CASCA: A Simple 2-D Mesh Generator

Version 1.4 User's Guide
May, 1997

Manual and Code Modifications by:
Daniel Swenson, Mark James, and Brian Hardeman
Kansas State University • Manhattan, Kansas

CASCA originally written by:
Paul Wawrzynek and Louis Martha
Cornell University • Ithaca, New York

Table of Contents

Introduction

CASCA is an interactive program for the creation of two-dimensional continuum finite element meshes. Its capabilities include three- and six-noded triangles and four and eight-noded quadrilaterals.

This manual is intended to be a simple description of the basic features of CASCA. It includes a brief tutorial, hints on addressing common problems, and menu descriptions.

Within the CASCA program, all user commands are made by clicking the mouse on one of the options displayed in the *menu window* (Figure 1). A *message window* is always present to prompt the user on the next step in the requested procedure. Entry into CASCA and I/O operations invoked during the running of CASCA are made from the *program control window*. This is the window from which the program was started. An *auxiliary window* is used for attribute definition.
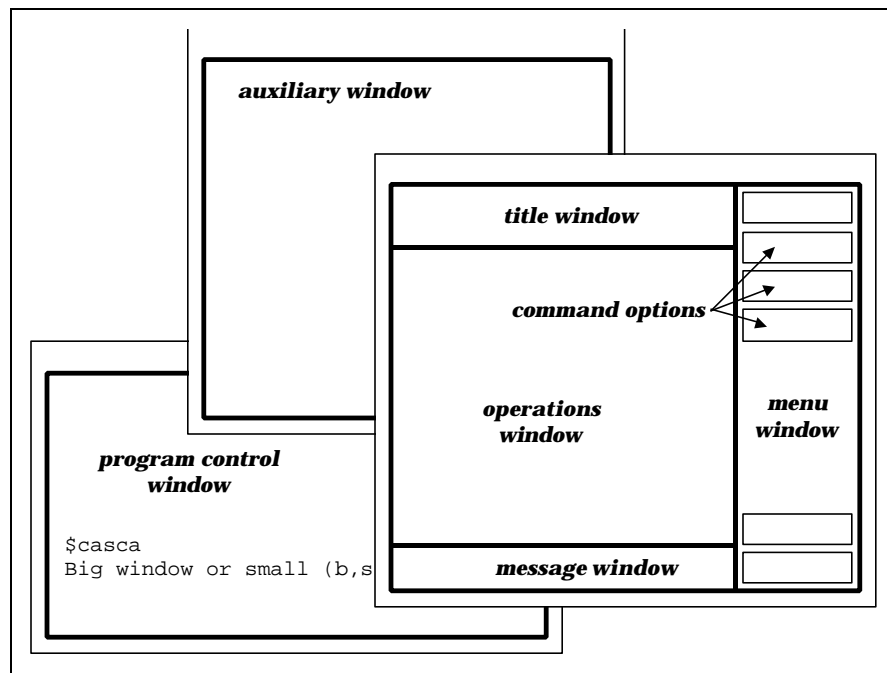
Figure 1: The CASCA window system

The CASCA program uses two types of cursors. The normal cursor has the shape of an arrow. When you see this cursor it means that the program is waiting for you to select a menu option or some other graphical input. The second cursor is a stylized wristwatch. When you see this cursor it means that either the program is processing data (e.g., generating a mesh), or it is waiting for input in the program control window.

The coordinate system used within the program is always fixed so that the x coordinate is horizontal, increasing to the right and the y coordinate is vertical, increasing going up.

CASCA was originally developed by Paul Wawrzynek and Louis Martha at Cornell University as a test bed for a hierarchical data base that was later implemented in FRANC3D. However, it has proven to be a useful tool, so it has continued to live beyond its initial application.

The fact that CASCA was originally written in 1987 is reflected in the menu structure. The menus were designed for display using low level XWindows commands. A rewrite of these low level commands has allowed us to port the code to Windows 95/NT in a simple fashion. The advantage is that the Unix and Windows 95/NT versions are identical; the disadvantage is that the program does not use dialog boxes or pull-down menus. We hope you will find the interface adequate, if not ideal.

CASCA Files

Two types of files are generated or used by the CASCA program. The contents of these files and their uses are discussed here. The *'s in the text are to be replaced by file names chosen by the analyst.

CASCA *.csc Files

The *.csc files are restart files generated by CASCA. A restart file allows one to save current work and recover it later. This is convenient when a mesh description cannot be completed at one sitting or to make modifications to an existing mesh. A *.csc file is created when the **Write** option (not **Write Mesh**) is selected in CASCA.

VIEW *.vw Files

The *.vw files are used to store views. A *.vw file is created when the **SAVE VIEW** menu is selected under **PAN**. **RESTORE VIEW** asks for a *.vw file name. The user gives the *.vw file name (no extensions) in the *program control window*.

Input *.inp Files

The *.inp files contain the mesh data used for input to finite element programs. These are human readable ASCII files that describe a mesh using nodes and elements in a format similar to those used by most other finite element programs. The *.inp file is written by CASCA using the **Write Mesh** option. The file format is given below.

```
**********************************************************************
************************  INPUT FILE FORMAT  ************************
**********************************************************************


======================================================================

                 Card Set 1:  Title card
                 Number of cards in set: 1

    Problem_title(Char*40) - Title of problem, 40 chars

======================================================================

                 Card Set 2:  Control card
                 Number of cards in set: 1

    Num_Nodes (I*4)
    Num_Elem (I*4)
    Num_Mat (I*4)    - Number of materials
    Prob_Type (I*4)  - Analysis type
                              0 Axisymmetric
                              1 Plane Stress
                              2 Plane Strain
                              3 Linear Bending

======================================================================
```

```
                     Card Set 3:  Material Properties
                     Number of cards in set: Num_Mat (See Card Set 2)

     FORMAT(I5, 14F10.2)

     Mat_Type(I*4)   - The material type
                         1 Linear elastic isotropic
                         2 Linear elastic orthotropic

     If Mat_Type = 1
         Young's modulus (R*8)
         Poisson's Ratio (R*8)
         Thickness (R*8)
         Fracture Toughness KIc (R*8)
         Density (R*8)

     If Mat_Type = 2
         Young's modulus in the 1 direction (R*8)
         Young's modulus in the 2 direction (R*8)
         Young's modulus in the 3 direction (R*8)
         Modulus of rigidity in the 12 direction (R*8)
         Poisson's ratio in the 12 direction (R*8)
         Poisson's ratio in the 13 direction (R*8)
         Poisson's ratio in the 23 direction (R*8)
         Rotation angle beta (R*8)
         Thickness (R*8)
         Fracture Toughness KIc in the 1 direction (R*8)
         Fracture Toughness KIc in the 1 direction (R*8)
         Density (R*8)

======================================================================

                     Card Set 4:  Connectivity
                     Number of cards in set: Num_Elem
     FORMAT(10I5)

     Elem_Num(I*4)        - Element number
     Material(I*4)        - Material number for element
     Elem_Nodes(1)(I*4)   - First node number
            .
            .
     Elem_Nodes(8)(I*4)   - Eighth node number

     Note:  Node numbers should be specified in a counter clockwise
     direction, starting at any corner node.  If Elem_Nodes has eight
     non-zero elements a Q8 is assumed, if 6 non-zero elements a T6 is
     assumed.

======================================================================

                     Card Set 5:  Nodal Coordinates
                     Number of cards in set: Num_Nodes

     Node_Number(I*4)  - Node number
     X_Coord(R*4)      - X coordinate of node
     Y_Coord(R*4)      - Y coordinate of node

======================================================================
```

CASCA Examples

In this portion of the manual, the use of the CASCA program is illustrated. The steps necessary to create a mesh are described. It is intended that you repeat the steps as they are described. The example consists of a plate with a hole, as shown in Figure 2.
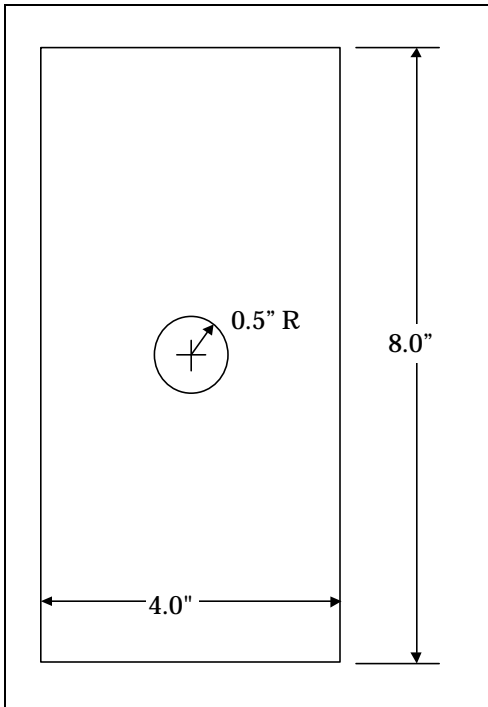

Figure 2: Schematic of plate with hole

We will generate meshes for this problem. The tutorials use only a subset of the options and features available in CASCA. However, the examples should give you the confidence to try other features, which are described in the menu reference section.

In the tutorial, menu options are indicated by bold text, such as **Data Size**. Text that you enter in the program control window are indicated with a typewriter font, such as `tutorial.csc`.

Plate with Hole Using Symmetry

Running the CASCA Program

Begin by running the CASCA program.

> **Windows 95/NT:** Start an MS-DOS window (select the menus Start\Programs\MS-DOS Prompt). In that window, go to the directory in which you want to save your analysis (for example cd "C:\My Documents\Example1"). Then type the path to the CASCA executable (for example "C:\My Documents\Casca\casca.exe"). You will be asked whether you want a big or small window, type s to select small. At this point a new CASCA window will be created.

> **Unix:** Select the window in which you will start CASCA (typically an xterm window started using the command xterm&). In that window, go to the directory in which you want to save your analysis (for example "cd ~/Example1"). Then type the path to the CASCA executable (for example "~/Casca/casca.exe"). You will be asked whether you want a big or small window, type s to select small. At this point a new CASCA window will be created.

Please note that the directory in which the executable is saved can be placed in the search path, so that CASCA can be run from any directory by typing only `casca`. Aliases and paths in Unix can be used the same way.

Set Scale: Setting an Appropriate Data Space

Initially you will have only three options: setting the data space (**Set Scale**), reading a restart file (**Read**), and adjusting your view (**RESET**, **MAGNIFY**, **ZOOM**, **PAN**, and **SNAP**). Because we are starting a new problem from scratch, we must select **Set Scale**.

In the **Set Scale** page you can change the world coordinates in the operations (mesh) window. By default, the window is 12 units wide by 12 units high with a grid spacing of one unit and the center of the grid at X=0.0, Y=0.0. This is satisfactory for our problem, so just select **RETURN**.

Geometry: Creating the Problem Outline

Grid lines are available to simplify geometry construction. Select **Grid** from the menu. The X in the middle of the screen marks the center of the grid (presently X=0.0, Y=0.0). When the grid is displayed, a selection near a grid point will snap-to the grid. In addition to grid snap-to, snap-to is always active to the ends of lines that the user has defined.

From the main menu, you should see a **Geometry** option. Geometry is constructed by connecting edges into closed faces. Presently, only lines, circles, and arcs are available. These allow you to specify the outline of your problem, which is the geometry used when generating a mesh. Select **Geometry**.
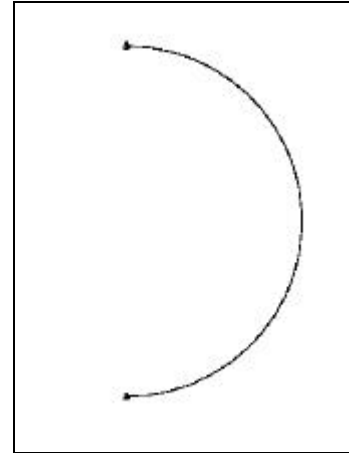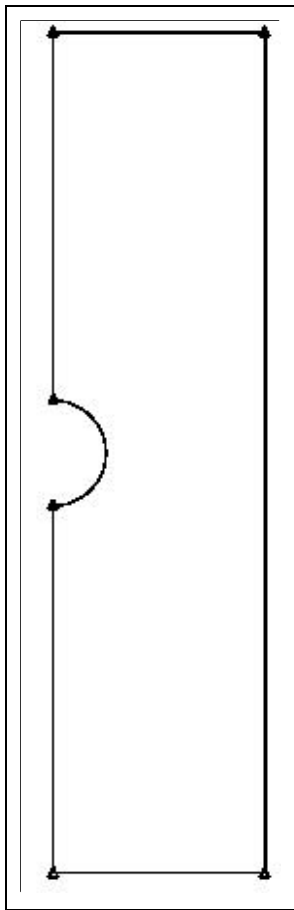
You are now presented with a number of options that you can

Figure 3: Circle arc

use to specify the outline of your object. For the plate with hole problem, we will begin with the hole. To take advantage of symmetry, we will mesh only the right half of the problem. First select **Get Circle**. By default, the **ARC** option is active. An arc is specified in the counter-clockwise direction by three points: beginning, ending, and the center. Our grid spacing is one unit, but want to specify the arc beginning at X=0.0, Y=0.5. Since we can not just click on the grid, we will use the **KEYPAD**. Select **KEYPAD** to display a numeric pad, then use the mouse to input 0.0 for the X coordinate, followed by **ENT** for enter, then input 0.5 for Y, and **ENT** again. The beginning point for the arc will now be displayed with a square. To enter the ending point for the arc, again select **KEYPAD** and enter the coordinates as X=0.0 and Y=-0.5. To complete the arc specification we need to indicate the center of the arc. We can do this using the grid since the center point lies on the grid. Just point to the center of the grid and click with the mouse. The arc will be displayed. Select **DONE** to indicate that you are finished defining the arc. If you select **QUIT**, the circle will be ignored. The display should be as shown in Figure 3.

To complete the plate outline we will use the **Lines Connected** option. Select this option. To start the connected lines, click on the top point of the circle arc. Then move up to the grid location corresponding to X=0.0, Y=4.0, click, move to the right 2 grid intersections to X=2.0, Y=4.0, click, down 8 grid intersections to X=2.0, Y=-4.0, click, to the left 2 grid intersections to X=0.0, Y=-4.0, click, and, finally, to the lower point of the circle arc and

Figure 4: Completed outline

click. To leave this mode of adding line segments, select **DONE**. The display should be like Figure 4. This completes the border definition. **RETURN** to the main page.

Subregions: Dividing the Geometry

From the main menu, you should now notice more options are available. The next one we will use is **Subregions**. This allows you to break your object up into a number of simpler regions that are more convenient for meshing. In the plate problem, we will divide the plate and hole into four separate regions for meshing. Select **Subregions** and you will see a number of options that are similar to those available on the geometry page.

Select the **Get Line** option, and specify a line from the right of the hole to the border on the right of the plate. Use the **Keypad** to specify the first point at X=0.5 and Y=0, then just click on the grid at point X=2.0 and Y=0.0. Select **DONE** (not **QUIT**) to accept this line. Repeat, just clicking on the grid points, to add lines above and below the hole at Y=2.0 and Y=-2.0. You now have divided the patched plate into four regions. The problem should look like Figure 5.

This is all the division that is necessary, you should now **RETURN** to the main menu. Note that when we added the new lines, we actually split the existing lines defining the geometry.

Subdivide: Specifying Nodal Points on the Edges

From the main menu, select **Subdivide.** In the subdivision page, one specifies nodal densities along the boundaries for all the regions in the structure. The arrow on each edge indicates its orientation, and is used for a varying density along the edges.

We will start with the two arcs that now define the hole. We will define 10 subdivisions on each quarter circle arc. To do this, select **Num Segments** and enter 10. Now click on both arcs defining the circle. You should see triangles to indicate the nodal densities. Also click on the three radial lines extending from the circle. Next select **Num Segments** and enter 4. Define this nodal density for the two horizontal segments on the top and the two segments on the bottom of the plate. To define the two segments on the right edge away from the circle,

Figure 5: Subregions added

select **Num Segments** and enter 6. We also want a finer mesh near the X axis, so select **Ratio** and enter 1 and 2 to define a 1:2 ratio. Click on the lower right line segment. Next, since the arrow of the upper right line segment is towards the X axis, select **Revert Ratio** and click on that line segment. Finally, return the ratio to 1:1, specify 5 divisions, and
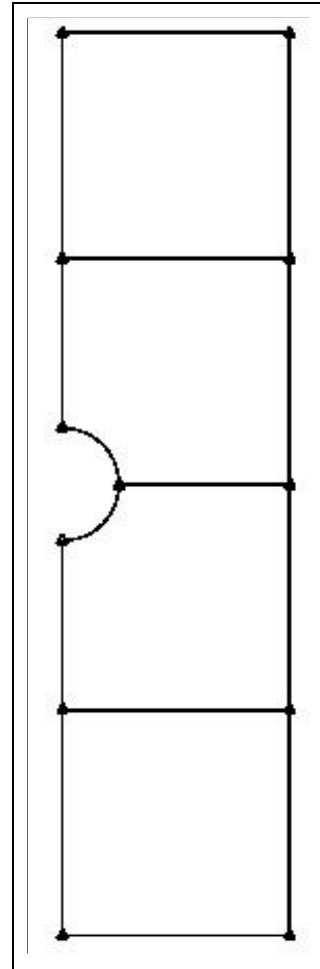
select **All Remaining** to finish the subdivision of all lines. The plate should now look like Figure 6.
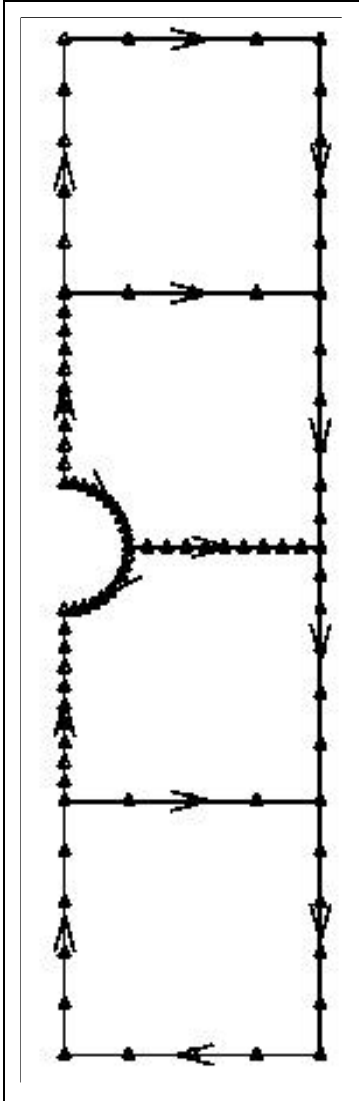
Figure 6: After line subdivision

As illustrated, the **Ratio** option can be used to specify a mesh with a density that varies along a line. For instance, selecting **Ratio** and entering 1 and 2 means the mesh size will vary a factor of two in the direction of the arrow defining the line segment. The **Revert Ratio** option can be used to change the arrow direction.

Mesh: Mesh Generation for Plate

Return to the main page. The next step is to generate meshes for the four regions. Select the **Mesh** option to move to the mesh page. The first two options on this page allow you to select element types. The defaults are Q8 quadrilateral elements, and T6 triangular elements. You must use these second order elements with FRANC2D/L.

All four regions of the patched plate can be meshed with the bilinear four sided meshing algorithm (**Bilinear 4 Side**). This algorithm requires a rectangular region with equal numbers of nodes on opposing sides.

The two regions adjacent to the hole have five sides. However, if we think of the arc on the circle as one side, the radial lines as each a side, and the opposing top and right box edges as one logical side, we have a four sided region with equal nodes on opposing sides. We mesh this by selecting the **Bilinear 4 Side** option and clicking in the region. A mesh is generated. If the program is not able to determine the four corners of the region, it prompts you to specify these points. Repeat selecting the **Bilinear 4 Side** option and clicking on the rest of the regions. The mesh is shown in Figure 7.
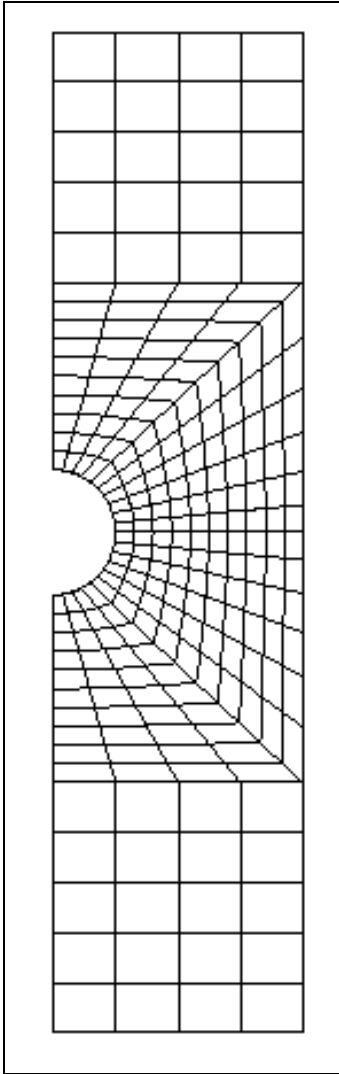
Figure 7: Final Mesh

(Note that it is not necessary to have regions that can be meshed using the **Bilinear 4side** option. For example, the **Transition** mesh option allows you to mesh arbitrary regions with arbitrary subdivision on the sides. This option can generate interior nodes or use only the boundary nodes.)

Meshing of the plate is now complete, you should **RETURN** to the main page. Create a CASCA restart file by means of the **Write** option. After you select **Write**, you must bring the *program control window* to the front and type in a file name (no extensions). Give a name such as `plate`, and a plate.csc file will be written.   A *.inp file can also be created for **FRANC2D/L** by selecting the **Write Mesh** option. Again specify the name `plate`, and a plate.inp file will be created.

Select **END** and **CONFIRM EXIT** to leave CASCA.

Example with Complete Hole and Different Meshing Algorithms

This example shows how to define a hole in a body. It also demonstrates some of the meshing options available in CASCA. We mesh the same plate with a hole used in the previous example (Figure 2). The steps are again given below.

Run CASCA

Start the CASCA program.

Set Scale

From the main page, select the **Set Scale** menu. This establishes the range of the coordinate system for the problem. We will use the default scale, so just select **RETURN**.

Geometry

Select **Grid** to display the grid. Next, select **Geometry** from the main menu. First use **Lines Connected** to generate the rectangular plate (this could also be done using multiple **Get Line** definitions). To define the hole, select **Get Circle**, click on **whole/ARC** to define a whole circle (the active option is given in capital letters). We must now define a point on the circle. Often the user could just click on a grid point, but since we want a hole with a radius of 0.5, we will use the keypad. Select **KEYPAD** and specify X=0.5, Y=0.0. Now define the center of the hole by clicking on the center grid point. Finally, select **DONE** to indicate the definition is complete. At this time, a circle has been defined, but it is still necessary to define this as a hole, not just a circular region. To define the hole, select **Specify Hole** and click inside the hole.

The **Attributes** menu allows the user to define the analysis type and material properties. Material properties are always attached to geometry faces. The mesh that will eventually be created on each face will inherit the material property attributes.

RETURN to the main menu.

SubRegions:

Select **SubRegions** from the main menu. Use Figure 8 as a reference to generate the subregions. A good way to do this is to first create the three radial lines extending from the hole (use **Get Line** and **KEYPAD** to give the starting points on the circle, then click on a grid point for the ending points). Create the horizontal line in the upper half using two **Get Line** commands, clicking on grid points to define the lines (a two step definition is needed because CASCA does not allow the insertion of crossing lines). Finally, complete the diagonal lines and lower horizontal line.

RETURN to the main menu.
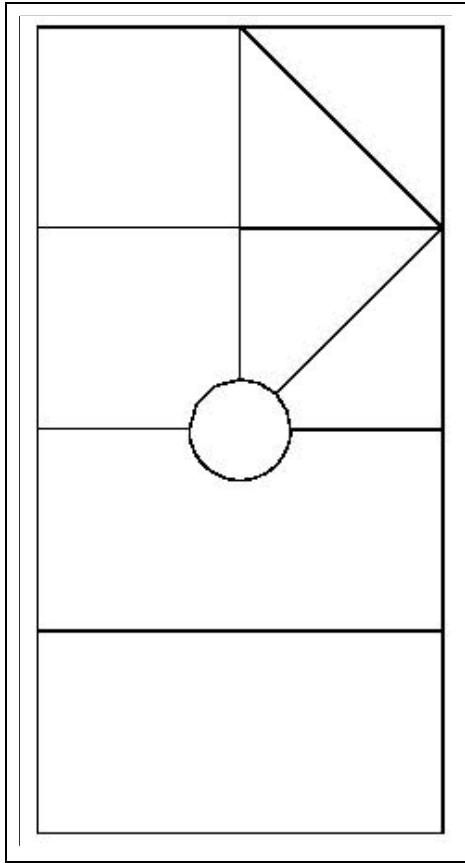
Subdivide the Edges

Figure 8: Plate subdivisions

Select the **Subdivide** menu option. We will first work only with the edges in the upper right of the figure. Use Figure 9 as a reference. The edges are subdivided into either 4 or 5 segments. In all cases, the ratio is 1:1.
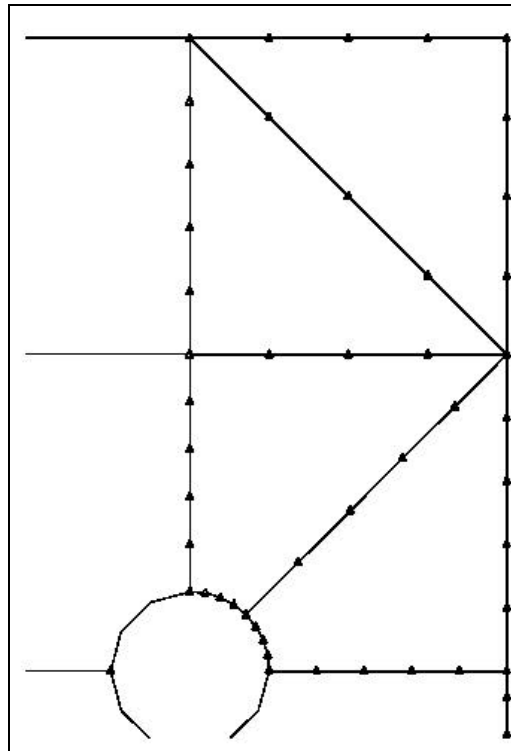
RETURN to the main menu.

Figure 9: Subdivided edges

Mesh the Regions

Select the **Mesh** menu option. Select the **Automatic** menu option. If the edge subdivisions matches those in Figure 9, the mesh 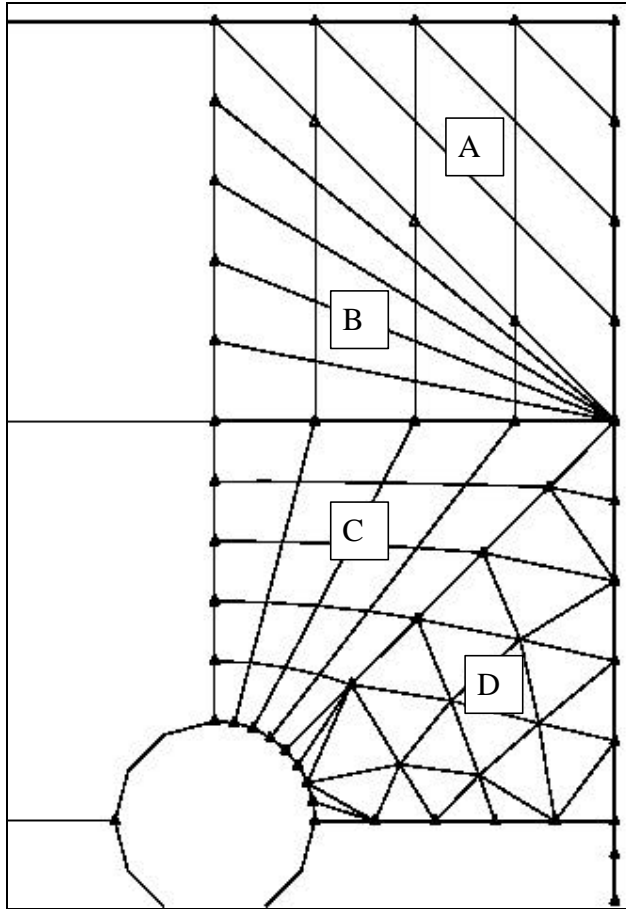should look like Figure 10. The automatic meshing algorithm attempts to use the most appropriate meshing scheme for whatever subdivisions have been specified on each face. In this case, the algorithm has used:

Figure 10: Meshed regions

- Triangular map for region A. This is used when there are the same number of subdivisions on each edge of a triangular face.
- Bilinear 3 side for region B. This is used when at least two sides of a triangular face have the same number of subdivisions.
- Bilinear 4 side for region C. Used when opposite edges of a quadrilateral face have matching subdivisions.
- Transition for region D. Transition can mesh regions with arbitrary number of edges and arbitrary subdivisions on each edge.

You can verify that these are in fact the algorithms that have been used by deleting (using **Delete**) each region and then manually selecting the meshing algorithm for each region.

This should give you identical meshes.

Exploring other Meshing Possibilities

Other algorithms on Region A:

Select **W/O Corner Pts** (it changes to **Prompt Corners**) to force CASCA to ask for a user definition of corners. New remesh region A several times using **Bilinear 3 Side**, selecting corner points in different orders. Note the effect on the resulting mesh.

Other algorithms on Region B:

Try the **Triangular Map** algorithm on region B (it won't work because the number of subdivisions is not equal on all three edges of the region). Make sure you delete the mesh in region B. Now try **Transition** meshing with **Generate Int Pt** (generate interior points).

This will create a mesh using triangular elements. Leave the mesh and select **Transition** again. Again select **Generate Int Pt**, but now click at several locations inside region B then select **DONE**. Each point you clicked on now becomes a node. This allows you to refine meshes in desired areas.

Other algorithms on Region D:

Return to the **Subdivide** menu and change the number of subdivisions on region D so that opposite edges of the quadrilateral have an equal number of subdivisions. Now remesh region D using **Bilinear 4 Side**.

Note that when the subdivisions on an edge change, the mesh that depends on the subdivisions will be deleted. This hierarchical dependence is also true for other entities. For instance, changing the geometry of the problem. Before you quit, try deleting a geometry or subregion edge and check the effect on the subdivisions and the mesh.

Using Q8 and T6 as the Default Element Type:

Mesh region C using **Bilinear 4 Side**. Now change the current element type to T6 (six-noded triangle) by selecting the **T6** menu option. Mesh region C again using **Bilinear 4 Side**. Experiment with meshing region C using the **Right**, **Left**, **UnJck**, and **Optm** menu options (these only apply to regions meshed with T3 or T6 element types).

Note, FRANC2D/L does not accept four-noded quadrilateral (Q4) or three-noded triangular (T3) elements. Use only Q8 or T6 elements. GEOCRACK accepts only T6 elements.

Notes on Using Bilinear 4 Side:

The **Bilinear 4 Side** meshing algorithm can be useful even for regions that have more than four edges. What is important is that there be a mapping of the region into a quadrilateral with equal subdivisions on opposite edges. For instance, go to the **Subdivide** menu and change the subdivisions on the edges of region E as shown in Figure 11. Use 10 subdivisions on the arc, 5 each on the opposite edges, and 5

Figure 11: Bilinear 4 Side Subdivisions

subdivisions with a ratio of 1:3 on the radial lines. When you return to the **Mesh** menu, use **Bilinear 4 Side** to mesh region E. Because you changed the subdivisions for region C, you will also need to remesh that region. Your mesh should look like Figure 12.
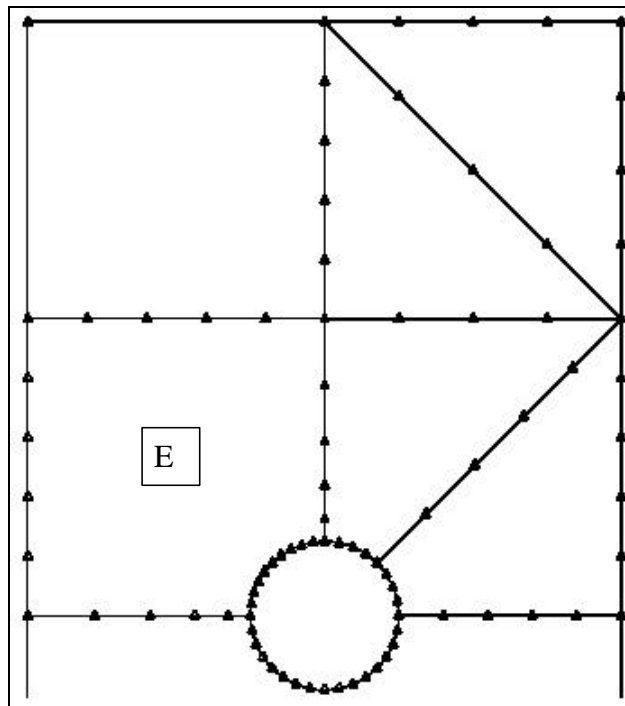
The important point is that although region E has 5 edges, the algorithm can recognize that the two edges with 5 subdivisions can be considered one edge with 10 sides, making it possible to use the **Bilinear 4 Side** algorithm.
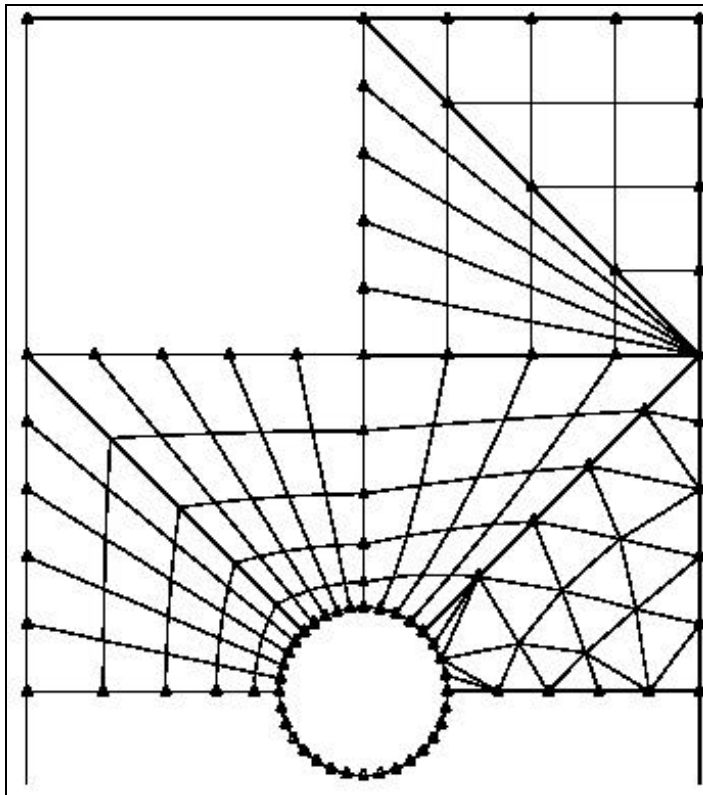
A similar situation can arise when a triangular region is defined by four sides (two of the edges form a straight line). In this case you can use either **Triangular Map** (if the total number of subdivisions is the same on all sides) or **Bilinear 3 Side** if only two sides have the same number of subdivisions. If prompted for the corners, start with the vertex opposite a side that has a different number of total subdivisions than the other two sides.

Figure 12: Meshed bilinear 4 side region

Final Comments:

For triangular regions, the **Triangular Map** algorithm requires an equal total number of subdivisions on three sides (a side may be composed of more than one edge). **Bilinear 3 Side** is most reliable predictable during automatic meshing if two of the three sides have the same number of subdivisions. All three of these meshing algorithms can work on irregularly shaped regions, but the user must define an appropriate discretization for each meshing algorithm.

The **Transition** meshing procedure can be used where it is impossible or difficult to define regularly shaped regions. The algorithm uses a quad-tree to recursively subdivide the region into smaller and smaller areas based on the subdivision density on the boundary of the region. Adjust the mesh density by adjusting the number and location of segments around the region.

Menu Description

This section describes each of the menu selections in CASCA.

Set Scale

The "Set Scale" initializes the scaling for the problem to be defined. Initially the window is a 12 x 12 grid. Each grid square is one unit wide.

Data Size

The "Data Size" is the window width and height in world coordinates. The default data size is 12 units. The current number of grid squares remains the same when the data size is changed; however, the spacing (described below) does change.

For example: change the data size to 24 units and the spacing changes to 2 units. Change the data size to 10 units and the Spacing XY to 2 units and the number of grid squares will be 5 per side. (Note that the effect of an odd number of grid squares is half a square on each side of the window rather than an even number across the window.)

Grid Center

The "Grid Center" is the position of the center of the default window. This allows more permanent mesh translation during model generation.   This is the reset position for the RESET menu option described below. The default center is (0,0).

Spacing XY

"Spacing XY" changes the distance for one grid square while maintaining the current Data Size. The default spacing is 1 unit.

Spacing X

"Spacing X" changes the spacing for only the X direction.

Spacing Y

"Spacing Y" changes the spacing for only the Y direction.

Restore

Restores the default spacings.

Geometry

Element generation can take place only after the problem geometry has been specified. Use the geometry commands to create **closed** geometric regions that can be meshed. Different material types must have different geometric regions.

Lines Connected

Draw multiple connected lines. Each segment can act as a side to a geometric region.

Get Line

Draw one line segment.

The line segment need not initially be attached to anything; however, before meshing can take place all segments must be part of some closed region.

Get Circle

Draw an ARC or a WHOLE circle. The option in all capital letters is in effect.
ARC: specify two points on the ends of the arch, then the arc center point. The arc will be drawn clockwise from the first point to the second point. Use the mouse to move the center point to adjust the arc radius.
WHOLE: specify a point on the edge of the circle, then specify the circle center. Use the mouse to move the center point to adjust the circle orientation and size.

Get Ellipse

Not implemented.

Get Bezier

Not implemented.

Delete

Select an item then hit Done. Repeat for more items.

Attributes

The analysis type and material types can be defined here. Usually this will take place before meshing; however, region material types can be changed after meshing as long as the regions were defined before meshing.

Analysis Type

Analysis type zero is Axisymmetric, type one is Plane Stress and type two is Plane Strain.

Materials

Multiple materials may be defined and may be either isotropic or orthotropic. Use the Attach option to change the material definition for a given region. Use the Check Region option to ensure that the material was assigned to the proper region.

Specify Hole

Holes must be identified so that the mesh generator will not automatically fill in a hole with elements. An unspecifed hole is simply another region.

Unspecify Hole

Change the hole back to a region.

Grid

Toggle the grid on and off.

Query

Click on end points to get coordinates.

Subregions

Sub-regions divide regions into smaller regions. Sub-regions that are part of the same region cannot have different material number assignments. Only regions have material assignments. Sub-regions simplify meshing in some cases and are especially useful for creating transition areas where element size will change significantly from one region to another. Use the sub-region commands in a similar manner to the Geometry commands.

Lines Connected

See description in Geometry (above).

Get Line

See description in Geometry (above).

Get Circle

See description in Geometry (above).

Split

Divides an edge by inserting a point. In most cases, splitting is handled automatically for the user. In some cases the user might want to control the location of a point. This allows the user to do that.

UnSplit

Reverses the Split command.

Subdivision Pts.

If the user has already proceeded to the Subdivide menu and subdivided edges, this command allows them to be displayed.

Grid

Displays grid and turns on snap-to.

Query

Click on end points to get coordinates.

Subdivide

Specify the distribution of elements in a region or sub-region. Before a region can be meshed the meshing algorithm must know the number of elements to place along each edge of a region or sub-region.

Num Segments

Number of elements along a sub-region edge. The default number of segments is four.

Ratio

The ratio of the longest segment to the shortest segment along an edge. The arrows on the edges denote the ratio direction. The ratio is always the length of the segment at the tail of the edge to the length of the segment at the head of the edge. The default ratio is one.

Quadratic/Linear

Not active.

Subdivide

Specify one or more edges to subdivide with the current number of segments and the current ratio. Use "Revert Ratio" to change the ratio for the next subdivision. To edit the number of subdivision points for an edge simply re-subdivide the edge with a different number of segments or a different ratio. Use the "Reset All" option described below to erase all of the subdivision points.

Revert Ratio

If the arrow of an edge is in the opposite direction that you want the ratio to operate, select revert ratio before selecting the edge for subdivision.

Num Segments

A quick way to change the number of subdivisions. The number is only local, returning to the Subdivide menu recovers the previously defined number of segments.

Ratio

Quick local change of the ratio, only local.

All Remaining

Subdivide all the edges that have not yet been subdivided.

Reset All

Erase all of the subdivision points for all of the edges. This is useful to restart the subdivision process rather than re-subdividing each edge.

Grid

Displays grid and turns on snap-to.

Mesh

After all the regions are subdivided they can be meshed. Currently four elements are supported: eight noded quadrilateral (Q8), four noded quadrilateral (Q4), six noded triangle (T6) and three noded triangle (T3).

Q8 Q4,   T6   T3

Select the desired element type as the default for mesh generation.

Right Left,   UnJck Optm

Select the desired element configuration for triangle element generation.

W/O Corner Pts

Toggle corner point specification on and off for Bilinear 3 and 4 Side element generation.

Bilinear 4 Side

Generate elements in a region using mapping to four sides of the region.

Bilinear 3 Side

Generate elements in a region using mapping to three sides of the region. This uses "Bilinear 4 Side" and collapses two of the corner points on top of each other.

Triangular Map

Generate elements in a region designed as a triangle. The mapping may be better using this method than for the "Bilinear 3 Side" method.

Transition

Generate elements in an arbitrary shaped region. The default is to use T6 elements unless T3 elements are specified. If "NO INTERIOR PT" is specified only the subdivision points on the region boundary will be used. If "GENERATE INT PT" is specified interior points will be generated to maintain triangle sizes consistent with the segment sizes along each edge.

Construct

Build elements manually, one edge at a time. May be useful for editing a transition mesh; however, even this is usually not needed. In some versions these options are not implemented.

Automatic

Generate elements for all regions not already meshed. The default element type will be used if possible, otherwise the T6 element will be used.

Delete

Unmesh a region. Point to the region to unmesh and hit Done.

Subdivision Pts.

Toggle the subdivision points on and off.

Mesh Boundary

Display only the mesh boundary. Go back into Mesh to redisplay the entire mesh.

Write Mesh

Write the current mesh to an output file. The file is formatted for FRANC input and will have a ".inp" file extension.

Read

Read a CASCA data file. This is a restart file written with the Write option below.

Write

Write a CASCA data file. The file is formatted for CASCA input using the READ option above and will have a ".csc" file extension. Use this for periodic saves during a working session as well as for restart files for later changes. Always save your work at the end of a session. CASCA does NOT save for you!

Grid

Toggle the grid on and off.

Read Grid

If the user has previously saved a mesh file, this can be read in to define a grid. This can be useful if the user wants to make modifications to previous work but does not have a saved *.csc file. Reading in a mesh as a grid allows the user to quickly rebuild the geometry by taking advantage of grid snap-to.

RESET

Reset the window to the startup view after Magnify, Zoom or Pan (described below).

MAGNIFY

Specify the new lower left and upper right corners of the display area. The current display perspective will be retained.

- ZOOM +

Magnify the display area keeping the same window center. Use the "+" to enlarge the image size and the "-" to reduce the image size.

PAN

The **PAN** menu allows the user to move the model inside the window. The pointer in the window acts as a potentiometer. Place the pointer just above center in the window and the viewing window will move up slightly. Place the pointer near the top of the window and the viewing window will move up much farther. Similar concepts apply for left, right and down. **PAN** also activates many of same view control menus available at a higher level. These include:

RESET

See above description.

SAVE VIEW

The current view is saved in a *.vw file. The user must type in the view file name in the *program control window*. The views are temporarily available as menu buttons for quick access.

RESTORE VIEW

Allows user to access a saved view file. The file name must be input in the *program control window*.

- ZOOM +

See above description.

MAGNIFY

See above description.

SNAP

Write a PostScript® output file of the current mesh or geometry model displayed. The file is named "gra$x$.ps," where x is the number of the output file for this session. Move each file to a new name or a new directory at the end of each session to prevent "gra0.ps" for the next session from overwriting "gra0.ps" from the previous session.

END

Exit the program. A restart file is NOT automatically written. Use "Write" to save your work before hitting **END** if you expect to restart later.

Acknowledgments

CASCA was originally developed by Paul Wawrzynek and Louis Martha at Cornell University. This work was done under the direction of Prof. Anthony Ingraffea, who has focused on numerical methods and crack growth. Brian Hardeman modified the gr aphics routines so that CASCA could be ported to Windows 95/NT. Mark James cleaned up some bugs and added some useful features.

Index