are the same as in View | Select by IDs. They are provided here for convenience.

Flip Selection

View | Flip Selection flips the selection status of all nodes and curves in the entire model. The nodes and curves currently selected will be unselected and vice versa.

Freeze Selected

View | Freeze | Freeze Selected freezes or hides the selected nodes and curves. The frozen nodes or curves are not displayed and are not modifiable unless the model integrity is at stake. This command allows you to focus on some particular parts of the model. For example, if you want to work on a particular floor of a three dimensional building, you may select the floor, flip the selection and freeze the selected entities.

Freeze All Except Selected

View | Freeze | Freeze All Except Selected freezes or hides all nodes and elements except the selected nodes and curves.

Thaw

View | Thaw allows you to thaw all frozen (hidden) nodes, elements and their dependents.

Mesh Model

View | Mesh Model shows or hide mesh model.

Annotate

View | Annotate prompts you with the above dialog box (Figure 3.7). It allows you to view annotations for nodes, elements, and mesh entities. Three annotation modes are available. You may annotate all entities, annotate selected entities, and erase existing annotations.

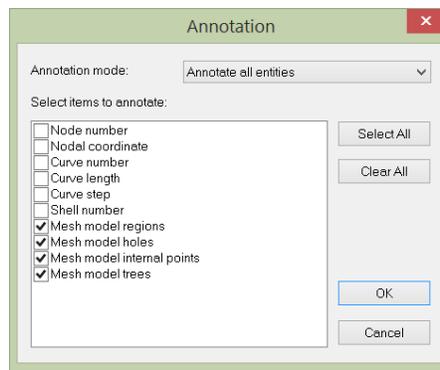


Figure 3.7

Clear Annotations

View | Clear Annotations clear all annotations.

Query

View | Query lets you query extensive input and output information for a single node, curves or finite element (Figure 3.8). For example, a node query will list node id, nodal coordinates.

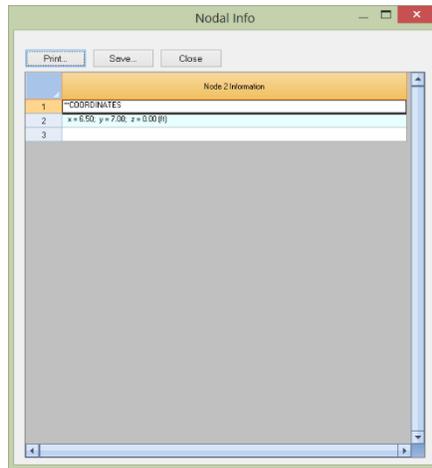


Figure 3.8

Distance

View | Distance allows you to graphically pick two nodes and calculate the their distance. It also lists the angle in degrees in XY plane and the angle in degrees from XY plane. The result is displayed on the status bar.

Render | Render Options

View | Render | Render Options prompts you with the following dialog box (Figure 3.9). It allows you to turn on or off the shading of the surfaces of shells as though they were illuminated from multiple light sources. It provides a way for you to realistically visualize the image of the model.

You have the option to apply different rendering percentages to different elements. Enter 100% for full rendering, 0% for no rendering, and anything in-between for partial rendering. The partial rendering (e.g. 50-80%) may be useful in identifying connectivity of elements to nodes.

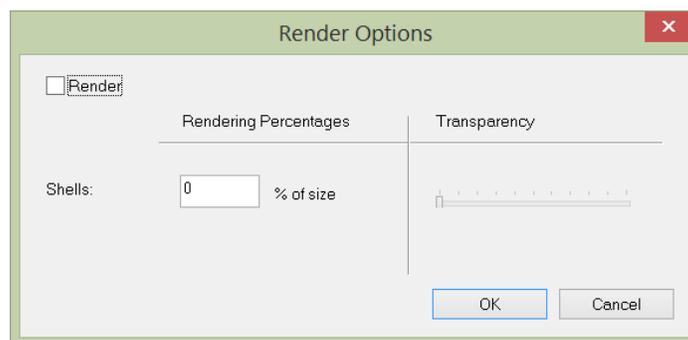


Figure 3.9

Render | Quick Render

View | Render | Quick Render turns rendering on or off. You may run this command by pressing F8.

Options | Drawing Grid

View | Options | Drawing Grid shows or hides the drawing grid. You may set up the grid by using the command Geometry | Drawing Grid. The grid coordinates are shown in the status bar while the mouse is moving in the model view. You may run this command by pressing F7.

Options | Global Axes

View | Options | Global Axes shows or hides the legend of the global axes in the bottom-left corner of the window. You may run this command by pressing F5.

Chapter 4: Geometry

The Geometry menu provides commands to define and/or assign curve steps; draw individual nodes and curves based on a drawing grid or existing nodes; parametrically generate lines of regular shape.

Curve Steps

Geometry | Steps prompts you with the following dialog box (Figure 4.1). It allows you to define and/or assign curve steps to selected curves in the model. An Id is assigned automatically to each curve step by the program and may not be changed. You may assign a label with 127 maximum characters to each curve step for easy identification.

You may add one or more curve steps by clicking the “New Rows” button. You may also print all sections in the list by clicking the “Print” button. The “Assign active curve step to currently selected curves” checkbox may be used to assign the active curve step to selected curves. The active curve step refers to the one that currently has focus in the list in the dialog box. In order for curve step assignments to take place, curves must be selected beforehand.

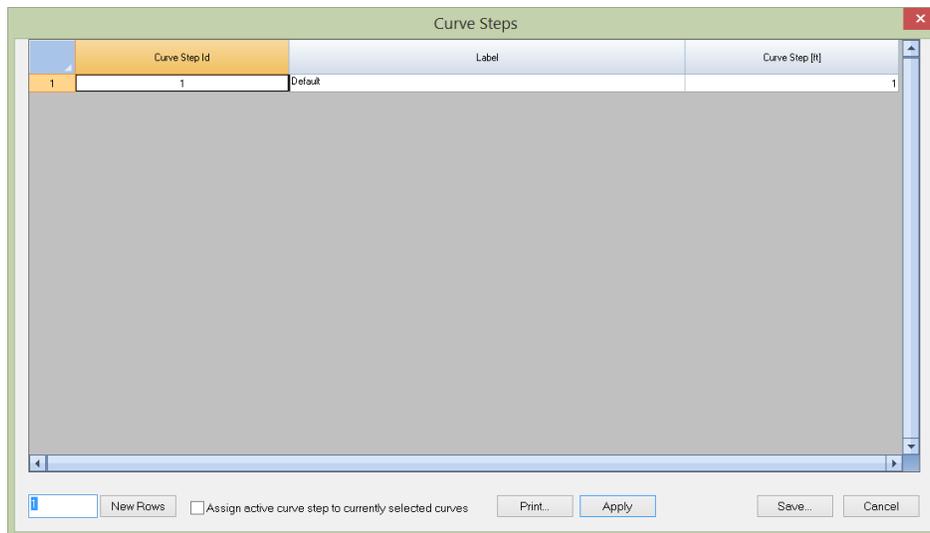


Figure 4.1

Drawing Grid

Geometry | Drawing Grid prompts you with the following dialog box (Figure 4.2). It allows you to generate a 1D, 2D or 3D rectangular grid for drawing or guidance. The distance list is a comma separated list that specifies multiple distances. For example, a distance list of “12, 2@14, 3@10” will generate distances of 12, 14, 14, 10, 10 and 10 in length units. You may specify a distance list for the X, Y, or Z direction or any combination of them.

You may specify an insertion point to translate and rotation parameters to rotate the grid. The drawing grid may be turned on or off by running the command View | Drawing Grid or by simply

pressing F7. You may regard the grid as a user defined coordinate system that can be changed at any time. The coordinates of the grid intersection under the mouse are displayed in the status bar. It helps you to identify correct points when drawing nodes or elements. The following example (Figure 4.3) shows the use of this command.

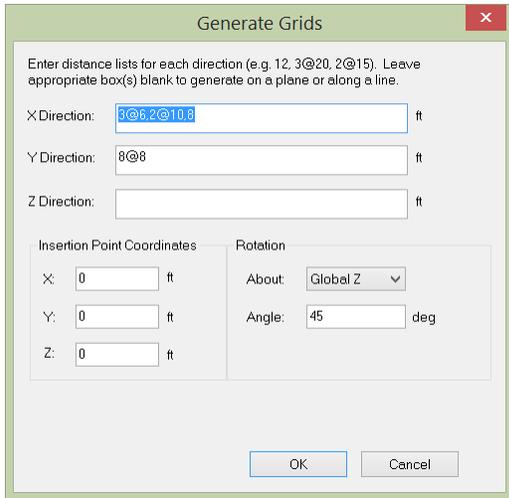


Figure 4.2

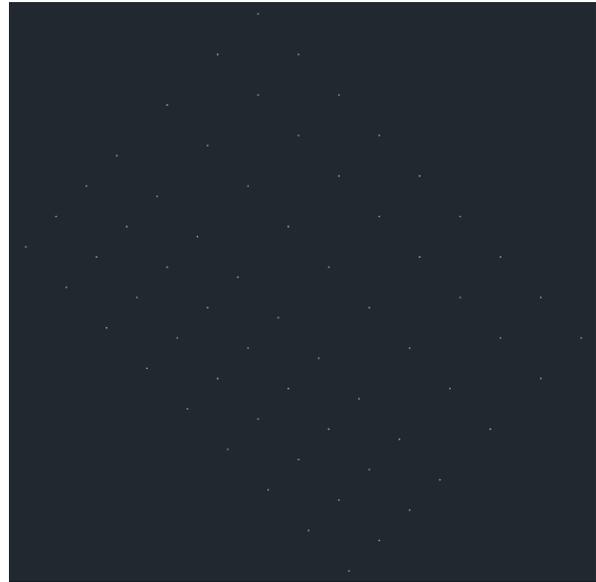
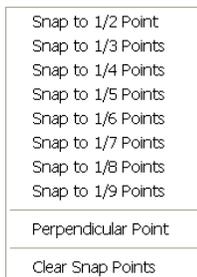


Figure 4.3

Object Snap

The Geometry | Object Snap pop-up menu provides options to snap node locations at 1/2, 1/3, ... 1/9th points on a curve under the mouse cursor. If you are drawing curves, you also have the option to snap to the perpendicular point on a curve from the last point. It is a good idea to turn off the drawing grid (by pressing F7) to avoid snapping to grid points while any of the snap options is on.



Draw Node

Geometry | Draw Node allows you to draw new nodes in the model. To draw a node, simply move the mouse, point to an intersection of the grid and click the left mouse button. You may also draw a node by entering nodal X, Y, and Z (optional) coordinates in the command window via the command “node”. This is very useful if you need to draw nodes outside the grid interactions. The command remains in effect until another command is selected, the right mouse button is clicked, or ESC is pressed.

Draw Line

Geometry | Draw Line allows you to draw new line (QL curve) in the model. To draw a line, simply move the mouse and click the left mouse button from point to point (Figure 4.4). The clicked points must be intersections on the grid or existing nodes. These points become the curve nodes. New nodes are created if necessary. Lines are drawn continuously. Right clicking the mouse once lets you start drawing lines from a new location. The lines drawn use the current curve step. You may use the Curve Steps in the Geometry menu to assign curve steps to them.

You may also specify a node by entering nodal X, Y and Z (optional) coordinates in the command window via the command “line”. This is very useful if you need to specify nodes outside the grid interactions. In addition, you may specify a node by entering an existing node number directly. You can combine the use of keyboard and mouse to draw curves.

You may turn on annotations for nodes and members while drawing. To do that, run View | Annotate. The command remains in effect until another command is selected, the right mouse button is clicked twice, or ESC is pressed.

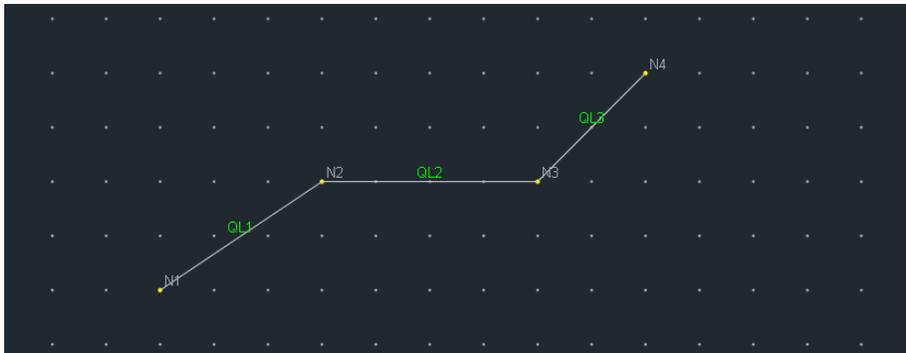


Figure 4.4

Draw Arc

Geometry | Draw Arc allow you to draw new arc (C curve) in the model. To draw an arc, simply move the mouse and click the left mouse button from point to point three times (Figure 4.5). The picked points must not be on the same line.

You may also specify a node by entering nodal X, Y and Z (optional) coordinates in the command window via the command “arc”. This is very useful if you need to specify nodes outside the grid interactions. In addition, you may specify a node by entering an existing node number directly. You can combine the use of keyboard and mouse to draw curves.

The command remains in effect until another command is selected, the right mouse button is clicked twice, or ESC is pressed.

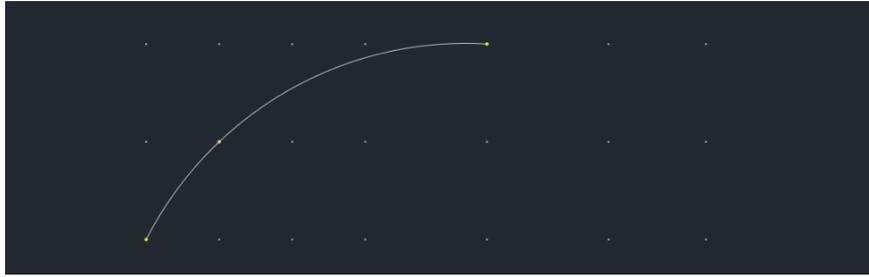


Figure 4.5

Draw Spline

Geometry | Draw Spline allow you to draw new spline (Q curve) in the model. To draw a spline, simply move the mouse and click the left mouse button from point to point at least four times (Figure 4.6).

You may also specify a node by entering nodal X, Y and Z (optional) coordinates in the command window via the command “spline”. This is very useful if you need to specify nodes outside the grid interactions. In addition, you may specify a node by entering an existing node number directly. You can combine the use of keyboard and mouse to draw curves.

The command remains in effect until another command is selected, the right mouse button is clicked twice, or ESC is pressed.



Figure 4.6

Quick Draw Line

Geometry | Quick Draw Line allows you to draw a line (QL curve) by specifying two nodes in the command window through the command “qline”.

Quick Draw Arc

Geometry | Quick Draw Arc allows you to draw an arc (C curve) by specifying three nodes (not on the same line) in the command window through the command “qarc”.

Quick Draw Spline

Geometry | Quick Draw Spline allows you to draw a spline (Q curve) by specifying at least four nodes in the command window through the command “qspline”.

Generate Lines

The Geometry | Generate Lines pop-up menu provides commands to quickly generate commonly used structural shapes in a model.

Generate Lines | Rectangular

Geometry | Generate Lines | Rectangular prompts you with the following dialog box (Figure 4.7). It allows you to quickly generate 1D lines, 2D lines (plane frame or grillage) or 3D lines (space frames). The distance list is a comma separated list that specifies multiple distances. For example, a distance list of “12, 2@14, 3@10” will generate distances of 12, 14, 14, 10, 10 and 10 in length unit. You may leave appropriate distance list(s) blank to generate on a plane or along a line. The generated members have the default curve step.

Generate Rectangular Lines

Enter distance lists for each direction (e.g. 12, 3@20, 2@15). Leave appropriate box(s) blank to generate on a plane or along a line.

X Direction: 5@1 ft

Y Direction: 5@1 ft

Z Direction: 5@1 ft

Insertion Point Coordinates

X: 0 ft

Y: 0 ft

Z: 0 ft

Rotation

About: Global Z

Angle: 0 deg

OK Cancel

Figure 4.7

The following example (Figure 4.8, 4.9) shows the uses of this command to generate a 2D frame (lines) on the XY plane with horizontal spans 10, 10, 18, 10, 10 ft. and vertical spans 15, 8, 8, 8 ft.

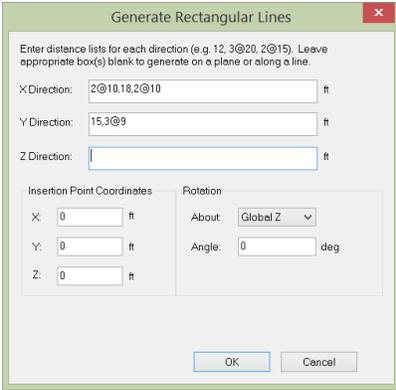


Figure 4.8

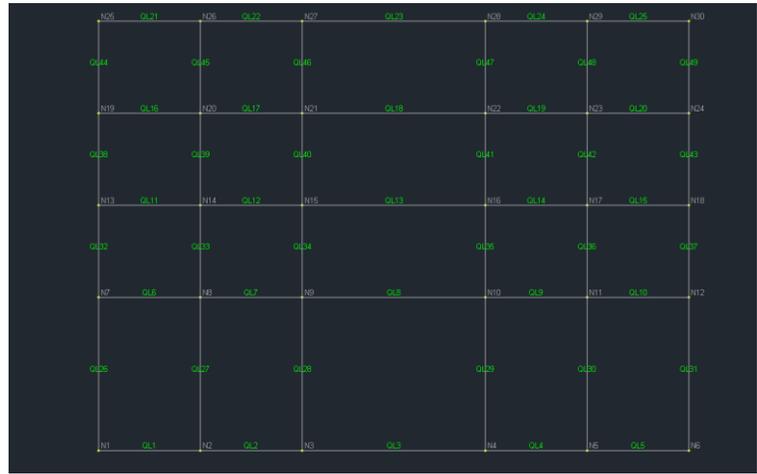


Figure 4.9

Generate | Cylindrical

Geometry | Generate Lines | Cylindrical prompts you with the following dialog box (Figure 4.11). It allows you to quickly generate 2D or 3D cylindrical lines. The distance list is a comma separated list that specifies multiple distances. For example, a distance list of “12, 2@14, 3@10” will generate distances of 12, 14, 14, 10, 10 and 10 in length unit. You may leave the Y direction distance list empty, in which case, a plane cylindrical lines will be generated. The generated members use the default curve step.

Using the input in Figure 4.11, a 3D cylindrical lines in Figure 4.12 is generated.

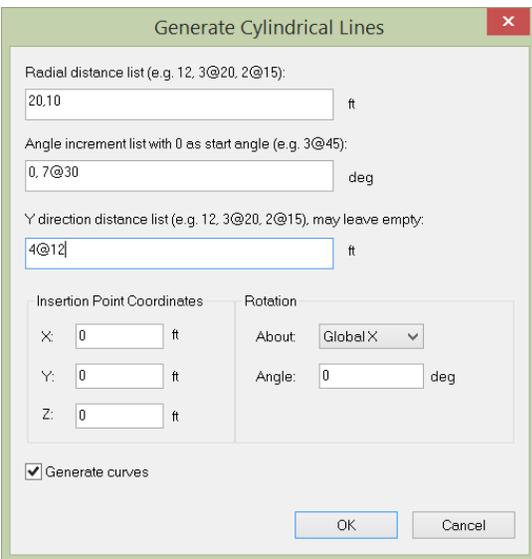


Figure 4.11

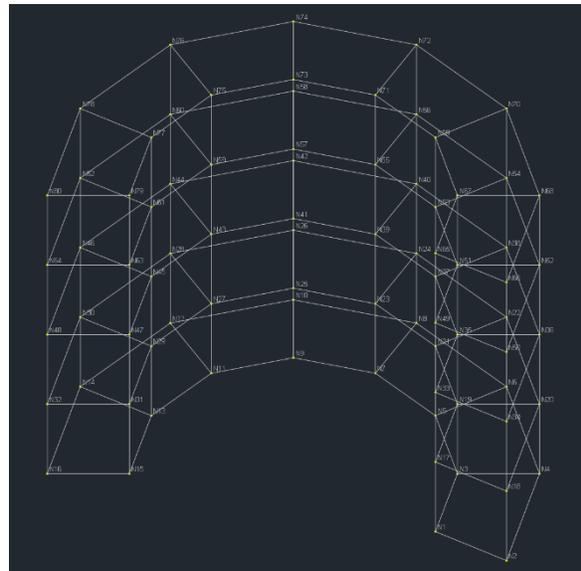


Figure 4.12

Generate Lines | Arc

Geometry | Generate Lines | Arc prompts you with the following dialog box (Figure 4.13). It allows you to quickly generate lines along an arc. You may specify an arc radius, the start and end angles, and the number of segments. You may specify an insertion point to translate and rotation parameters to rotate the generate lines. The generated lines use the default curve step.

The following example (Figure 4.14) shows lines generated along an arc using the input from Figure 4.13.

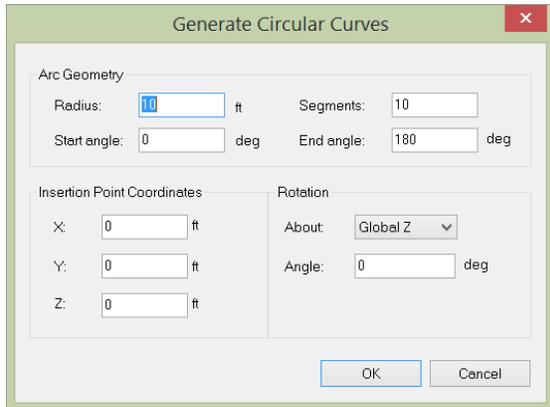


Figure 4.13

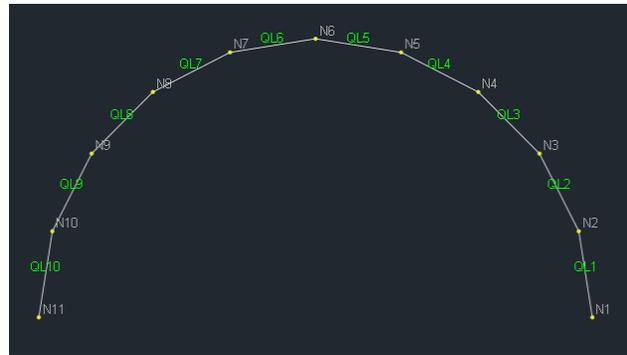


Figure 4.14

Chapter 5: Regions

The Regions menu provides commands to define and edit regions of different types.

Add Plane Region

Regions | Add Plane Region displays the following dialog (Figure 5.1) and allows you to add a plane region to mesh model. You must first select a closed sequence of curves as the boundary of the region. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate.

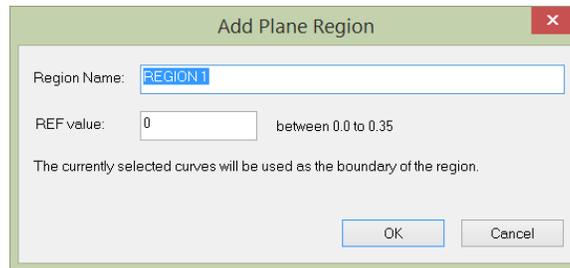


Figure 5.1

Add Cylinder Region

Regions | Add Cylinder Region displays the following dialog (Figure 5.2) and allows you to add a cylinder region to mesh model. You must first select a closed sequence of curves as the boundary of the region. Angle X, Angle Y, and Angle Z parameters are the directional angles of the straight line that defines the surface orientation. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate.

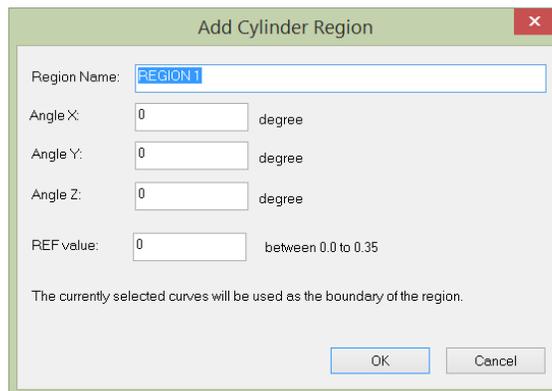


Figure 5.2

Add Sphere Region

Regions | Add Sphere Region displays the following dialog (Figure 5.3) and allows you to add a sphere region to mesh model. You must first select a closed sequence of curves as the boundary of the region. Center X, Center Y, and Center Z parameters are the coordinates of the sphere center. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate.

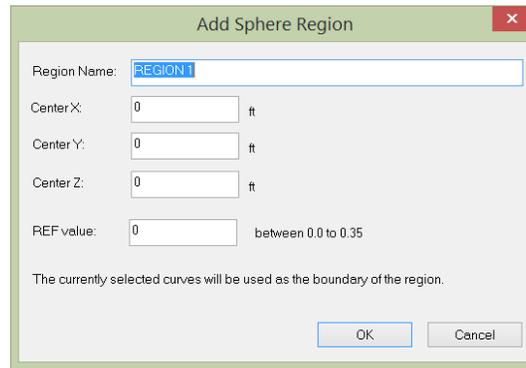


Figure 5.3

Add Rev Region

Regions | Add Rev Region displays the following dialog (Figure 5.4) and allows you to add a rev region to mesh model. You must first select a closed sequence of curves as the boundary of the region. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate.

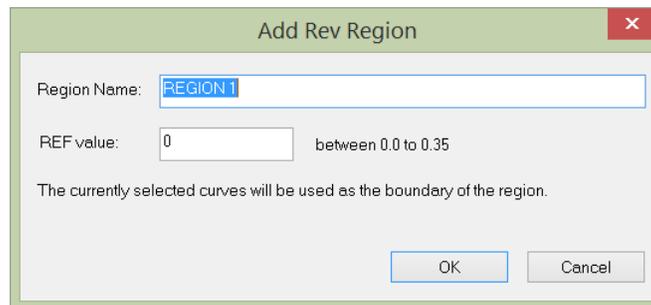


Figure 5.4

Add General Patch Region

Regions | Add General Patch Region displays the following dialog (Figure 5.5) and allows you to add a general patch region to mesh model. You must first select a closed sequence of curves as the boundary of the region. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate.

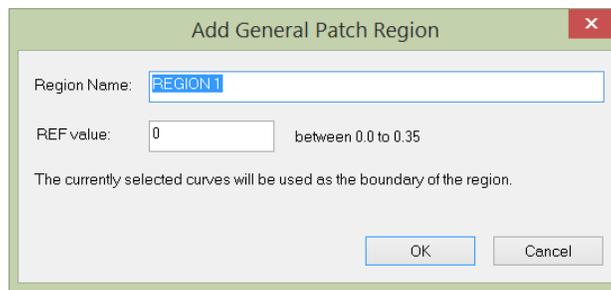


Figure 5.5

Add Hole

Regions | Add Hole displays the following dialog (Figure 5.6) and allows you to add a hole to a plane region. You must first select a closed sequence of curves as the boundary of the hole.

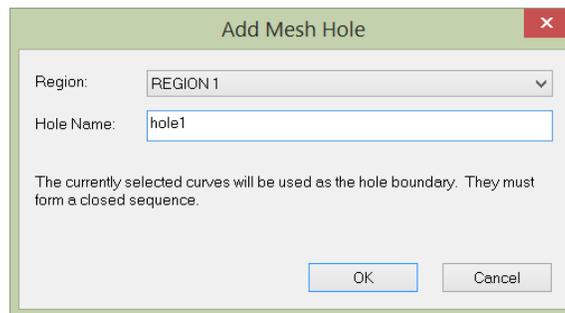


Figure 5.6

Add Internal Points

Regions | Add Internal Points displays the following dialog (Figure 5.7) and allows you to add internal points to a plane region. You must first select existing nodes as the location of the internal points in the mesh region. The STEP parameter is the desired shell4 size while RAD is the radius of the refined mesh around the internal points.

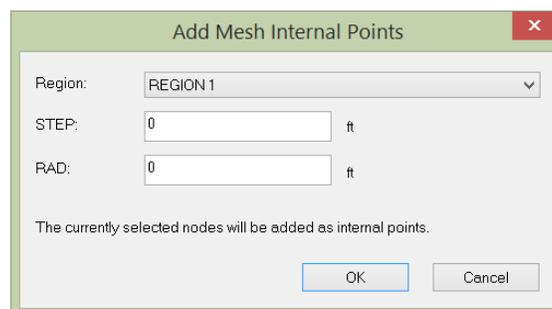


Figure 5.7

Add Tree

Regions | Add Tree displays the following dialog (Figure 5.8) and allows you to add a tree to a plane region. You must first select curves as the branches of the tree. *Please note that disconnected branches are not allowed. Tree curves must not form internal (closed) contours.*

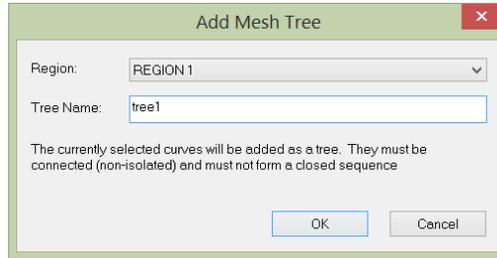


Figure 5.8

Edit Regions

Regions | Edit Regions are duplicate commands under Tables.

Delete Region

Regions | Delete Region displays the following dialog (Figure 5.9) and allows you to delete an existing region.

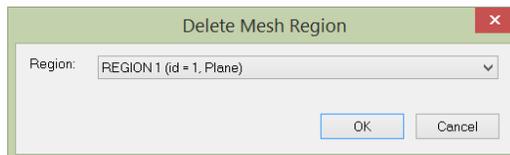


Figure 5.9

Delete Holes

Regions | Delete Holes displays the following dialog (Figure 5.10) and allows you to delete one or more holes in a plane region.

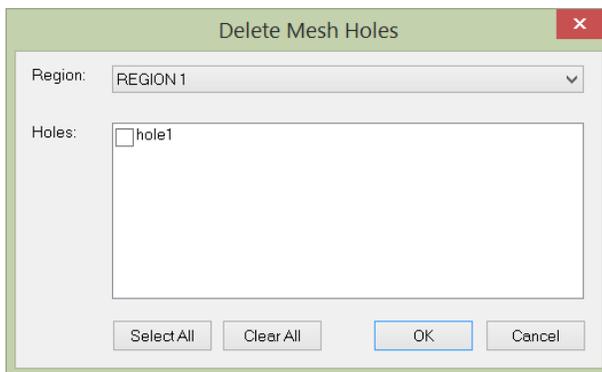


Figure 5.10

Delete Internal Points

Regions | Delete Internal Points displays the following dialog (Figure 5.11) and allows you to delete one or more internal points in a plane region.

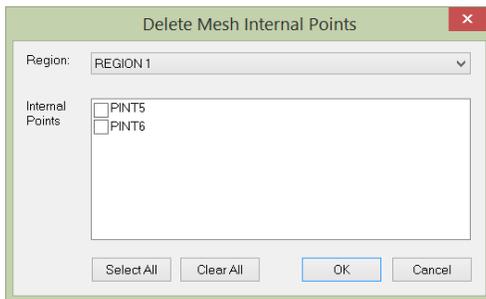


Figure 5.11

Delete Trees

Regions | Delete Trees displays the following dialog (Figure 5.12) and allows you to delete one or more trees in a plane region.

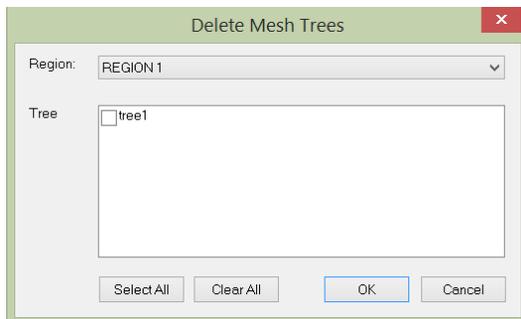


Figure 5.12

Clear Mesh Model

Regions | Clear Mesh Model allows you to clear the mesh model. This will delete all regions and its dependents such as holes, internal points and trees.

Activate Regions

Regions | Activate Regions displays the following dialog (Figure 5.13) and allows you to inactivate/activate one or more regions before generating mesh.

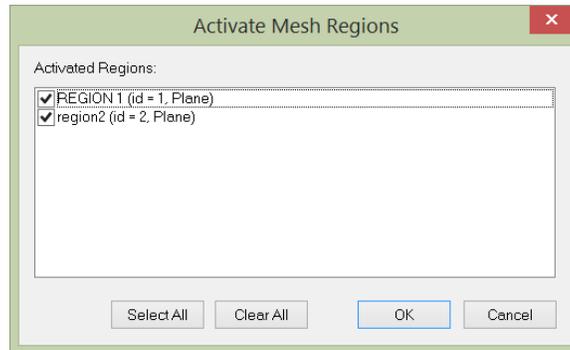


Figure 5.13

View Mesh Model

Regions | View Mesh Model shows or hide mesh model.

Chapter 6: Tables

The Tables menu provides commands to define and edit nodes, curves and regions through spreadsheets.

Curve Steps

Tables | Curve Steps command is the same as Geometry | Curve Steps. It is provided here for convenience only.

Nodes

Tables| Nodes prompts you with the following dialog box (Figure 6.1). It allows you to enter nodes in a spreadsheet. Each node includes the nodal coordinates and the selection status. You may not modify the nodal Ids.

An empty row is allowed if all rows below it are empty (except the nodal Id and status fields). You may not delete the existing nodes in the dialog box. To delete the existing nodes, you must dismiss this dialog box and run the command Edit | Delete.

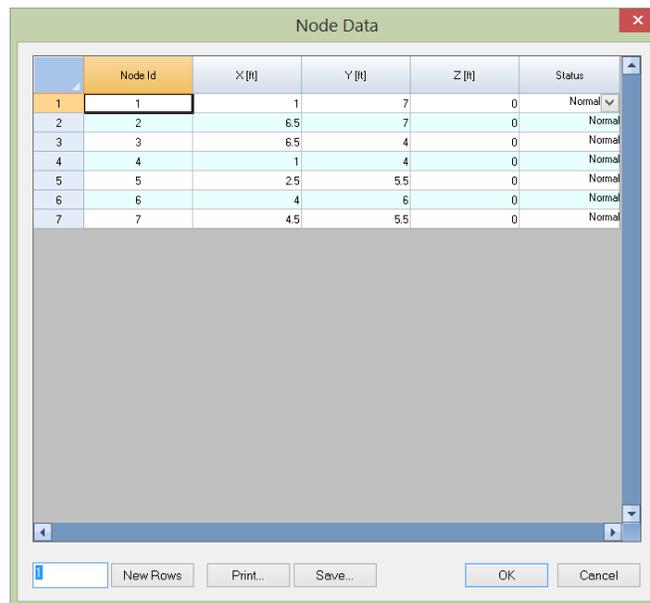


Figure 6.1

Curves

Tables| Curves prompts you with the following dialog box (Figure 6.2). It allows you to enter curves in a spreadsheet. Each curve includes the Ids of curve nodes, curve STEP Ids, and the selection status. You may not modify the curve Id. All other Ids must be valid (defined). Curve STEP combo boxes are provided for you to correctly pick and apply proper curve STEP Ids to selected

curves. An empty row is allowed if all rows below it are empty except the curve Id and status fields. You may not delete the existing curves in this dialog box. To delete the existing curves, you must dismiss this dialog box and run the command Edit | Delete.

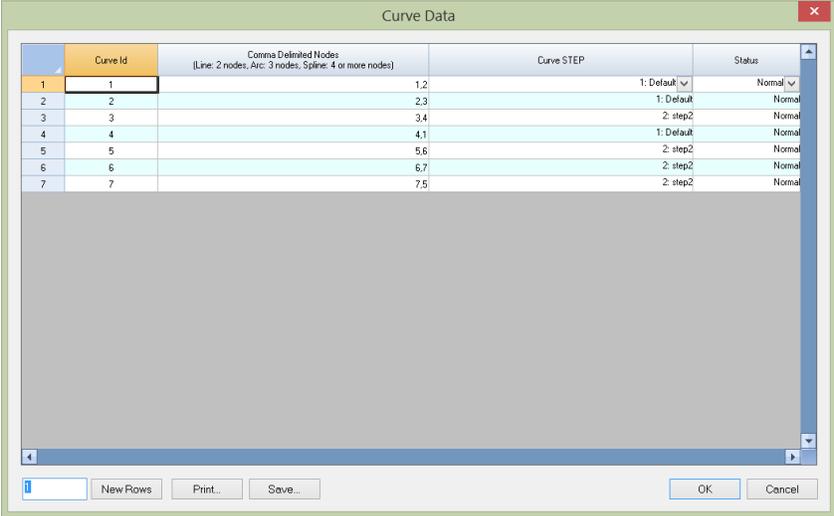


Figure 6.2

Plane Regions

Tables| Plane Regions prompts you with the following dialog box (Figure 6.3). It allows you to define and/or edit plane regions in a spreadsheet. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate. Boundary curve Ids must form a closed sequence. The boundary curves may appear in arbitrary order. Their actual order will be restored automatically during the preprocessing stage.

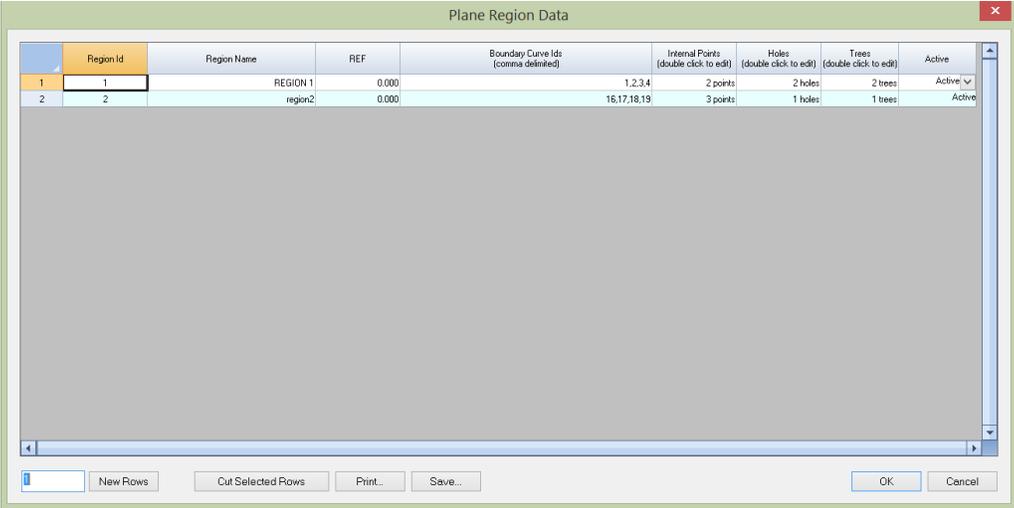


Figure 6.3

When double-clicking a Internal Points cell, the following dialog (Figure 6.4) will be displayed that allows you to define and/or edit internal points within the selected region. The STEP parameter is the desired shell4 size while RAD is the radius of the refined mesh around the internal points.

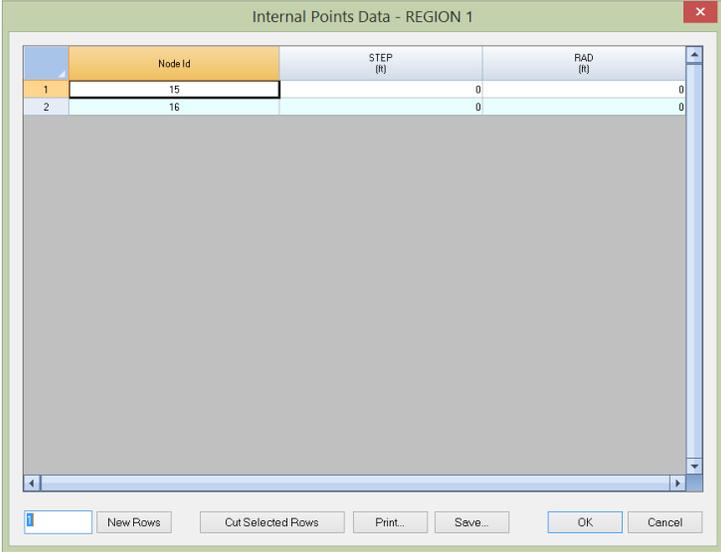


Figure 6.4

When double-clicking a Holes cell, the following dialog (Figure 6.5) will be displayed that allows you to define and/or edit holes within the selected region. The hole curves may appear in arbitrary order. Their actual order will be restored automatically during the preprocessing stage.

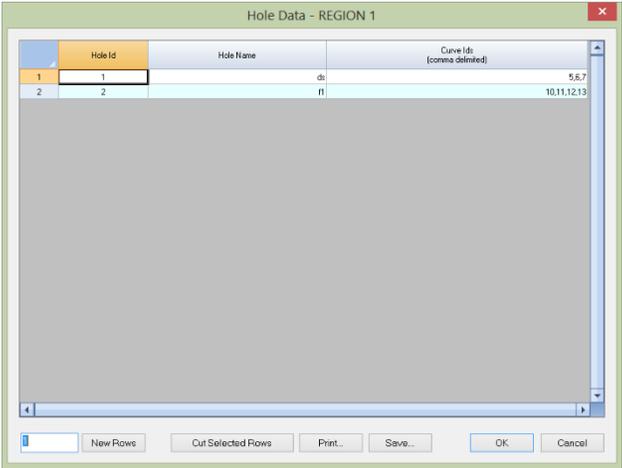


Figure 6.5

When double-clicking a Trees cell, the following dialog (Figure 6.6) will be displayed that allows you to define and/or edit trees within the selected region. Boundary curve Ids of the hole must form a closed sequence. The tree curves may appear in arbitrary order. Their actual order will be restored automatically during the preprocessing stage.

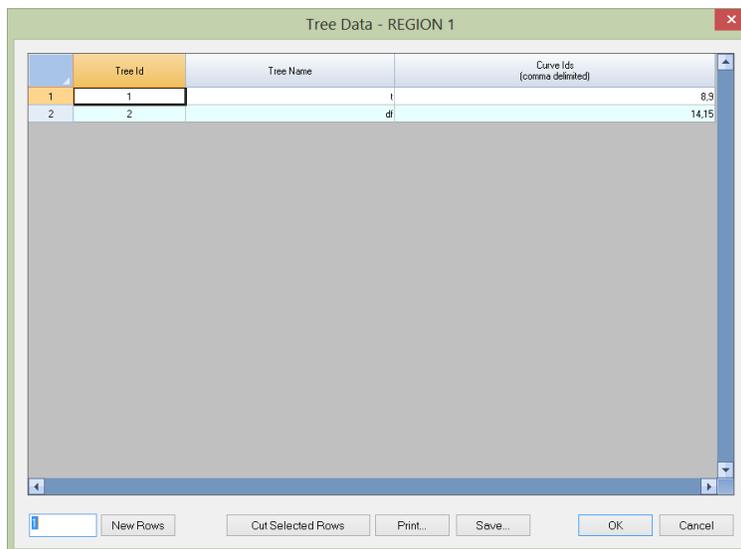


Figure 6.6

Cylinder Regions

Tables| Cylinder Regions prompts you with the following dialog box (Figure 6.7). It allows you to define and/or edit cylinder regions in a spreadsheet. Angle X, Angle Y, and Angle Z parameters are the directional angles of the straight line that defines the surface orientation. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate. Boundary curve Ids must form a closed sequence. The boundary curves may appear in arbitrary order. Their actual order will be restored automatically during the preprocessing stage.

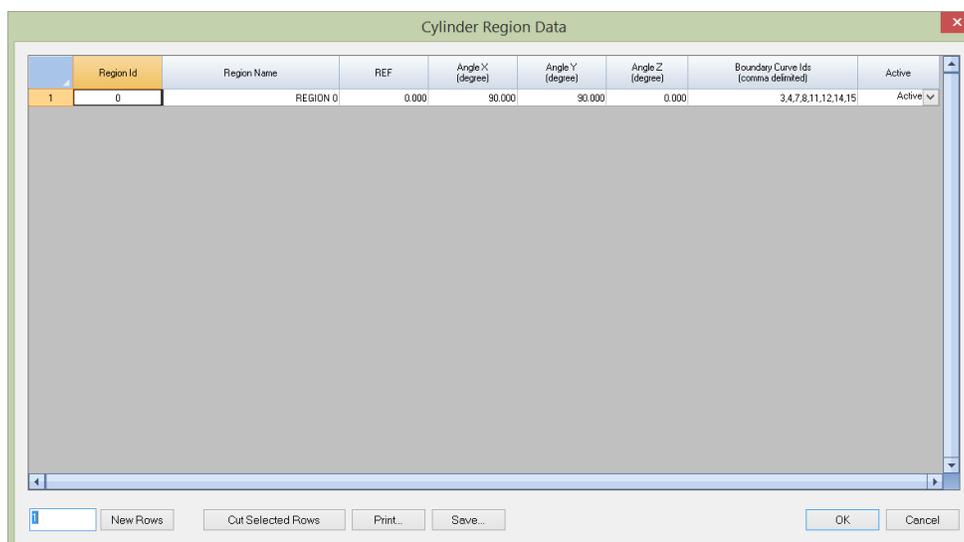


Figure 6.7

Sphere Regions

Tables| Sphere Regions prompts you with the following dialog box (Figure 6.8). It allows you to define and/or edit sphere regions in a spreadsheet. Center X, Center Y, and Center Z parameters are the coordinates of the sphere center. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate. The number of edges in a sphere region can be 2, 3, or 4. Each edge can contain any number of curves of different types. Curves in all edges must form a close sequence. The curves in each edge may appear in arbitrary order. Their actual order will be restored automatically during the preprocessing stage.

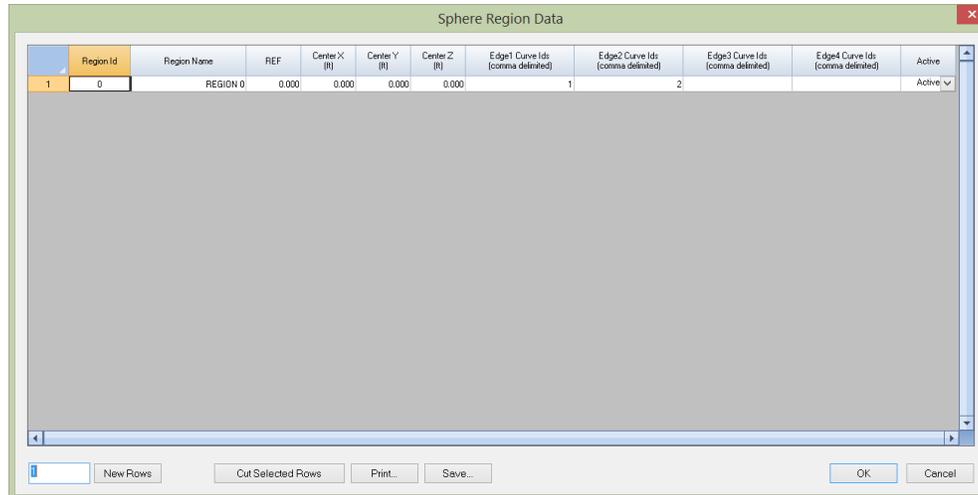


Figure 6.8

Rev Regions

Tables| Rev Regions prompts you with the following dialog box (Figure 6.9). It allows you to define and/or edit rev regions in a spreadsheet. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate. The number of edges in a sphere region must be 4. Each edge can contain any number of curves of different types. Two edges must contain arc curves for the top and bottom arcs. Curves in all edges must form a close sequence. The curves in each edge may appear in arbitrary order. Their actual order will be restored automatically during the preprocessing stage.

General Patch Regions

Tables| General Patch Regions prompts you with the following dialog box (Figure 6.10). It allows you to define and/or edit general patch regions in a spreadsheet. Refer to the Introduction chapter for the meaning of REF value. If the variation of the curve steps in the region boundary is small, REF value of 0 is appropriate. The number of edges in a sphere region can be 2, 3, or 4. Each edge can contain any number of curves of different types. Two edges must contain arc curves for the top and bottom arcs. Curves in all edges must form a close sequence. The curves in each edge may appear in arbitrary order. Their actual order will be restored automatically during the preprocessing stage.

Rev Region Data

Region Id	Region Name	REF	Edge1 Curve Ids (comma delimited)	Edge2 Curve Ids (comma delimited)	Edge3 Curve Ids (comma delimited)	Edge4 Curve Ids (comma delimited)	Active	
1	0	R0	0.000	14	27.7	29	12.28	Active

New Rows Cut Selected Rows Print... Save... OK Cancel

Figure 6.9

General Patch Region Data

Region Id	Region Name	REF	Edge1 Curve Ids (comma delimited)	Edge2 Curve Ids (comma delimited)	Edge3 Curve Ids (comma delimited)	Edge4 Curve Ids (comma delimited)	Active	
1	0	R0	0.000	1	2	3	4	Active

New Rows Cut Selected Rows Print... Save... OK Cancel

Figure 6.10

Chapter 7: Generate

The Generate menu provides commands to generate mesh based on the mesh model or an external mesh file.

Generate Mesh

Generate | Generate Mesh displays the following dialog (Figure 7.1) and allows you to generate mesh based on the current mesh model. The default mesh result file (.r3a) will be displayed if the mesh generation is successful. You have the options to include the mesh definition in the result file. During the mesh generation, an important mesh raw file (*.SUR) is created. You can save this file so you can generate mesh directly using this file within QuadMaker or Real3D programs.

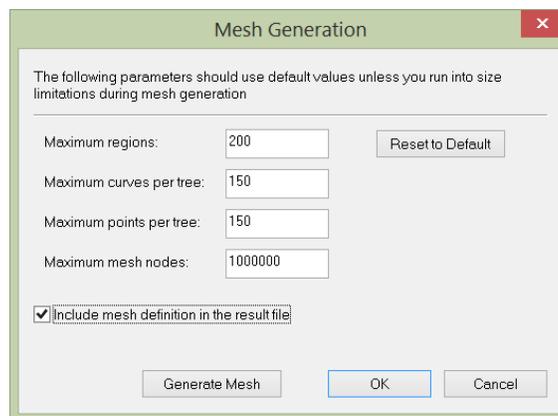


Figure 7.1

Generate Mesh From File

Generate | Generate Mesh From File allows you to generate mesh from a previously generated mesh model raw file (*.SUR).

Chapter 8: Settings

The Settings menu provides commands related to settings for model data and graphical entities in model views. Some of these settings may be applied beyond the current model, that is, they may be saved for use in future models.

Units & Precisions

Settings | Units & Precisions prompts you with the following dialog box (Figure 8.1). You may select different units and precisions for various physical measurements used in the model. You may run this command as many times as you like. You may convert existing data associated with a unit in the model by checking or unchecking the check box to the right of that unit. For example, if you mistakenly enter all nodal coordinates in a wrong length unit, you may select the correct length unit and uncheck the conversion checkbox to correct nodal coordinate input.

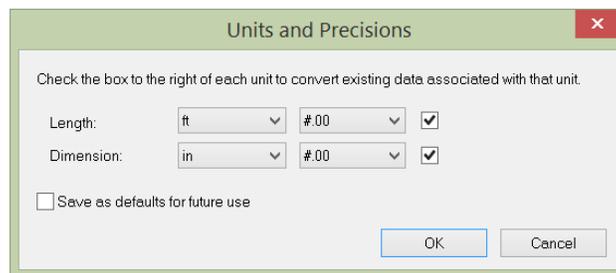


Figure 8.1

You may set the precision for each unit in either decimal or scientific format. Precision settings are used in displaying data in graphics and spreadsheets.

By checking “Save as defaults for future use”, units will be saved in the registry for use in future models.

Data Options

Settings | Data Options prompts you with the following dialog box (Figure 8.2).

Distance tolerance is used for distance comparisons in certain commands such as Edit | Merge Nodes. Distances less than distance tolerance are considered zero by the program.

The “Undo/redo levels” sets the maximum undo/redo levels which the program will perform. The program requires extra computer memory for each undo/redo level. The default undo levels setting is 200. Depending on your computer memory and model sizes, you may want to set undo levels to be smaller.

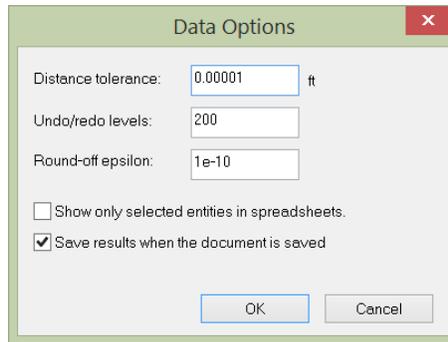


Figure 8.2

Round-off epsilon is used to truncate floating point numbers such as those found in results.

New Origin

Settings | New Origin prompts you with the following dialog box (Figure 8.3). It allows you to reset the model origin. In particular, the origin may be set at the current model center. This allows you to center the model so its view may be rotated more smoothly.

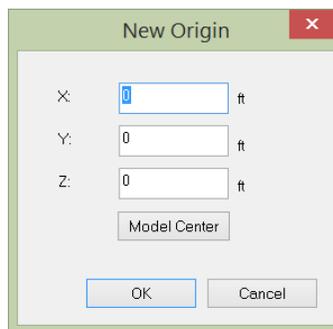


Figure 8.3

Vertical Axis

Settings | Vertical Axis prompts you with the following dialog box (Figure 8.4). It allows you to set either global Y (default) or global Z as the vertical axis for the model. Vertical axis has impact on the preset views such as front/back, left/right, top/bottom and isometric views. It also affects the way real time rotation works.

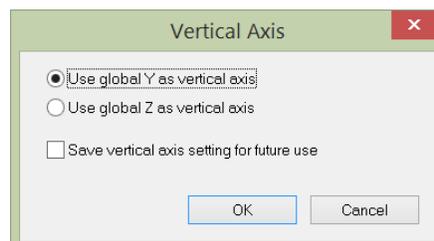


Figure 8.4

Graphic Scales

Settings | Graphic Scales prompts you with the following dialog box (Figure 8.5). You may set scales for graphical entities. By “Save as defaults for future use”, these scales will be saved in the registry for future uses.

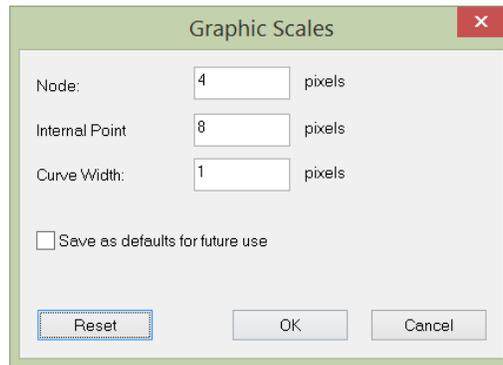


Figure 8.5

Colors

Settings | Colors prompts you with the following dialog box (Figure 8.6). It allows you to set colors of different graphical entities in the model. You may modify the color(s) of one or more items at a time. By checking “Use white background for image captures”, a white background will be used for the captured image even if a different background color is used in the model views. This option will reduce the amount of ink required to print the images. Color settings can be optionally saved to the registry for future use.

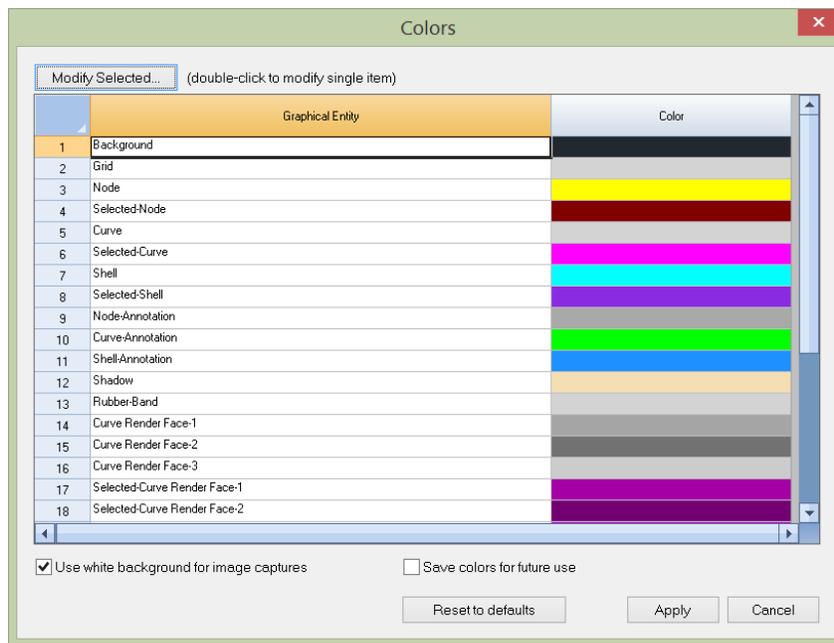


Figure 8.6

Preferences

Settings | Preferences prompts you with the following dialog box (Figure 8.7). The “Data Folder” determines the default folder or directory when a file is saved or opened. The “Automatic file backup period” determines how frequently the model files are saved automatically. Enter 0 for no auto-backups. Backup files have the extension “qdm.qdm”. By default, rubber-banding is enabled while drawing curves, shells or bricks. You may want to disable this feature if your computer graphic card is not fully OpenGL compatible.

Preferences are always saved in the registry for future use.

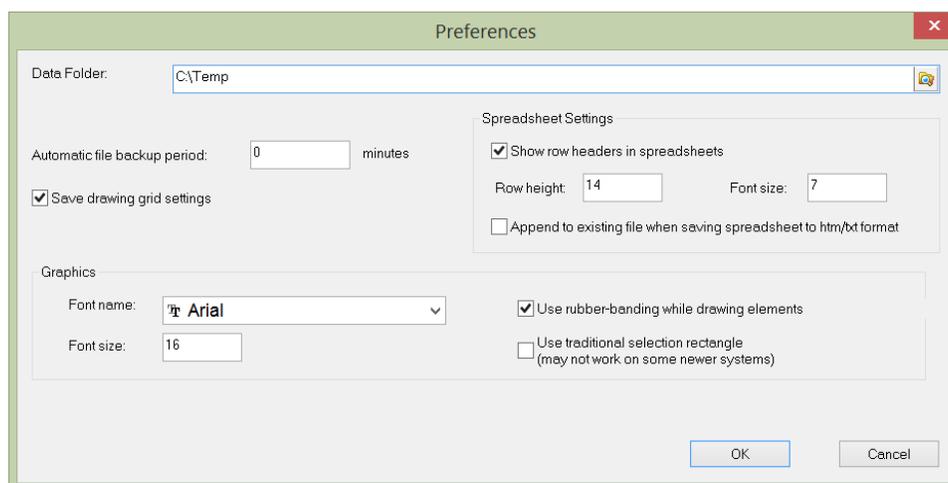


Figure 8.7

Enable/Disable Hardware Acceleration

Settings | Enable/Disable Hardware Acceleration allows you to turn on or off OpenGL hardware acceleration. By default, OpenGL hardware acceleration is turned on. If you encounter graphics problems (due to faulty graphics drivers) in the software, you may want to try to disable hardware acceleration. The graphics performance may degrade significantly but should be acceptable on most modern systems.

This command may also be useful when using virtual Windows on Mac machine, and ARM64-based Windows.

Toolbars | Main Toolbar

Settings | Toolbars | Main Toolbar shows or hides the main toolbar.

Toolbars | View Toolbar

Settings | Toolbars | View Toolbar shows or hides the toolbar that contains commands for controlling the view.

Toolbars | Status Bar

Settings | Toolbars | Status Bar shows or hides the status bar.

Toolbars | Command Bar

Settings | Toolbars | Command Bar shows or hides the command window.

Tools | Copy Command History

Settings | Tools | Copy Command History copies the history in the command window to the clipboard. You may then paste the command history content to a text editor using Ctrl + V. A command history is associated with each open document.

Tools | Clear Command History

Settings | Tools | Clear Command History clears the history in the command window. You may want to copy the command history before running this command.

Chapter 9: Window

The Window menu provides commands to create new windows and arrange existing windows. In this program, a window may be used interchangeably with a view. The program has two types of views: model view and result view.

New Window

Window | New Window creates a new window or view based on the current view. You may create different model views with different display settings with respect to zooming, panning, annotations etc.

Close

Window | Close closes the current window.

Close All

Window | Close All closes all windows that are currently open. You will be prompted to save file(s) if necessary.

Tile Horizontal

Window | Tile Horizontal arranges all opened windows horizontally.

Tile Vertical

Window | Tile Vertical arranges all opened windows vertically.

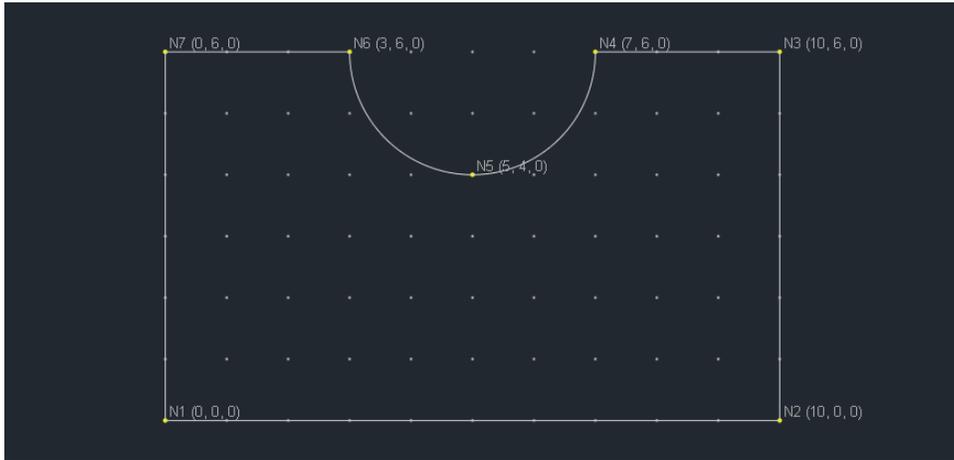
Tile Cascade

Window | Cascade arranges all opened windows in an overlapped manner

Tutorials

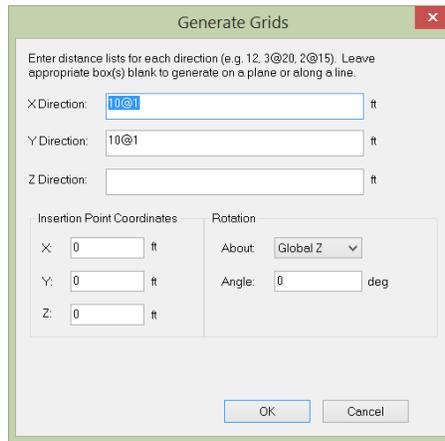
Tutorial 1 – Plane Region

In this tutorial, we are going to use various methods to create the following plane mesh model.



Method 1 – Through GUI

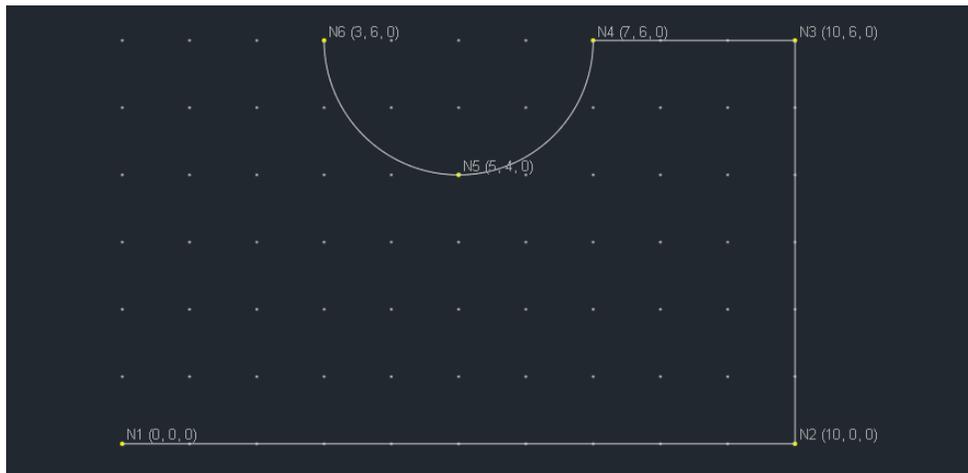
1. Go to Geometry | Drawing Grid. Set distance lists 10@1 for both X Direction and Y Direction. Click on OK button. This will create a grid of 10 x 10 with grid cell width and height 1 (ft). We can turn on annotations for node numbers, coordinates by running View | Annotate.



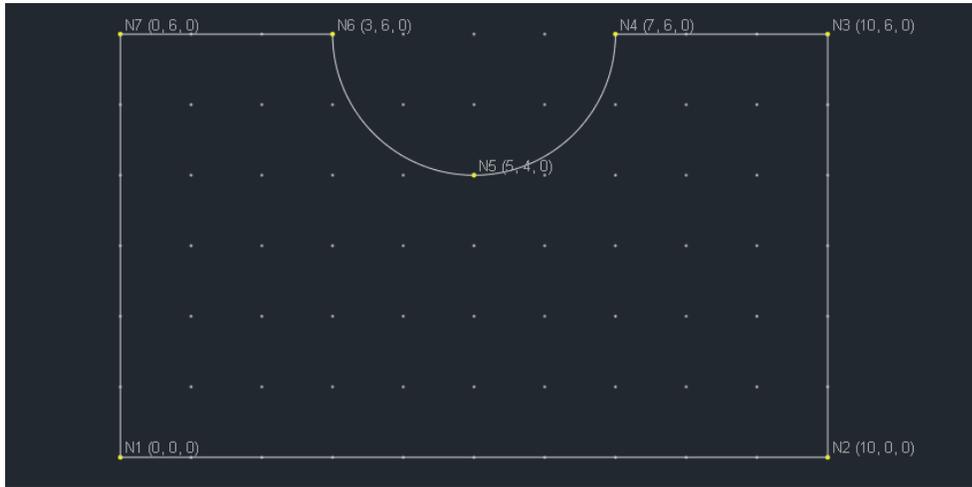
2. Go to Geometry | Draw Line menu. Now the mouse cursor becomes a pencil ready for you to draw lines. As you move the mouse over the grid points, their coordinates are shown on the status bar. Pick (0, 0, 0) as the first point (aka. node), (10, 0, 0) as the second point, (10, 6, 0) as the third point, and (7, 6, 0) as the fourth point. Right click the mouse twice (or press ESC key once) to exit the Draw Line mode. You can turn on annotations for node number and coordinates by View | Annotate. Now your screen should look like this:



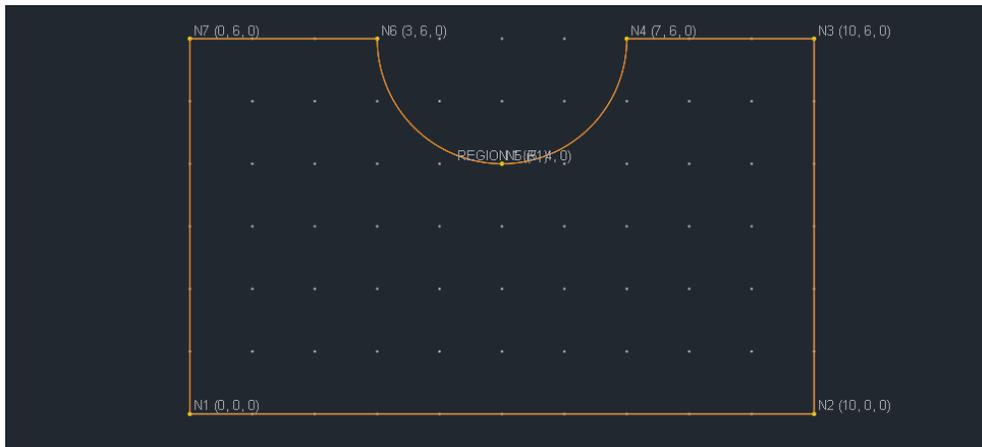
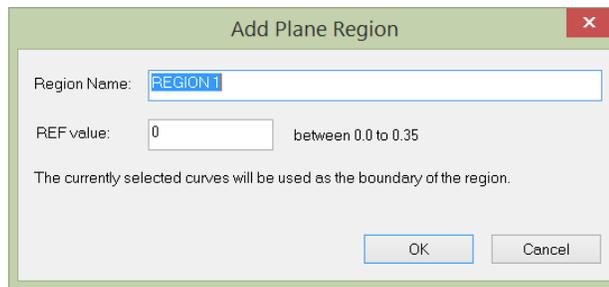
- Go to Geometry | Draw Arc menu. Now the mouse cursor becomes a pencil ready for you to draw arcs. Pick $(7, 6, 0)$ as the first point, $(5, 4, 0)$ as the second point, and $(3, 6, 0)$ as the third point. Right click the mouse once (or press ESC key once) to exit the Draw Arc mode. Now your screen should look like this:



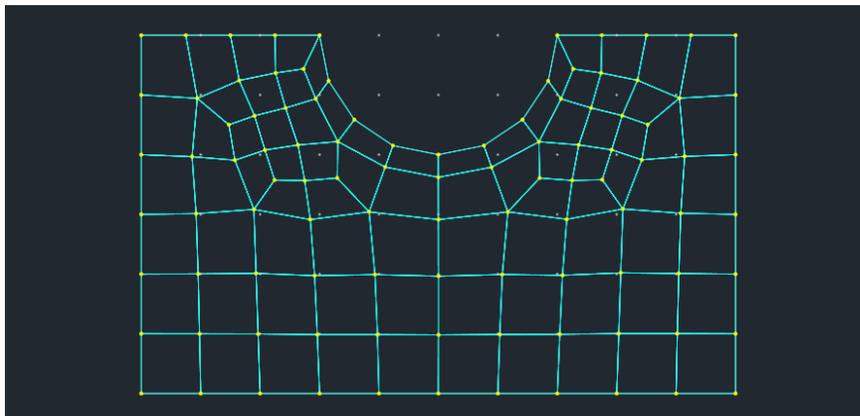
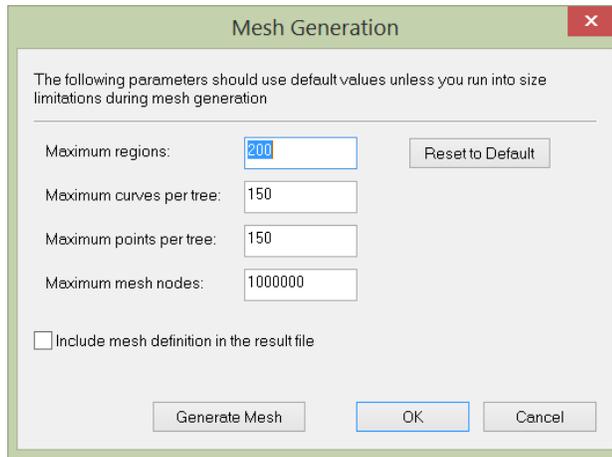
- Go to Geometry | Draw Line menu. Now the mouse cursor becomes a pencil ready for you to draw lines. Pick $(3, 6, 0)$ as the first point, $(0, 6, 0)$ as the second point, $(0, 0, 0)$ as the third point. Right click the mouse twice to exit the Draw Line mode. Now your screen should display a closed boundary of the region.



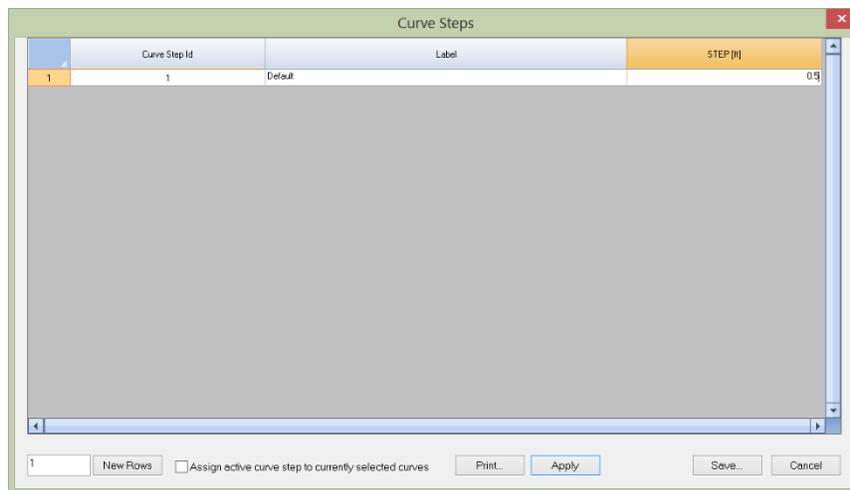
- Select all the curves by mouse window select or simply by press CTRL + A. All curves are now highlighted denoting that they are selected. Now go to Regions | Add Plane Region. Accept the default region name and REF value. Click on OK button. This creates a plane region with the boundary shown. You can show or hide mesh model by View | Mesh Model.



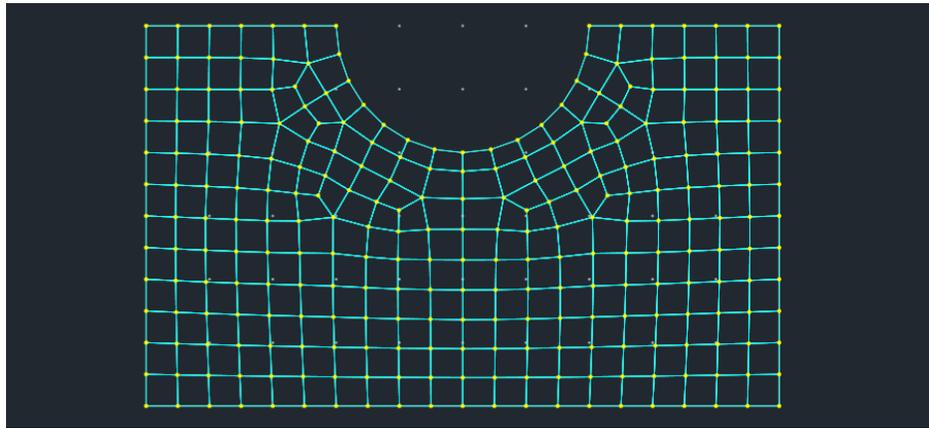
- Now we can generate mesh by Generate | Generate Mesh. Accept the default mesh generation parameters and click on Generate Mesh button. Now a separate result window displays the generated mesh result. To avoid clutter, you can turn off the annotations for node number and node coordinates by running View | Annotate. The generated node and shell data are available from Tables | Nodes and Tables | Shell4s.



7. All curves so far use the default curve step of 1 ft. This will make the mesh too coarse. We can rectify this by first closing the result window, then go to Geometry | Curve Step. Modify the step for “Default” from 1 ft to 0.5 ft. Click on Apply button. Note, we can create more curve steps here and assign different curve steps to different curves. For now, we are going to use an uniform curve step for all curves. Lets save the mesh model to a file called “Tutorial_01.qdm”.



8. Now we can run Generate | Generate Mesh again. This results a finer generated mesh.



9. During the mesh generation process, an important raw mesh SUR file (Tutorial_01.SUR) is generated. You can use this file to generate mesh in QuadMaker or Real3D program. You can also export the generated node and element data to a text file by running File | Export Nodes & Shells. This text file can then be used by other programs to import the mesh data.

Method 2 – Through Command Window

1. If the command window is not visible, press F3.
2. Type in command “line”. At the command prompt, type in coordinate “0, 0” then press Enter key; type in coordinate “10, 0” then press Enter key; type in coordinate “10,6” then press Enter key; type in coordinate “7,6” then press Enter key. This will create the three lines as shown below. Press ESC key to exit the Draw Line mode. Notice we are omitting Z coordinate in the command window. 0 will be used for the Z coordinate for each point entered.

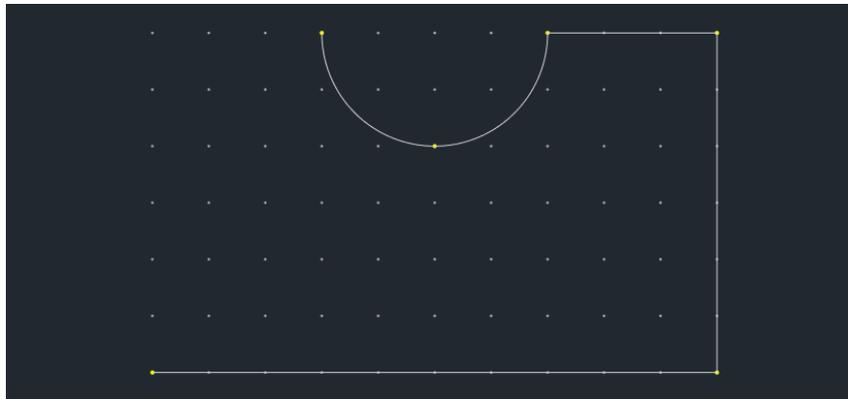
```

Command Window
line
Nodal Coordinates: 0,0
Nodal Coordinates: 10,0
Nodal Coordinates: 10,6
Nodal Coordinates: 7,6
Enter node ID or nodal coordinates:

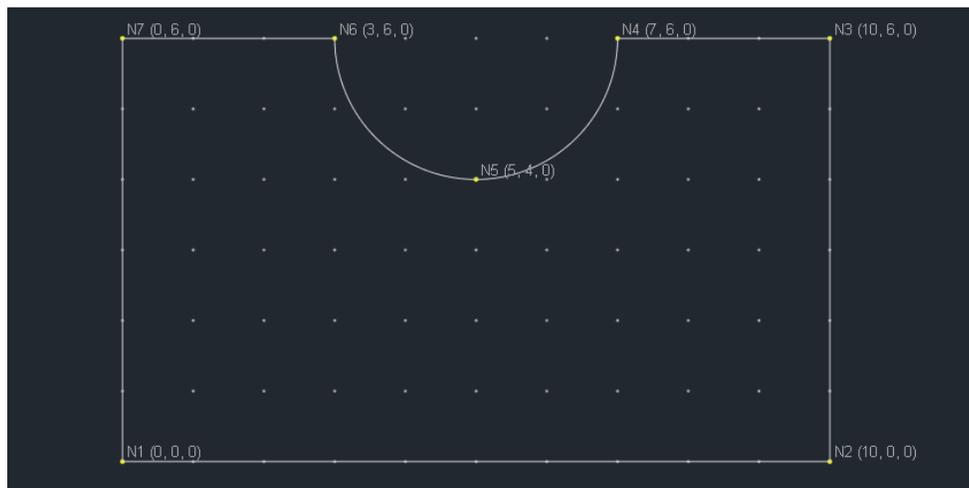
```



Type in command “arc”. At the command prompt, type in coordinate “7, 6” then press Enter key; type in coordinate “5, 4” then press Enter key; type in coordinate “3,6” then press Enter key; Press ESC key to exit the Draw Arc mode. Now your screen should look like this.



3. Type in command “line” again. At the command prompt, type in coordinate “3, 6” then press Enter key; type in coordinate “0, 6” then press Enter key; type in coordinate “0,0” then press Enter key; Press ESC key to exit the Draw Line mode. At this point, we have all boundary curves of the region.



4. The rest of the steps are similar to those in Method 1

Method 3 – Through Spreadsheets

- Go to Tables | Nodes, generate 6 new rows by entering 6 in the bottom-left input box and clicking on “New Rows” button, then enter the node coordinates in the spreadsheet as shown below. Click on OK. This will create all 6 nodes (aka. points). Notice node ids are automatically assigned and cannot be changed. An empty row is allowed if all rows below it are empty (except the nodal Id and status fields). You may not delete the existing nodes in the dialog box. To delete the existing nodes, you must dismiss this dialog box and run the command Edit | Delete.

The screenshot shows the 'Node Data' dialog box with a table containing 7 rows of data. The columns are Node Id, X [ft], Y [ft], Z [ft], and Status. The data is as follows:

	Node Id	X [ft]	Y [ft]	Z [ft]	Status
1	1	0	0	0	Normal
2	2	10	0	0	Normal
3	3	10	6	0	Normal
4	4	7	6	0	Normal
5	5	5	4	0	Normal
6	6	3	6	0	Normal
7	7	0	6	0	Normal

At the bottom of the dialog box, there is an input field with the number '6', a 'New Rows' button, 'Print...' and 'Save...' buttons, and 'OK' and 'Cancel' buttons.

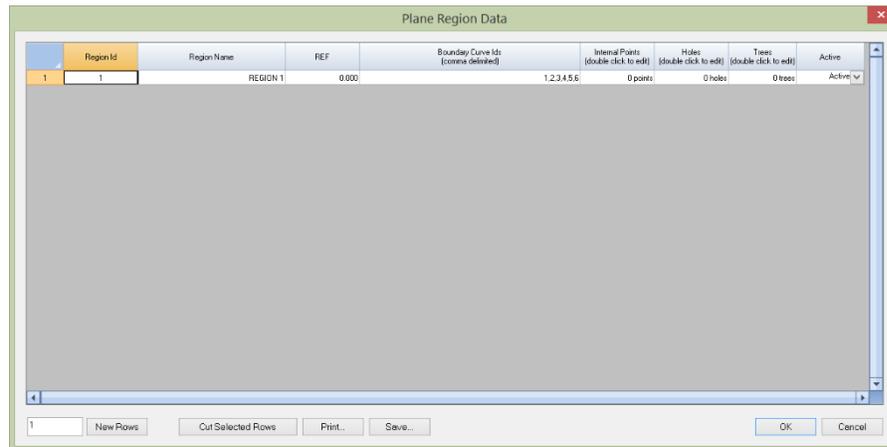
- Go to Tables | Curves, generate 6 new rows by entering 6 in the bottom-left input box and clicking on “New Rows” button. Enter a comma delimited nodes and select curve step for each curve in the spreadsheet as shown below. Click on OK. This will create all 6 curves needed for the region. Notice curve ids are automatically assigned and cannot be changed. An empty row is allowed if all rows below it are empty except the curve Id and status fields. You may not delete the existing curves in this dialog box. To delete the existing curves, you must dismiss this dialog box and run the command Edit | Delete.

The screenshot shows the 'Curve Data' dialog box with a table containing 6 rows of data. The columns are Curve Id, Comma Delimited Nodes (Line: 2 nodes, Arc: 3 nodes, Spline: 4 or more nodes), Curve STEP, and Status. The data is as follows:

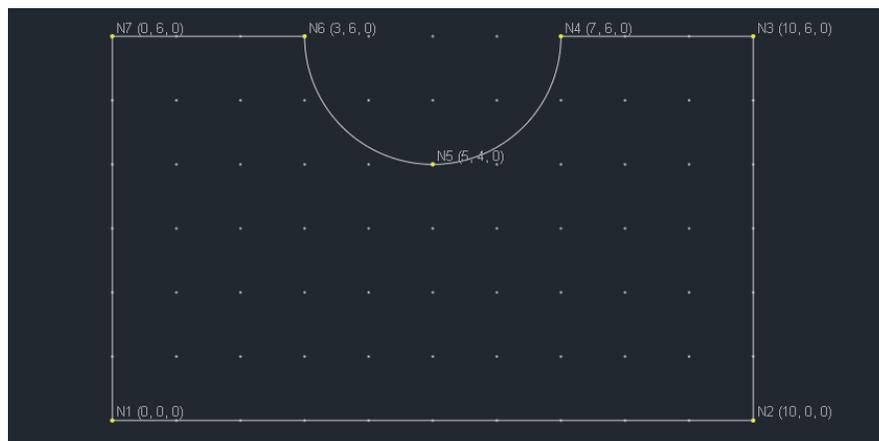
	Curve Id	Comma Delimited Nodes (Line: 2 nodes, Arc: 3 nodes, Spline: 4 or more nodes)	Curve STEP	Status
1	1	1,2	1: Default	Normal
2	2	2,3	1: Default	Normal
3	3	3,4	1: Default	Normal
4	4	4,5,6	1: Default	Normal
5	5	6,7	1: Default	Normal
6	6	7,1	1: Default	Normal

At the bottom of the dialog box, there is an input field with the number '6', a 'New Rows' button, 'Print...' and 'Save...' buttons, and 'OK' and 'Cancel' buttons.

- Go to Tables | Plane Regions. Generate 1 new row and enter the region info as below. Note the boundary curve Ids can be entered in arbitrary order. The solver will restore the curve order automatically. Click on OK button to create the plane region.



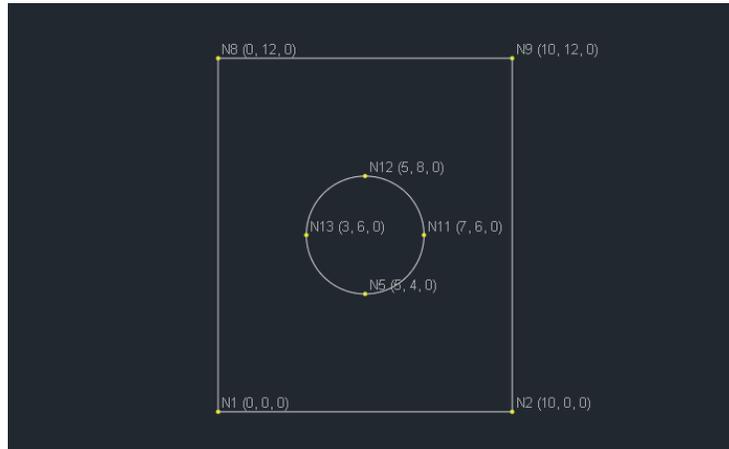
Now we should see the screen as the following.



- The rest of the steps are similar to those in Method 1.

Tutorial 2 – Plane Region with a Hole

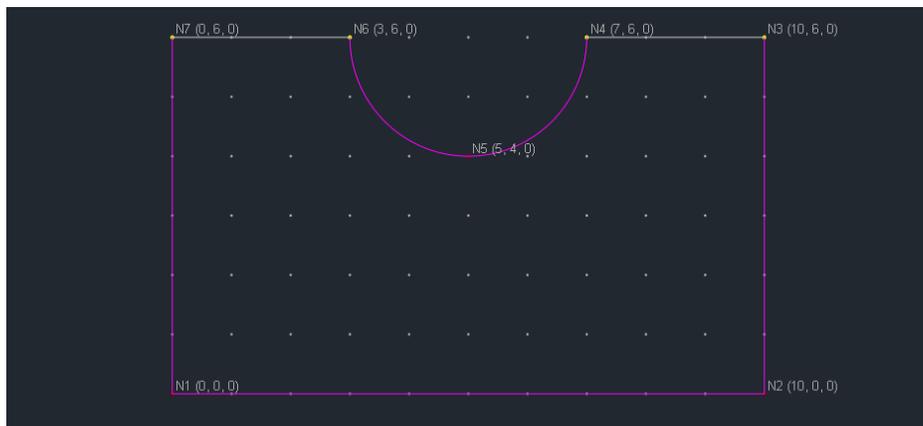
In this tutorial, we are going to create the following plane mesh model a hole.



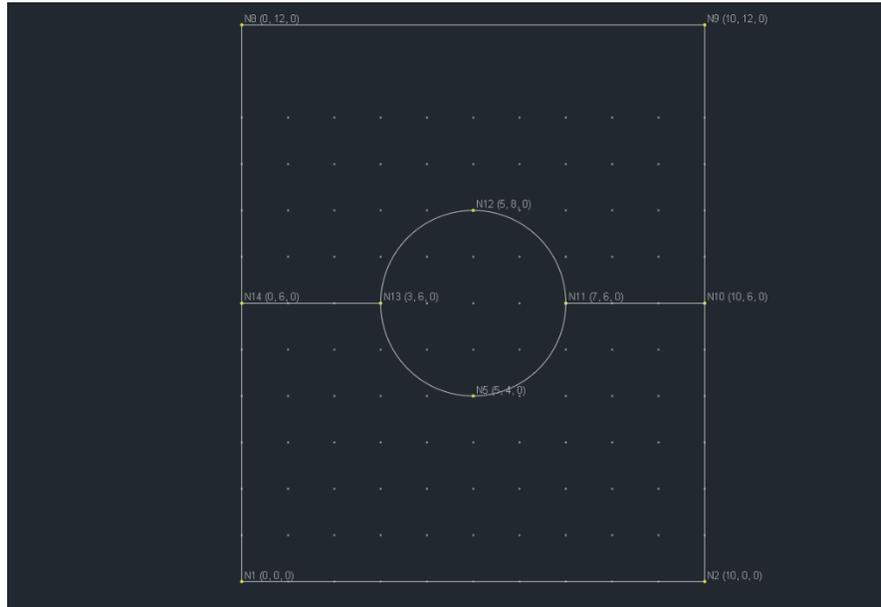
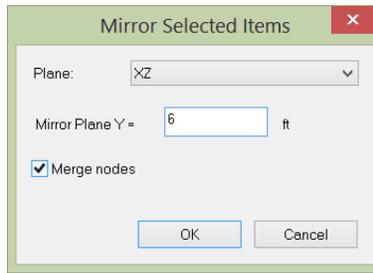
Method 1: Use Two Regions

This model is the sum of the model in the Tutorial 1 and its reflection. We can use Edit | Mirror command to create the reflected half of the mesh model.

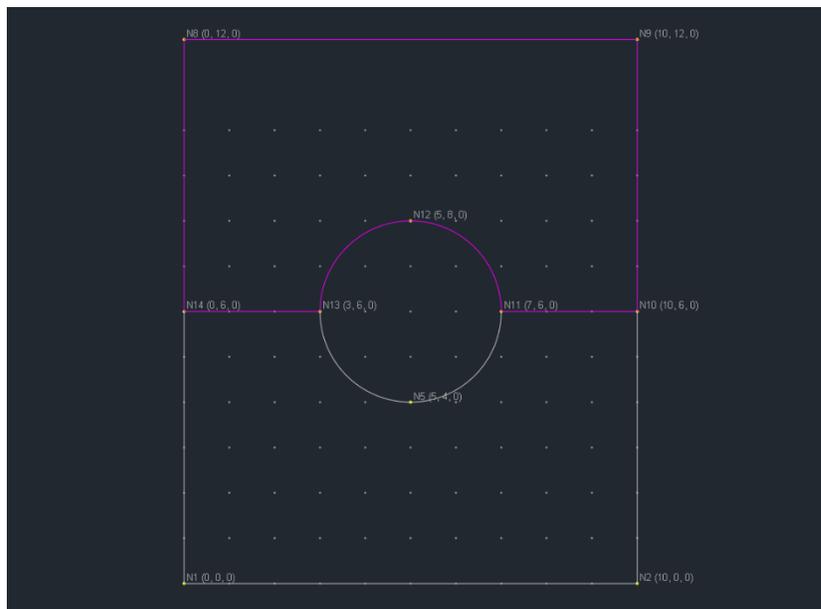
1. Open the “Tutorial_01.qdm” file saved in the previous tutorial. Save it as “Tutorial_02a.qdm”.
2. Hide the mesh model by View | Mesh Model so we can see our selection in the following step.
3. Select all the curves except the top two horizontal ones to avoid creating duplicate curves after mirroring command.

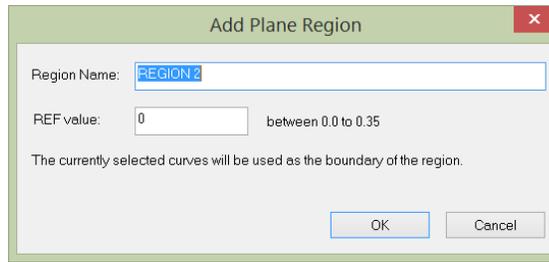


4. Go to Edit | Mirror, set the mirror parameters as below.

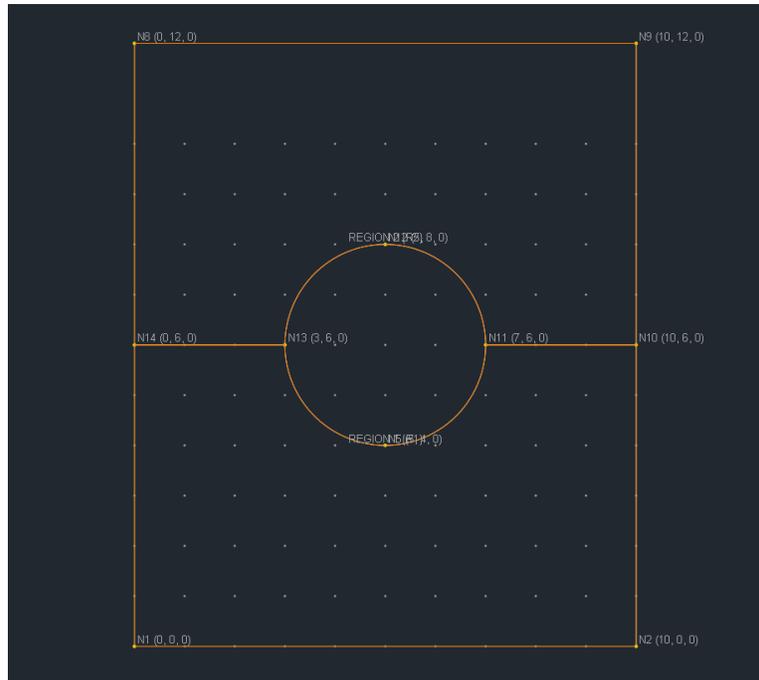


5. Select the reflected part of the mesh model. Add a new region “Region 2” by Regions | Add Plane Region as below.

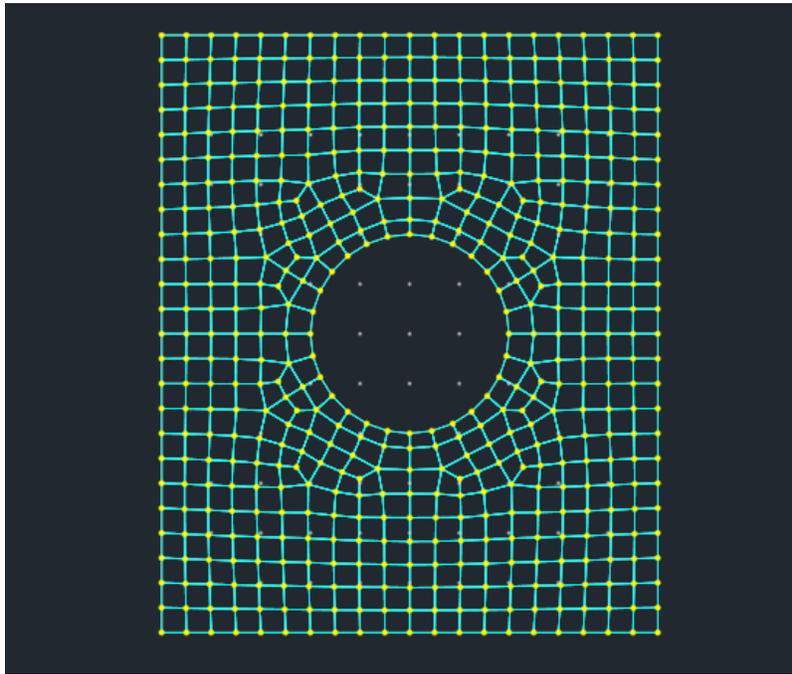




6. Show the mesh model by View | Mesh Model

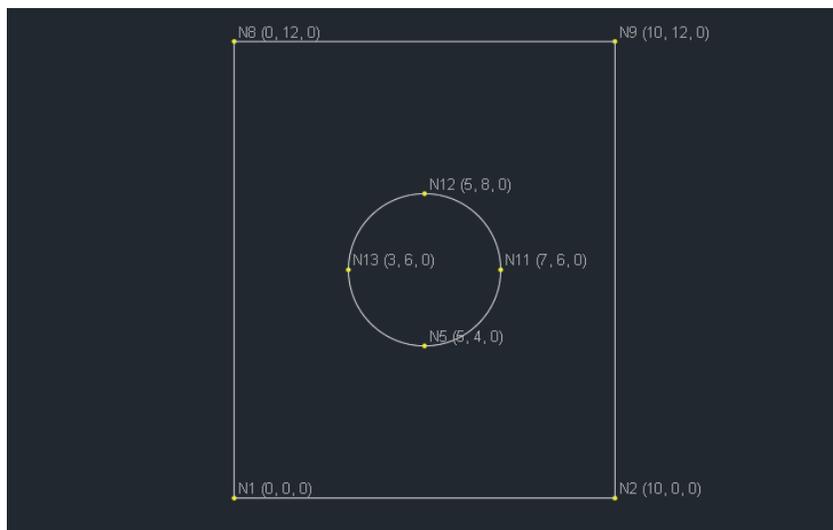


7. Run Generate | Generate Mesh. The mesh result would look like the follow. Turn off anotations to avoid clutter by running View | Annotate. The generated node and shell data are available from Tables | Nodes and Tables | Shell4s.

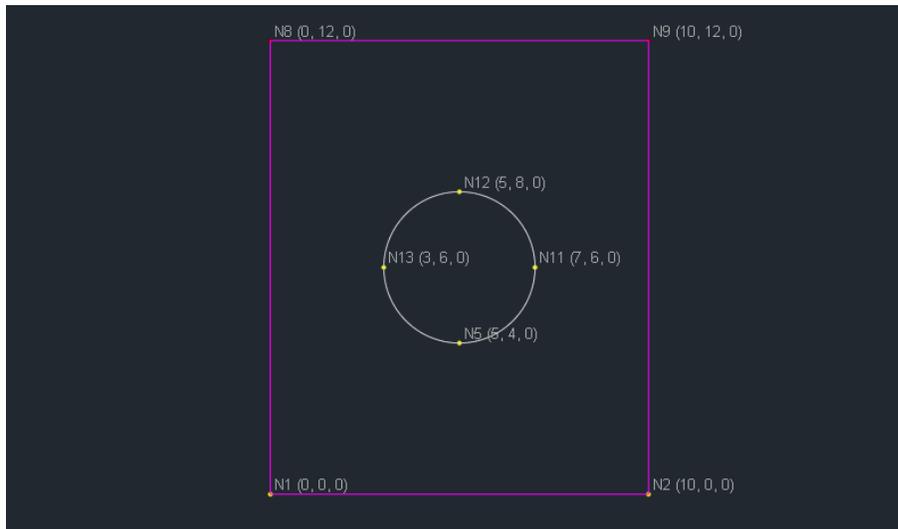


Method 2: Use One Region with One Hole

1. Start a new mesh model file. Save it as “Tutorial_02.qdm”.
2. Draw four line curves and two arc curves as in Tutorial 1



3. Select the four line curves as the boundary of the region. Go to Regions | Add Plane Region.



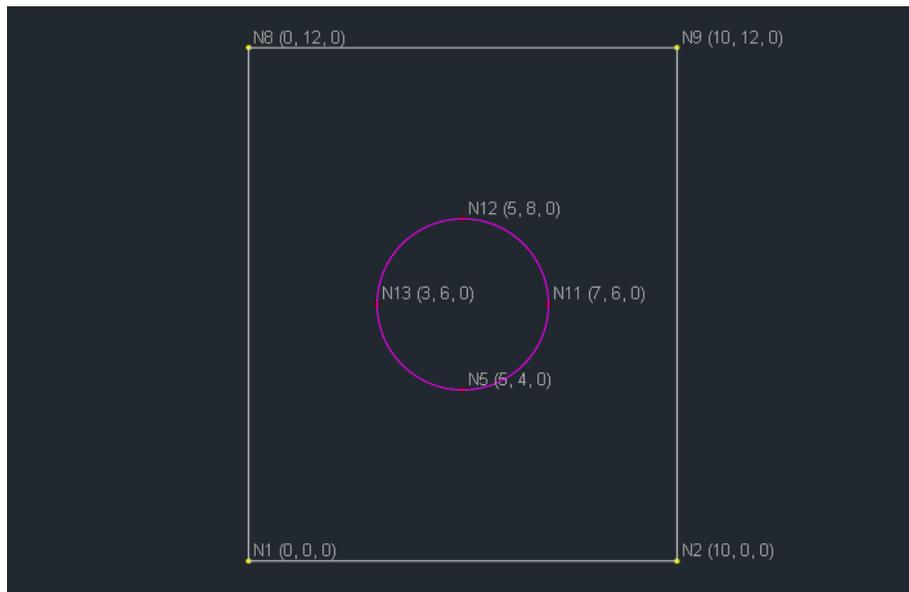
Add Plane Region ✖

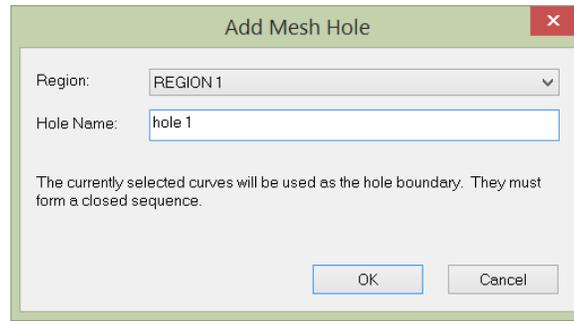
Region Name:

REF value: between 0.0 to 0.35

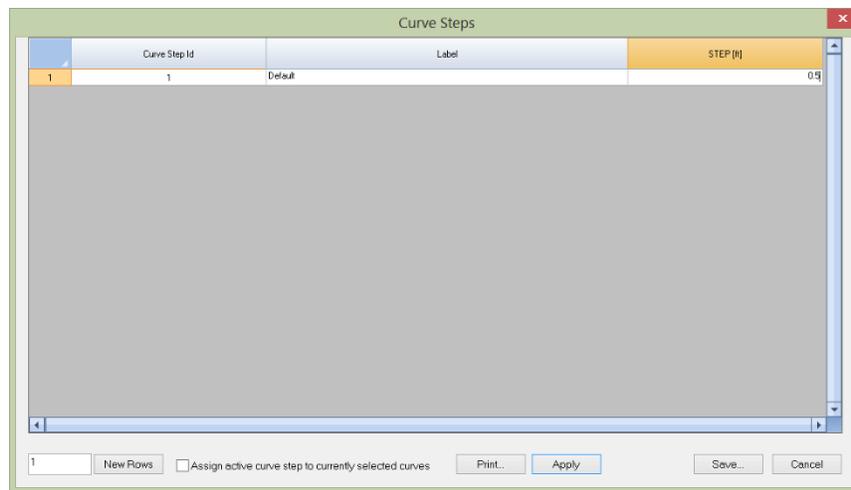
The currently selected curves will be used as the boundary of the region.

4. Select the two arcs. Go to Regions | Add Hole.

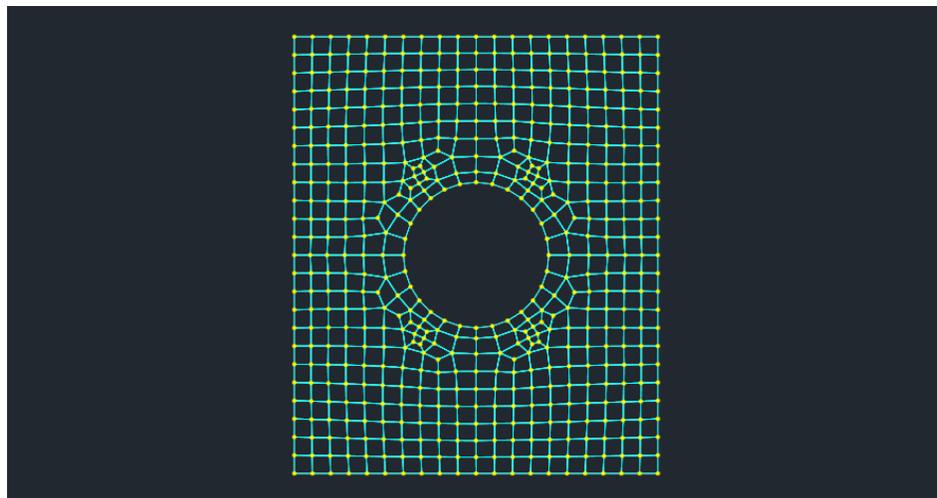




5. All curves so far use the default curve step of 1 ft. This will make the mesh too coarse. We can rectify this by first closing the result window, then go to Geometry | Curve Step. Modify the step for “Default” from 1 ft to 0.5 ft. Click on Apply button. Note, we can create more curve steps here and assign different curve steps to different curves. For now, we are going to use an uniform curve step for all curves.

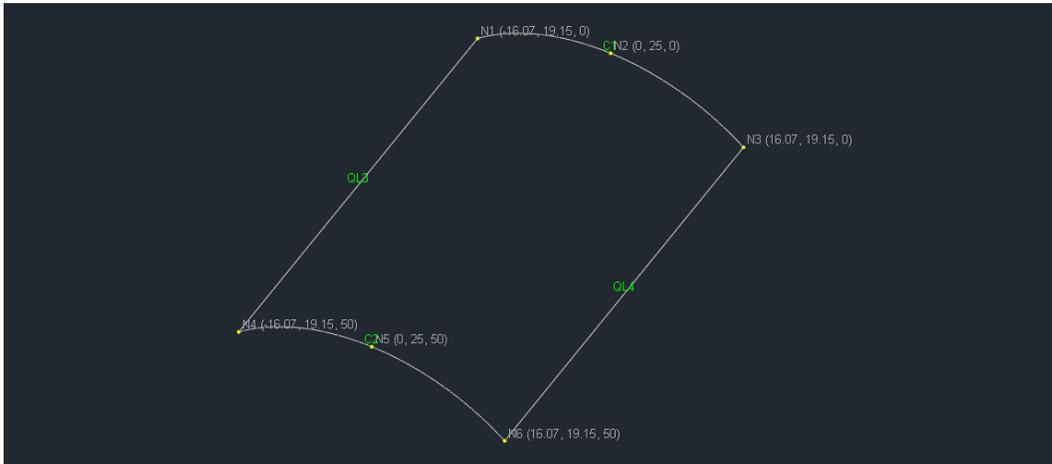


6. Now we can run Generate | Generate Mesh again. The mesh result will be displayed in a separate window. To avoid clutter, you can turn off annotations by running View | Annotate. The generated node and shell data are available from Tables | Nodes and Tables | Shell4s.



Tutorial 3 – Cylinder Region (Scodelis-Lo Roof)

In this tutorial, we are going to create the following cylinder mesh model (Scodelis-Lo roof).

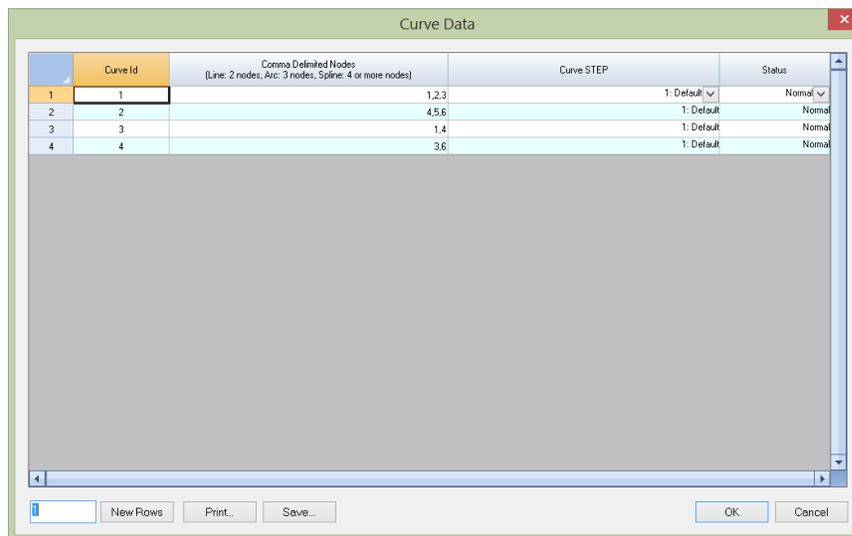


We are going to use spreadsheets to create the mesh model.

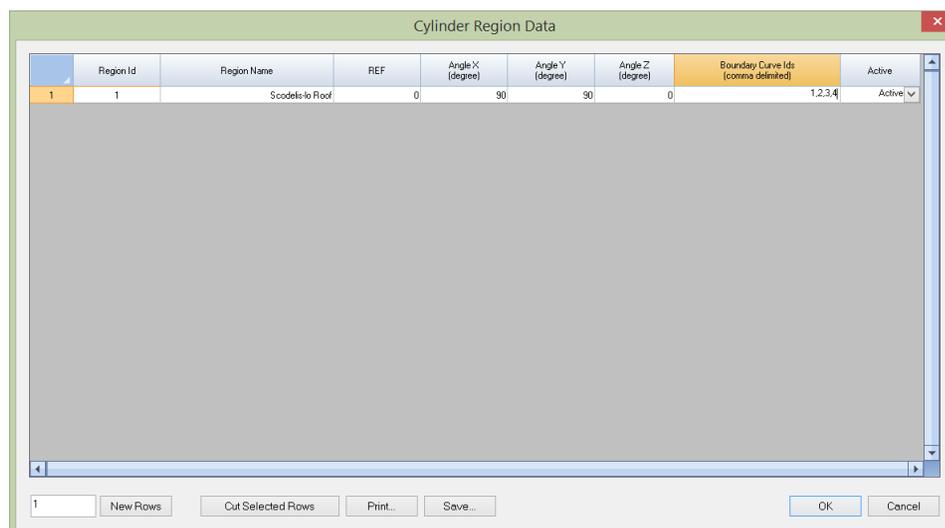
1. Go to Tables | Nodes. Generate 6 new rows by entering 6 in the bottom-left input box and clicking on “New Rows” button. Enter the nodal coordinates for each node as shown below.

	Node Id	X [ft]	Y [ft]	Z [ft]	Status
1	1	-16.07	19.15	0	Normal
2	2	0	25	0	Normal
3	3	16.07	19.15	0	Normal
4	4	-16.07	19.15	50	Normal
5	5	0	25	50	Normal
6	6	16.07	19.15	50	Normal

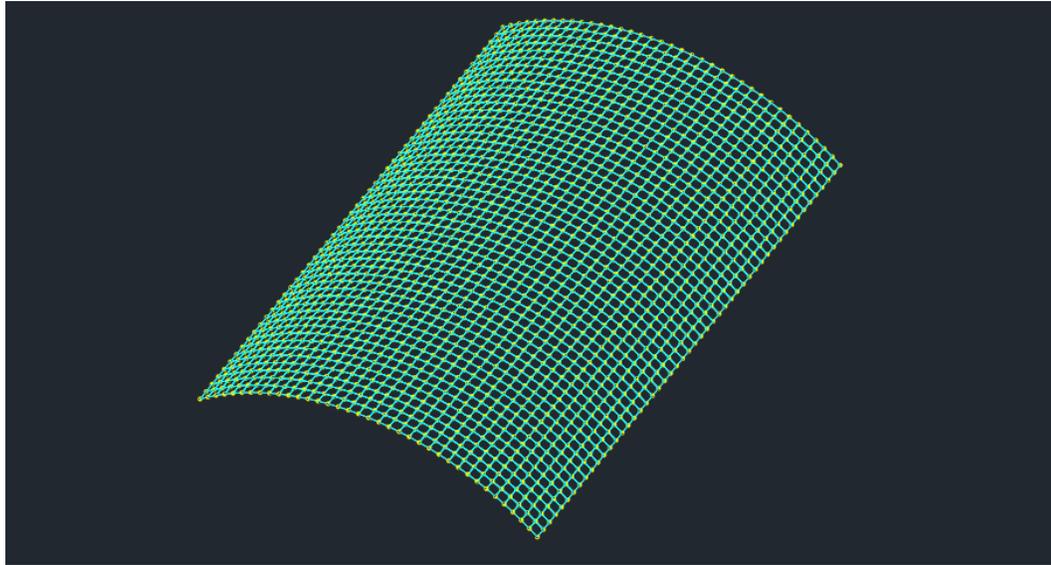
2. Go to Tables | Curves. Generate 4 new rows by entering 4 in the bottom-left input box and clicking on “New Rows” button. Enter the comma delimited node Ids for each curve. We will use the same Default curve step (1 ft) for now.



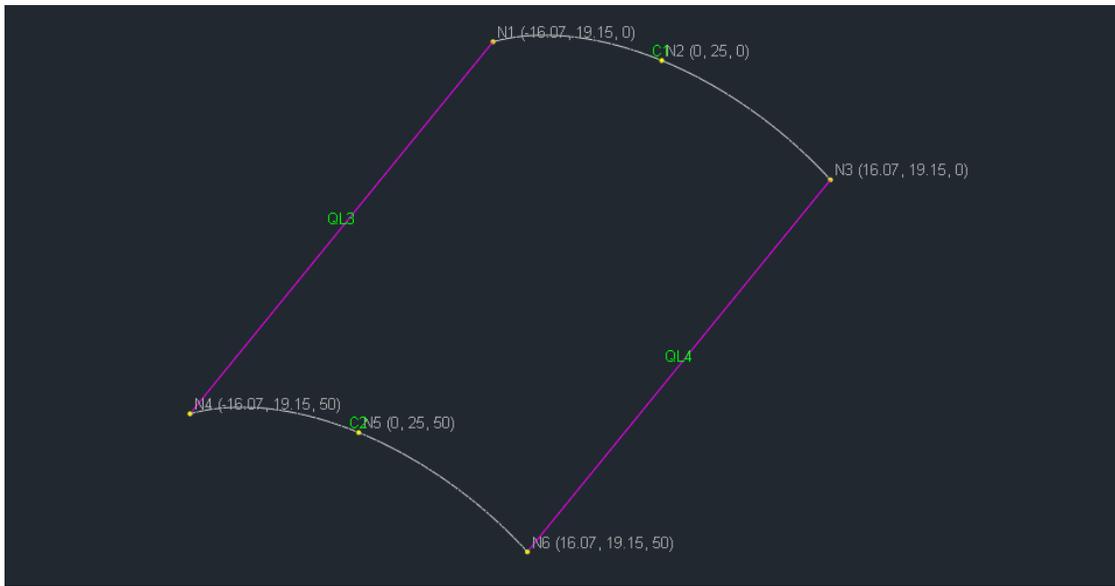
- Go to Tables | Cylinder Region. Enter the region info as shown below. Note the boundary curve Ids can be entered in arbitrary order. The solver will restore the curve order automatically. Angle X, Angle Y, and Angle Z parameters are the directional angles of the straight line that defines the surface orientation. Click on OK button. This will create the cylinder region. To display the mesh model better, we can use View | Preset Views | Isometric.



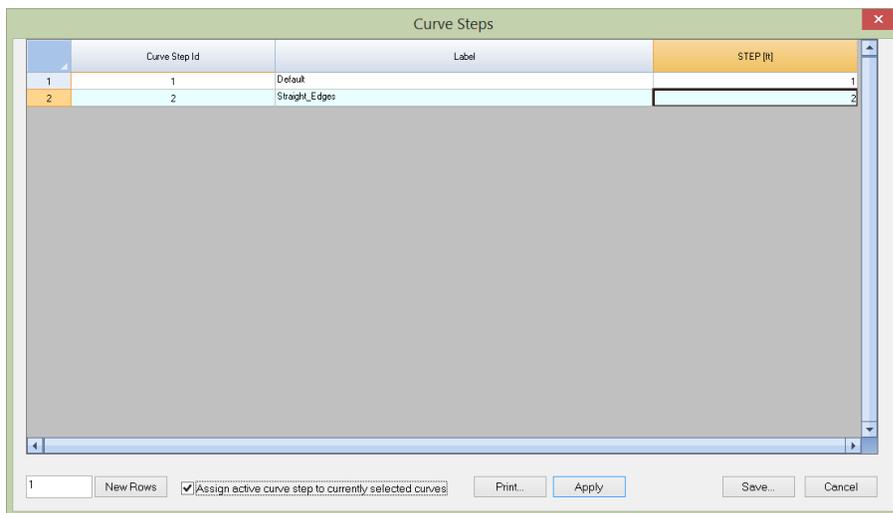
- Now we can run Generate | Generate Mesh. The mesh result will be displayed in a separate window. To avoid clutter, you can turn off annotations by running View | Annotate. The generated node and shell data are available from Tables | Nodes and Tables | Shell4s.



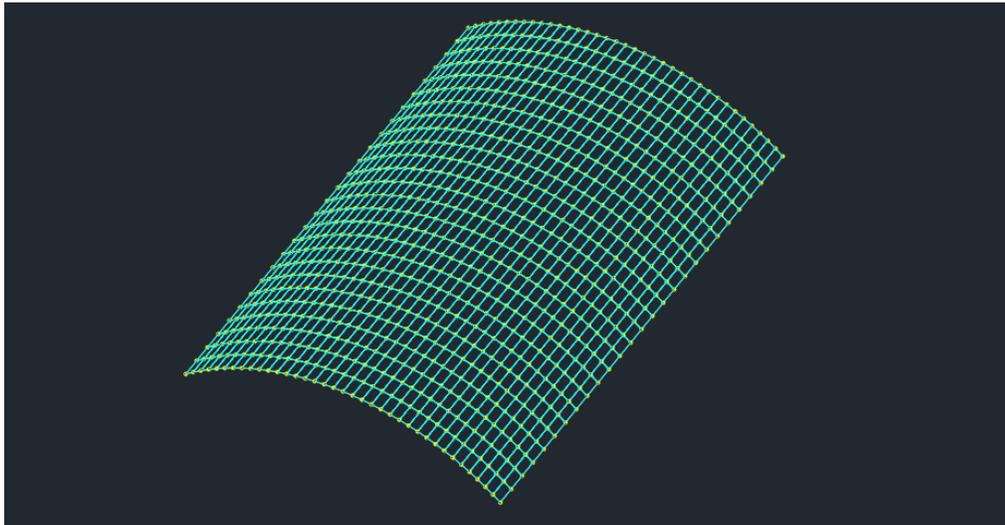
- Let us use a different curve steps for the two straight lines. First, hide the mesh model by running View | Mesh Model. Then select the two straight lines as below.



- Go to Tables | Curve Steps. Add a new curve step called “Straight_Edges” and set STEP value to be 2 ft. Make sure “Assign active step to currently selected curves” is checked. Click on Apply button. This will replace the Default curve step with Stragith_Edges curve step for the two selected straight lines.

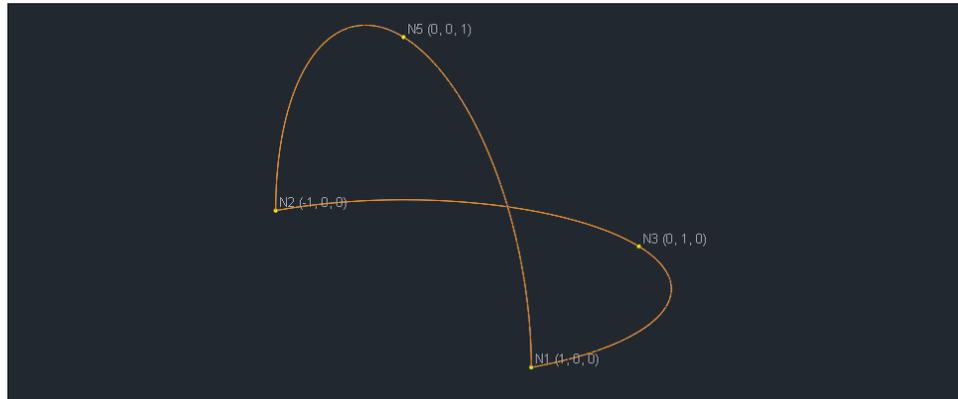


8. Now we can run Generate | Generate Mesh again. Notice the mesh size along the roof straight edges is doubled.



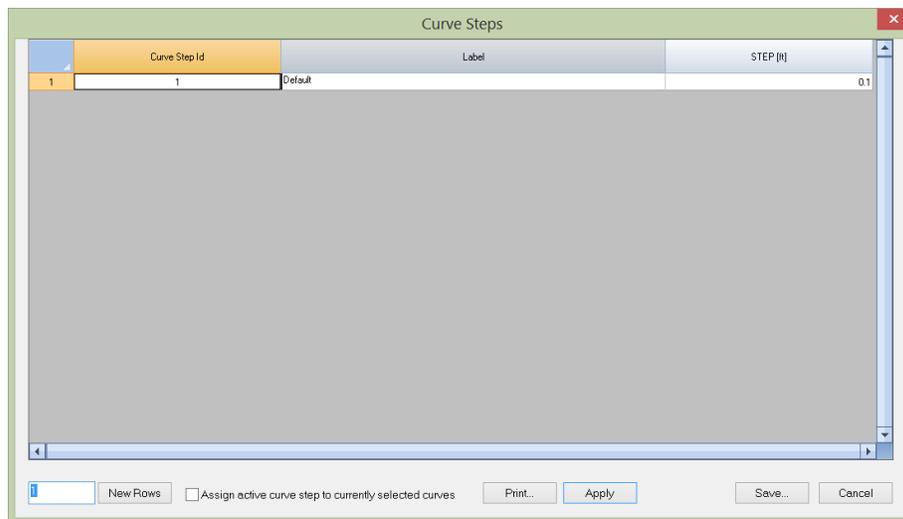
Tutorial 4 – Sphere Region

In this tutorial, we are going to create the following one quarter of a sphere mesh model.

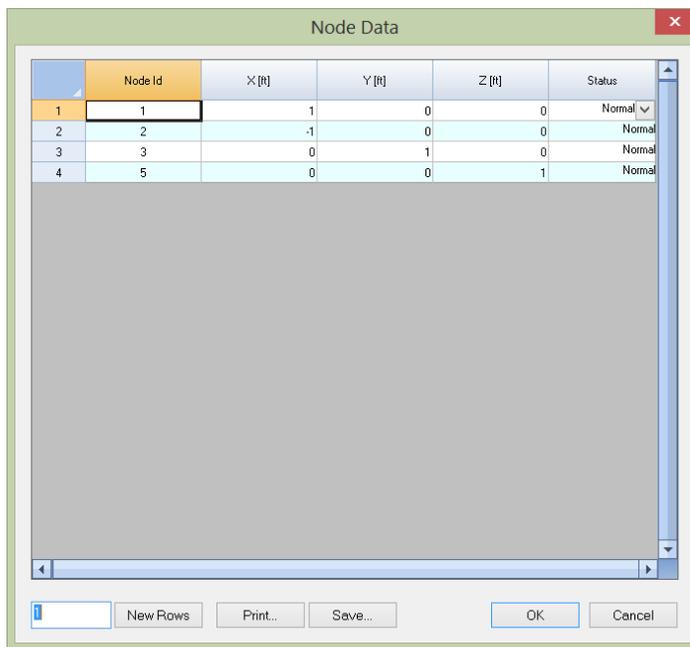


We are going to use spreadsheets to create the mesh model.

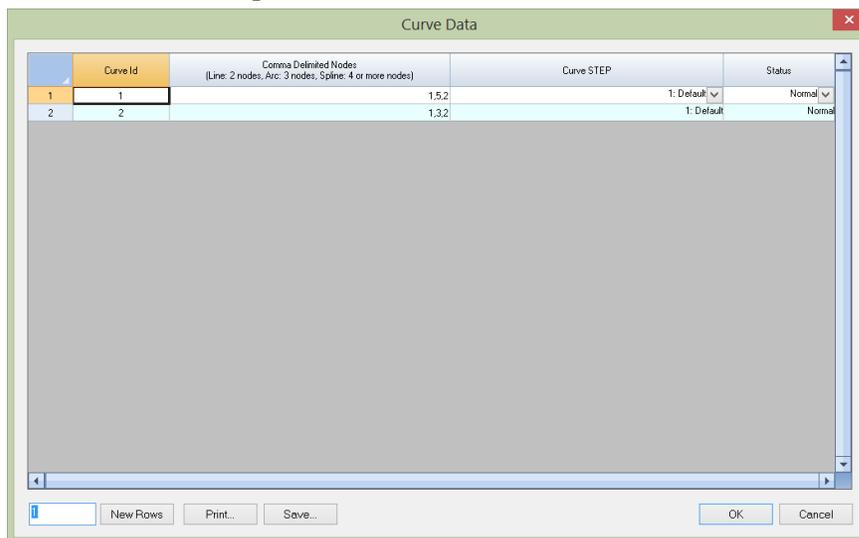
1. Go to Tables | Curve Steps. Change the Default curve step from 1.0 ft to 0.1 ft. Click on Apply button.



2. Go to Tables | Nodes. Generate 4 new rows by entering 4 in the bottom-left input box and clicking on "New Rows" button. Enter the nodal coordinates for each node as shown below.



- Go to Tables | Curves. Generate 2 new rows by entering 2 in the bottom-left input box and clicking on “New Rows” button. Enter the comma delimited node Ids for each curve (arc). We will use the Default curve step (0.1 ft).

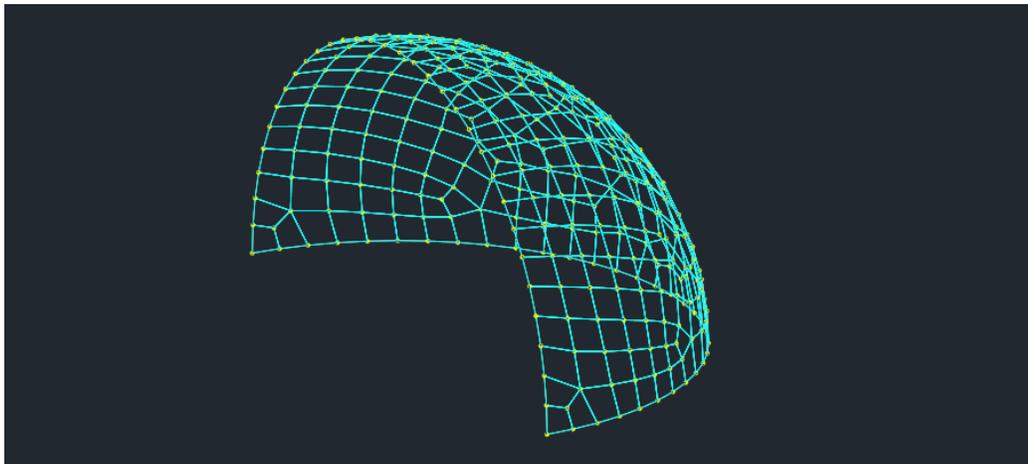


- Go to Tables | Sphere Region. Enter the region info as shown below. Note the edges and curves in each edge can be entered in arbitrary order. The solver will restore the curve order automatically. Center X, Center Y, and Center Z parameters are the coordinates the sphere center. Click on OK button. This will create the sphere region. To display the mesh model better, we can use View | Preset Views | Isometric.

Sphere Region Data

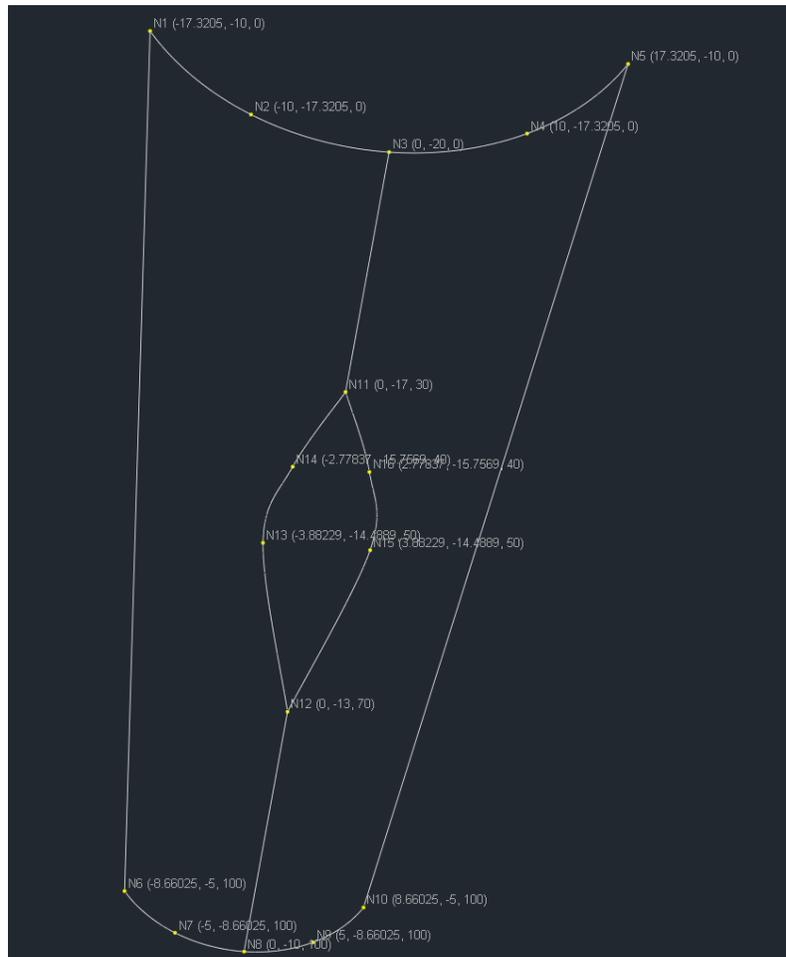
Region Id	Region Name	REF	Center X (R)	Center Y (R)	Center Z (R)	Edge1 Curve Ids (comma delimited)	Edge2 Curve Ids (comma delimited)	Edge3 Curve Ids (comma delimited)	Edge4 Curve Ids (comma delimited)	Active
1	0	REGION 0	0.000	0.000	0.000	0.000	1	2		Active

- Now we can run **Generate | Generate Mesh**. The mesh result will be displayed in a separate window. To avoid clutter, you can turn off annotations by running **View | Annotate**. The generated node and shell data are available from **Tables | Nodes** and **Tables | Shell4s**.



Tutorial 5 – Rev Region

In this tutorial, we are going to create the following mesh model that consists of two rev (of revolution) regions.



We are going to use spreadsheets to create the mesh model.

1. Go to Tables | Nodes. Generate 16 new rows by entering 16 in the bottom-left input box and clicking on “New Rows” button. Enter the nodal coordinates for each node as shown below.

Node Data ✖

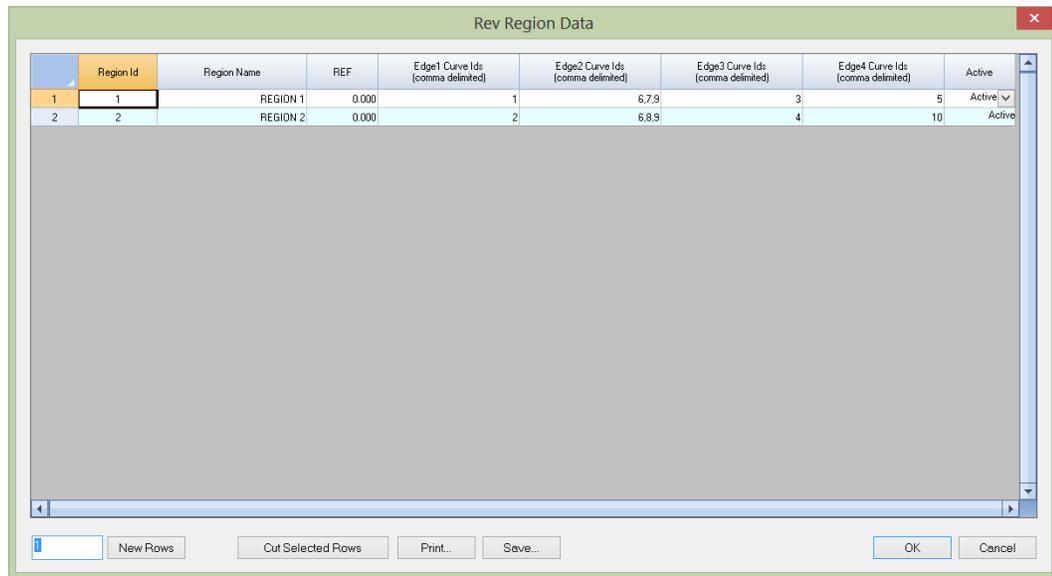
	Node Id	X [ft]	Y [ft]	Z [ft]	Status
1	1	-17.3205	-10	0	Normal
2	2	-10	-17.3205	0	Normal
3	3	0	-20	0	Normal
4	4	10	-17.3205	0	Normal
5	5	17.3205	-10	0	Normal
6	6	-8.66025	-5	100	Normal
7	7	-5	-8.66025	100	Normal
8	8	0	-10	100	Normal
9	9	5	-8.66025	100	Normal
10	10	8.66025	-5	100	Normal
11	11	0	-17	30	Normal
12	12	0	-13	70	Normal
13	13	-3.88229	-14.48889	50	Normal
14	14	-2.77837	-15.7569	40	Normal
15	15	3.88229	-14.48889	50	Normal
16	16	2.77837	-15.7569	40	Normal

- Go to Tables | Curves. Generate 10 new rows by entering 10 in the bottom-left input box and clicking on “New Rows” button. Enter the comma delimited node Ids for each curve. Notice we have used all three types of curves: 2-node line, 3-node arc, and 4-node spline. We will use the Default curve step of 1 ft.

Curve Data ✖

	Curve Id	Comma Delimited Node Ids (Line: 2 nodes, Arc: 3 nodes, Spline: 4 or more nodes)	Curve STEP	Status
1	1	1,2,3	1: Default	Normal
2	2	3,4,5	1: Default	Normal
3	3	6,7,8	1: Default	Normal
4	4	8,9,10	1: Default	Normal
5	5	1,6	1: Default	Normal
6	6	3,11	1: Default	Normal
7	7	11,14,13,12	1: Default	Normal
8	8	11,16,15,12	1: Default	Normal
9	9	8,12	1: Default	Normal
10	10	5,10	1: Default	Normal

3. Go to Tables | Rev Region. Enter the region info as shown below. Note the edges and curves in each edge can be entered in arbitrary order. The solver will restore the curve order automatically. This will create two rev regions. To display the mesh model better, we can use View | Preset Views | Isometric.



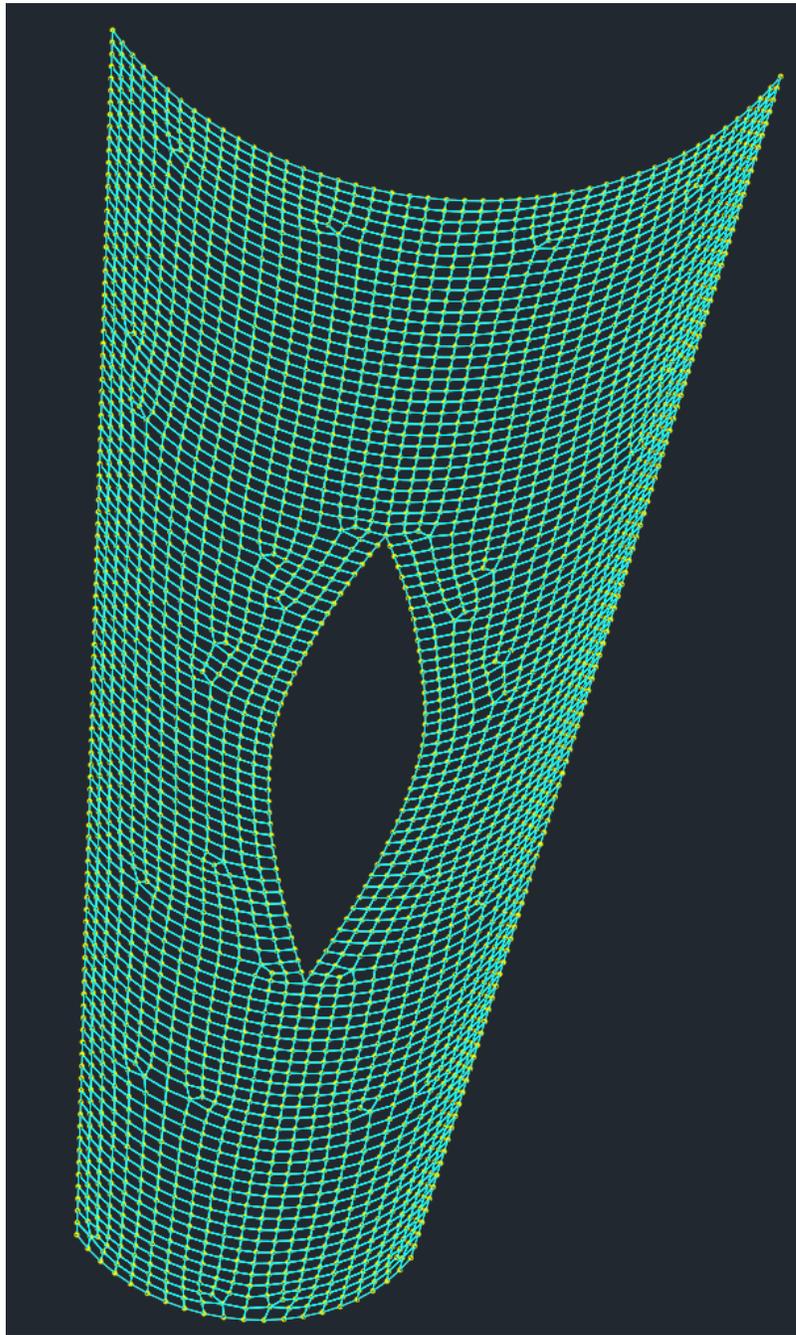
The screenshot shows a dialog box titled "Rev Region Data" with a close button (X) in the top right corner. The dialog contains a table with the following data:

	Region Id	Region Name	REF	Edge1 Curve Ids (comma delimited)	Edge2 Curve Ids (comma delimited)	Edge3 Curve Ids (comma delimited)	Edge4 Curve Ids (comma delimited)	Active
1	1	REGION 1	0.000	1	6,7,9	3	5	Active
2	2	REGION 2	0.000	2	6,8,9	4	10	Active

At the bottom of the dialog, there is a search bar, a "New Rows" button, a "Cut Selected Rows" button, a "Print..." button, a "Save..." button, an "OK" button, and a "Cancel" button.

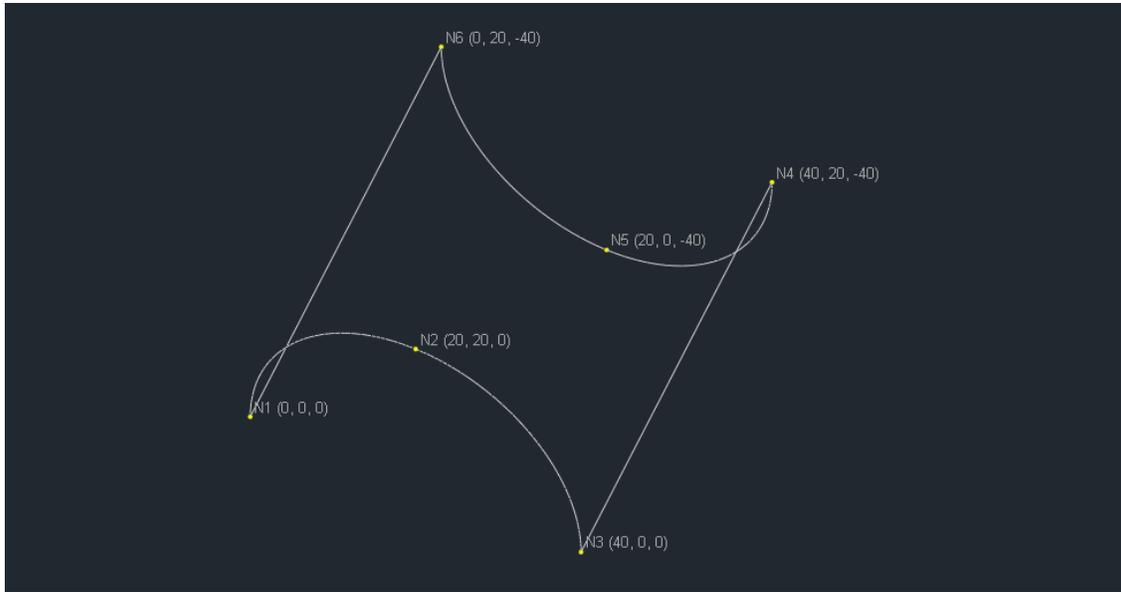


4. Now we can run Generate | Generate Mesh. The mesh result will be displayed in a separate window. To avoid clutter, you can turn off annotations by running View | Annotate. The generated node and shell data are available from Tables | Nodes and Tables | Shell4s.



Tutorial 6 – General Patch Region

In this tutorial, we are going to create the following general patch mesh model. General-type patches have to be used in situations where no other information about the patch (i.e. actual type of surface) is available except of its boundary curves.

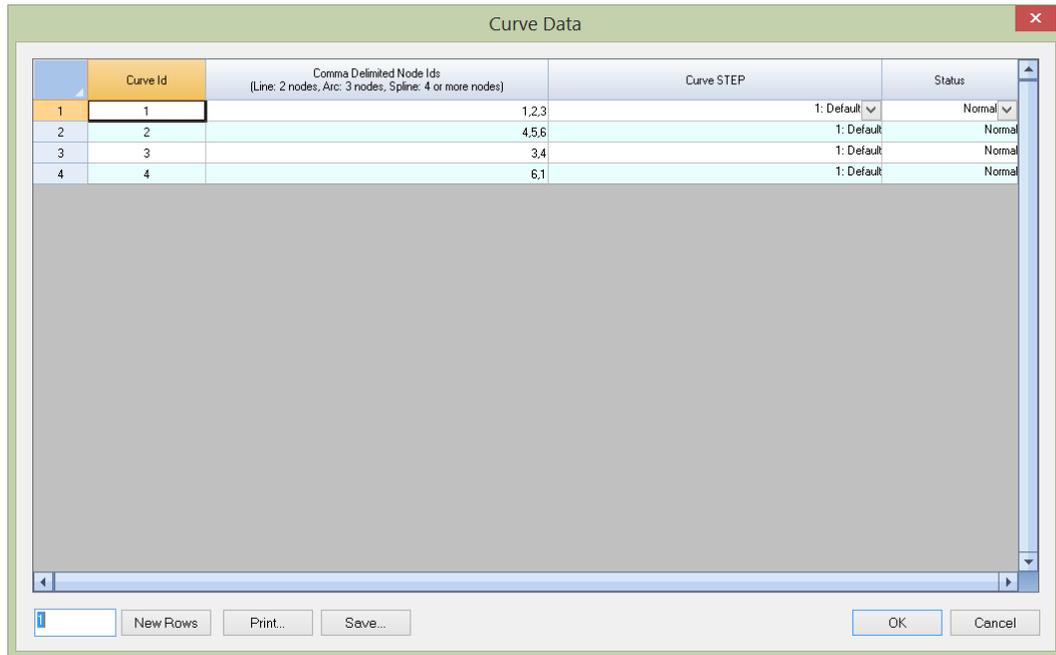


We are going to use spreadsheets to create the mesh model.

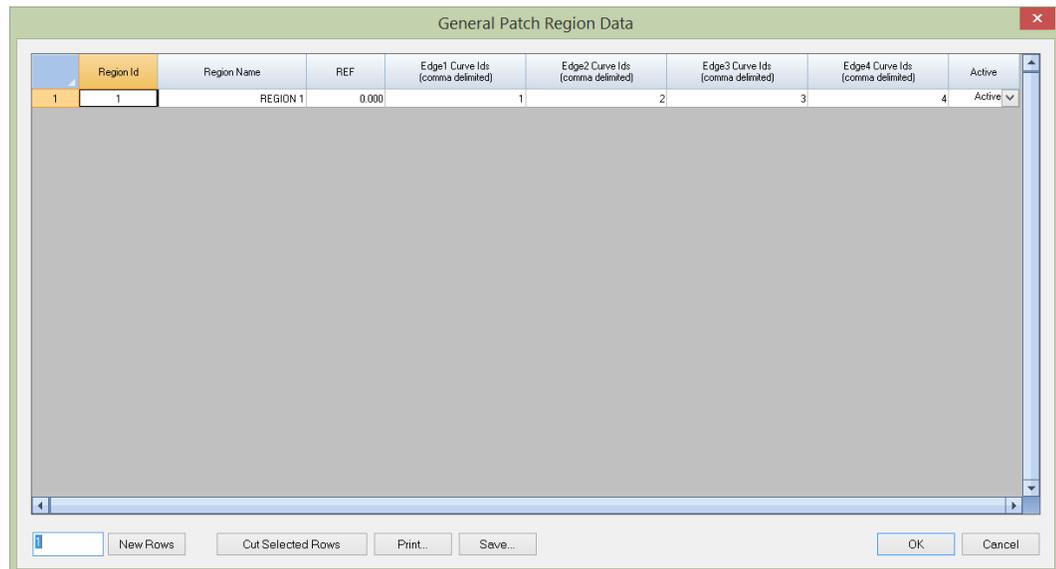
1. Go to Tables | Nodes. Generate 6 new rows by entering 6 in the bottom-left input box and clicking on “New Rows” button. Enter the nodal coordinates for each node as shown below.

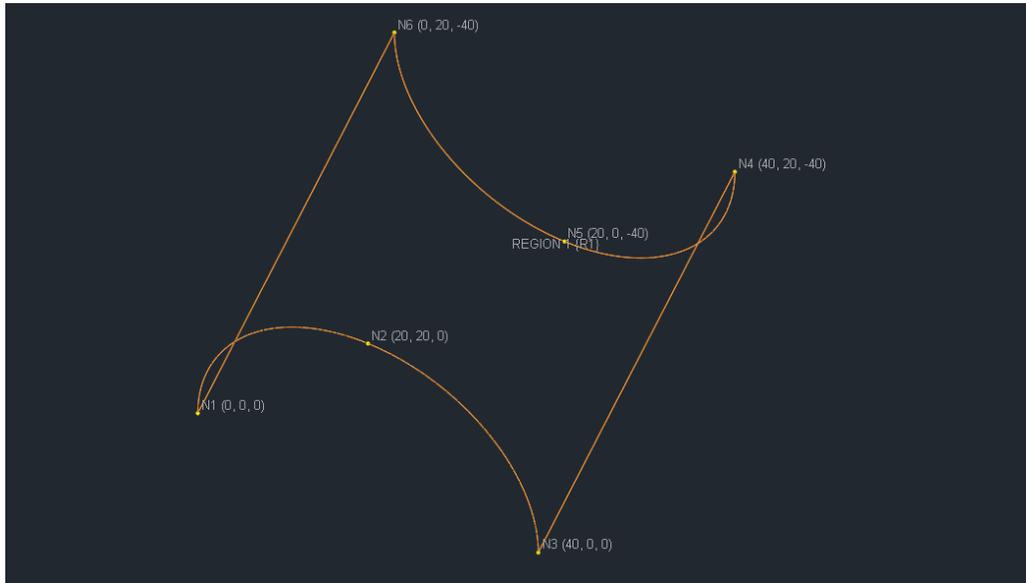
	Node Id	X [m]	Y [m]	Z [m]	Status
1	1	0	0	0	Normal
2	2	20	20	0	Normal
3	3	40	0	0	Normal
4	4	40	20	-40	Normal
5	5	20	0	-40	Normal
6	6	0	20	-40	Normal

- Go to Tables | Curves. Generate 4 new rows by entering 4 in the bottom-left input box and clicking on “New Rows” button. Enter the comma delimited node Ids for each curve. We will use the Default curve step of 1 ft.

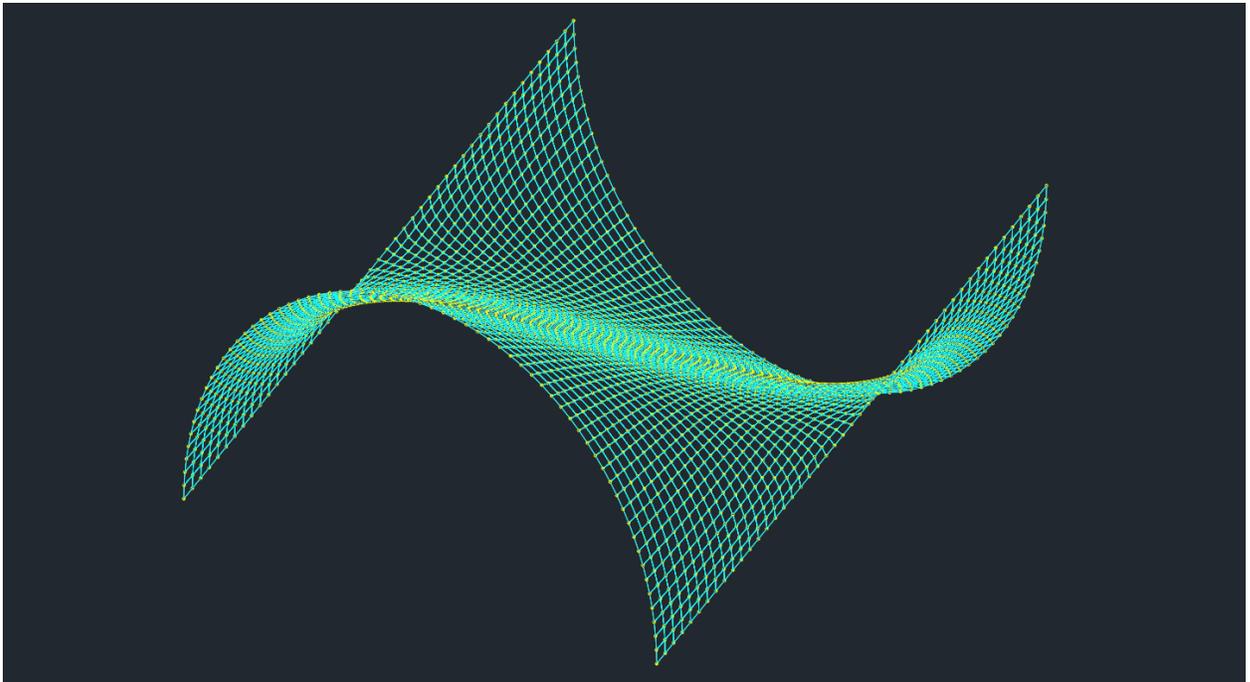


- Go to Tables | General Patch Region. Enter the region info as shown below. Note the edges and curves in each edge can be entered in arbitrary order. The solver will restore the curve order automatically. This will create the general patch region. To display the mesh model better, we can use View | Preset Views | Isometric.



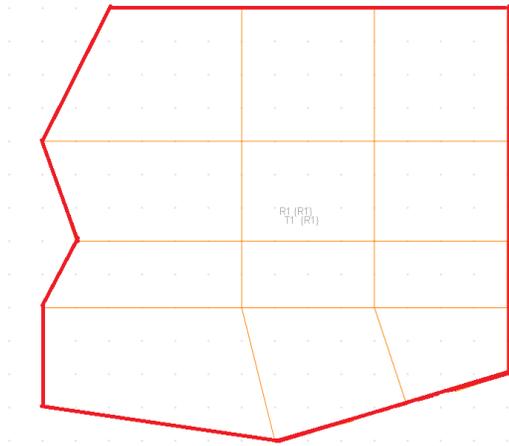


4. Now we can run Generate | Generate Mesh. The mesh result will be displayed in a separate window. To avoid clutter, you can turn off annotations by running View | Annotate. The generated node and shell data are available from Tables | Nodes and Tables | Shell4s.

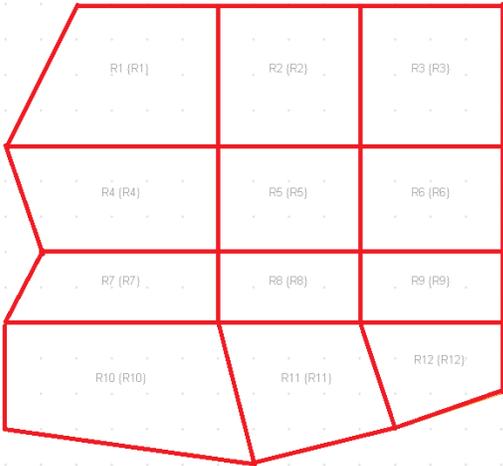
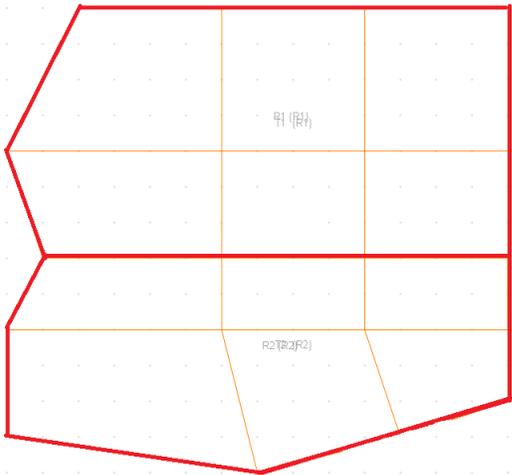


Tutorial 7 – Floor with Grid of Beams

A floor with multiple intersecting curves is shown below. If we were to define one region (boundary lines colored red) and a bunch of internal lines or tree(s) (lines colored orange), then the mesh model would be invalid. This is because that the tree segments would form two internal loops which are not permitted.



To properly create a mesh model, we can use either two regions (boundary lines colored red) with corresponding internal lines or trees (lines colored orange) as show on the left image below, or a separate region for each closed loop with a total of twelve regions as shown on the right image below.



References

1. Computational Mechanics Australia Pty.Ltd., “Quad_Surface 4.0 User Manual”, 2010
2. Computations & Graphics, Inc., “Real3D Manual”, 2020
3. K.J. Bathe, “Finite Element Procedures” Prentice-Hall, Inc., 1996