TECHNICAL REPORT

FOR

NASA Grant NAG-2-541

NASA Technical Monitor: Alex Woo

Institution: The Radiation Laboratory
Department of Electrical Engineering
and Computer Science
The University of Michigan
Ann Arbor, MI 48109-2122

August 15, 1990

Report Author: Chongyu Hua

Report Title: Users Manual for AUTOMESH-2D "A Program of Automatic Mesh Generation for Two-Dimensional Scattering Analysis by the Finite Element Method"

Principal Investigator: John L. Volakis
Telephone (313)764-0500

UMR0368

Table of Contents

Introduction

AUTOMESH-2D is a computer program specifically designed as a pre-processor for the scattering analysis of two-dimensional bodies by the finite element method. This program was developed due to a need for reducing the effort required to define and check the geometry data, element topology, and material properties. AUTOMESH-2D has the following functions and features:

1) The exterior boundary of a finite element analysis domain can be set to *rectangular* or *circular*.

2) The shape of a conducting body and dielectric interfaces can be arbitrary and be modeled by any combinations of straight lines, arcs and arbitrary points.

3) Different element densities can be designated for different parts of the analysis domain.

4) Different material properties can be specified for different parts of the analysis domain.

5) AUTOMESH-2D is user-friendly and easy to use.

AUTOMESH-2D employs an algorithm suggested by Cavendish [1]. A graphics package M-PLOT, developed by Professor K-P. Beier at the Univer-

sity of Michigan, provides basic tools for the graphic parts of the AUTOMESH-2D.

Structure of AUTOMESH-2D

A detailed description of the algorithm employed in AUTOMESH-2D is given by Cavendish [1]. This manual only provides a description of the program and its modules.

There are six modules in the program as shown in Figure 1. In the first modules, the following five parameters are specified: (1) data input device; (2) echo mode of input data; (3) the shape of the exterior boundary of an analysis domain; (4) mesh generation of structures; and (5) node generation index N. The definition of each parameter will become obvious later.

The Data Input Module requires the user to provide information such as the geometries of the outer boundary of the domain and the boundaries specifying a conducting body and dielectric interfaces. In addition, it requires information on the mesh densities and the material properties of the domain.

conducting bodies, layers of special media, mesh densities, and material properties.

The Node Generation Module automatically generates interior nodal points consistent with mesh density information. This process is illustrated in Figure 2. A subdomain with a specified *mesh density factor* r is shown in Figure

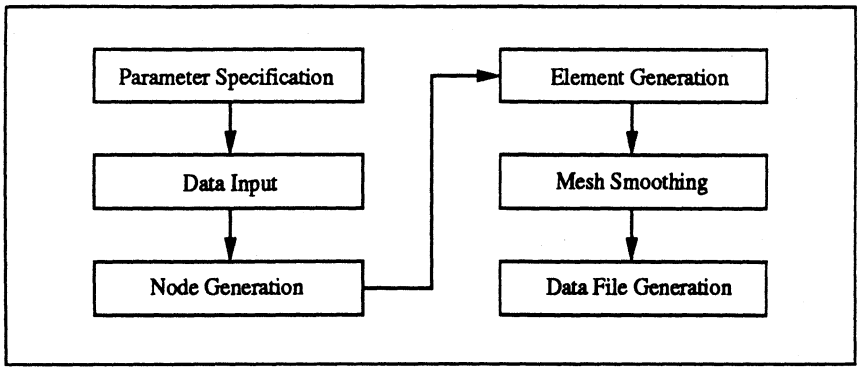


Figure 1: Six Modules in AUTOMESH-2D

2(a). A rectilinear net in which the given subdomain is contained is defined according to the value of r and shown in Figure 2(b). Starting from the lower left hand subsquare of the rectilinear net and working from bottom to top and from left to right, an attempt is made to *randomly* generate one interior node in each subsquare. In any subsquare, a generated point will be retained as an interior node if and only if it lies in the subdomain in Figure 2(a) and all other nodal points (boundary nodes and previously generated nodes) lie outside of the disc of radius r centered at the point in question. The *node generation index* N, specified at Parameter Specification Module, is defined as the maximum number of attempts which are made to generate a retainable random point in any one subsquare. If no point is successfully generated after N consecutive attempts, then no node is designated in that subsquare and the next subsquare is considered.

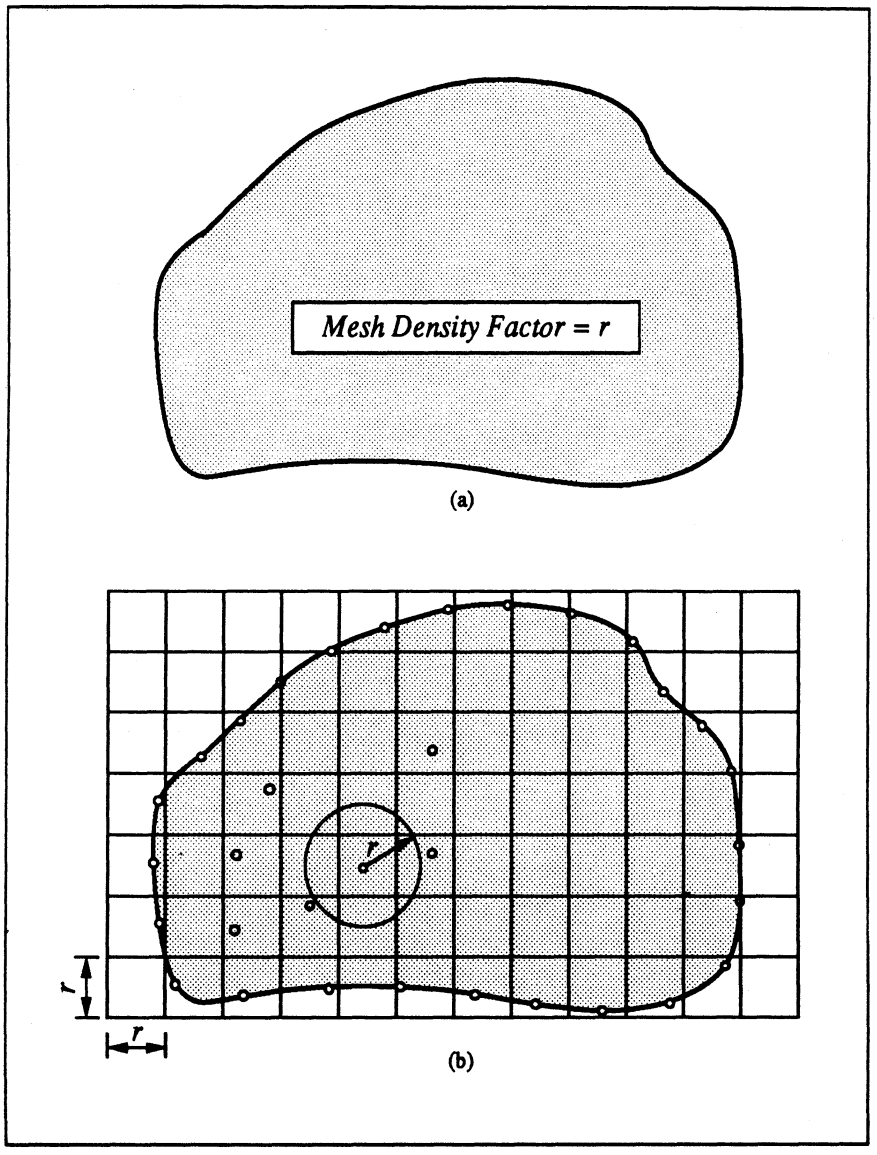


Figure 2: Generation of Interior Nodal Points

The Element Generation Module is used to interconnect nodal points to form valid triangulation consistent with requirement that elements be as close to equilateral triangles as possible.

A Mesh Smoothing Module is designed to perturb the triangulation by a smoothing technique so that elements are more closely equilateral triangles.

Finally, a data file is generated by the Data File Generation Module. This data file is in a format consistent with the main program for finite element analysis [2, 3].

Running AUTOMESH-2D

The procedure for running AUTOMESH-2D is relatively easy. After the user types *automesh* dialogue will begin for entering information. Three examples will be shown later. The following paragraphs present footnotes to some items which may cause confusion.

Input data device. Control information for a mesh generation can either be entered from the ***keyboard***, or be read from a *control file*. This option is selected at the beginning when parameters are specified. When the ***input data device keyboard*** is *ON*, then all control data are entered from the **keyboard**. Otherwise, a file containing input directives will be asked for input. The default option is keyboard *ON*.

Echo mode of the program. When this option is set to *OFF*, no echo data of the current run will be saved to a file. On the other hand, when this

option is set to *ON*, then all data which are either entered from the keyboard or read from a control file will be written to an echo file for later use as a control file. The default for this option is *OFF*.

Mesh generation of structures. This parameter can be used to specify whether nodal points will be generated ***inside*** conducting bodies. The default option is set to *NO*.

Spacing of nodes on the exterior boundary. Nodal points on the boundary of an analysis domain are equally spaced.

Boundary of structures. The boundary of structures is the boundary of conducting bodies.

Layers of special media. This option is designed for specifying different material properties for certain parts of the analysis domain such as a coated layer around a conducting body. In addition to the material properties for the other portion of the domain, different material properties are required for the layers of special media (if any).

Zones of higher mesh density. This option can be used for specifying different mesh densities for certain portion of the domain such as the region around a corner. This option will be effective *only* when the *mesh density factor* for a special zone is *less* than the *mesh density factor* for the adjacent portion in the domain.

Definition of a boundary. The shape of a boundary (for a conducting

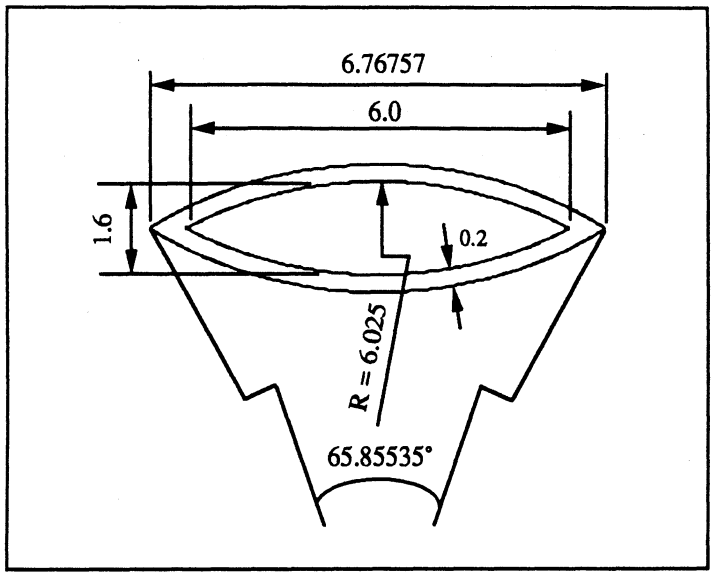


Figure 3: The Geometry of Ogival Coated Cylinder

body, or a layer of a special medium, or a zone of higher mesh density) can be defined by any combination of a number of curve segments. Three types of curve segments can be used: (1) a straight line, (2) an arc, and (3) an arbitrary curve segment. When defining a boundary, a starting point on the boundary must be specified, with the remaining curve segments defining the boundary entered in a *counterclockwise* order.

Example Problems

Example 1: Ogival coated cylinder in rectangular domain. This problem involves the use of most functions and features provided by AUTOMESH-2D. The geometry of the conducting body is shown in Figure 3, and is coated

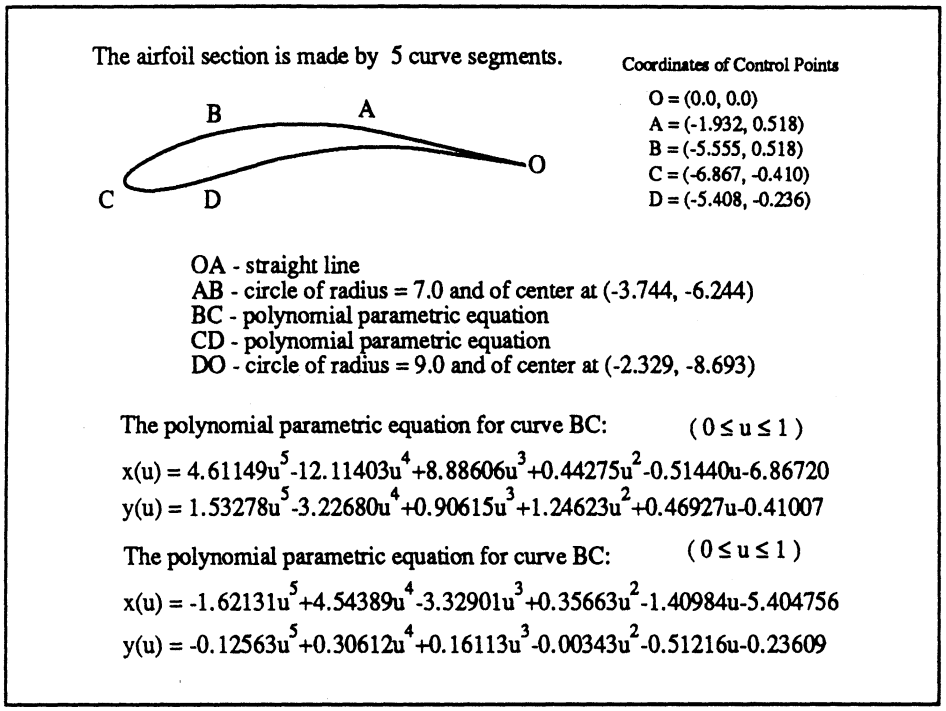


Figure 4: The Geometry of An Airfoil

with a dielectric layer of thickness 0.2 having material properties of $\epsilon_r = 3$ and $\mu_r = 1$. The keyboard dialogue, the echo file, and the finite element mesh are shown in Appendix I.

Example 2: Ogival coated cylinder in circular domain. The purpose of this example is to show the use of a control file for data input. Since the geometry of the conducting body in this example is the same as that in Example 1, the echo file generated from Example 1, *example1.echo*, is mod-

ified and stored in a new file, *example2.echo*. The only changes are in the definition of an exterior boundary and the specification of mesh densities. The information entered from the keyboard, the control file (*example2.echo*), and the finite element mesh are shown in Appendix II.

Example 3: An airfoil in rectangular domain. The geometry of the airfoil is shown in Figure 4. This example problem shows the generation of a relatively complex boundary. The data entered from the keyboard, the echo file, and the finite element mesh are shown in Appendix III.

Reference

[1] Cavendish, J. C., "Automatic Triangulation of Arbitrary Planar Domains for the Finite Element Method", *International Journal for Numerical Methods in Engineering*, Vol. 8, pp. 679-696, 1974.

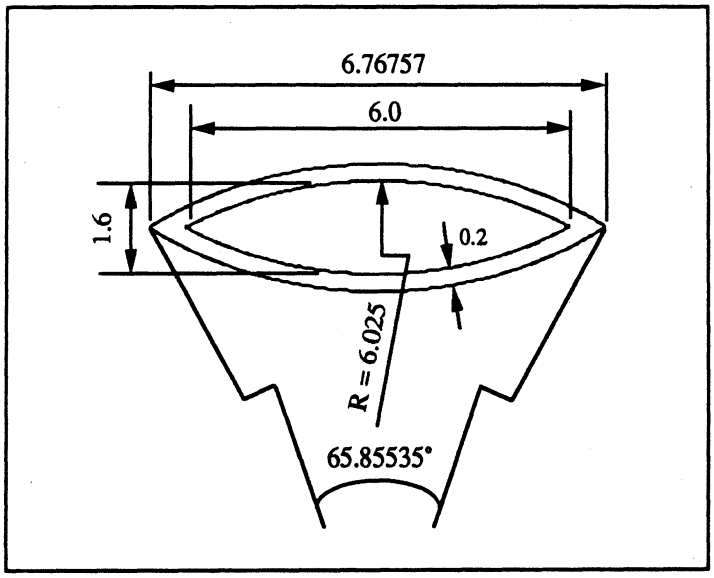
[2] Collins, J. D. and J. L. Volakis, *A Combined Finite Element and Boundary Integral Formulation for Solution via CGFFT of Two-Dimensional Scattering Problems*, University of Michigan Radiation Laboratory, Technical Report No. 025921-6-T, September, 1989, 73pp.

[3] Collins, J. D., J. M. Jin and J. L. Volakis, *A Combined Finite Element-Boundary Element Formulation for Solution of Two-Dimensional Problems via CGFFT*, University of Michigan Radiation Laboratory, Technical Report No. 025921-11-T, February, 1990, 45pp.

Appendix I

Results for Example 1

Problem Geometry:



Shape of the Domain: Rectangular

```
-------------------------------------------------------
      EXAMPLE 1     Rectangular Boundary Domain
-------------------------------------------------------

% automesh

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

             A TRIANGULAR MESH GENERATOR
             FOR FINITE ELEMENT ANALYSIS

                 Radiation Laboratory
                  Department of EECS
               The University of Michigan
                     Ann Arbor

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

 -------- C U R R E N T   P A R A M E T E R S --------

        1. Input data device keyboard is : ON

        2. Echo mode of the program is   : OFF

        3. Exterior boundary is          : RECTANGLE

        4. Mesh generation of structures : NO

        5. Node generation index is      : 5

        6. Commence mesh generation

      To alter any of the above parameters type
      the corresponding item number or 6 to exit
      this menu or 0 to stop the program

2

 -------- C U R R E N T   P A R A M E T E R S --------

        1. Input data device keyboard is : ON

        2. Echo mode of the program is   : ON

        3. Exterior boundary is          : RECTANGLE

        4. Mesh generation of structures : NO

        5. Node generation index is      : 5

        6. Commence mesh generation
```

```
      To alter any of the above parameters type
      the corresponding item number or 6 to exit
      this menu or 0 to stop the program

6

 E X E C U T I O N   B E G I N S

 >> Enter a file name to save the echo data

example1.echo

 ========== EXTERIOR BOUNDARY==========

 >> Enter the X and Y coordinates of two
 >> diagonal corner points to define the
 >> rectangular exterior boundary

 --> Corner No.1:

-3.7 -1.4

 --> Corner No.2:

3.7 1.4

 >> What is the nodal spacing on the boundary ?

0.1

 ========== BOUNDARY OF STRUCTURES ==========

 >> Note: The boundary of structures can be
 >>       specified by any combination of
 >>       the following curve segments in
 >>       a COUNTERCLOCKWISE order:
 >>       (1) Straight lines;
 >>       (2) Arcs; and
 >>       (3) Arbitrary curves.

 >> Enter the number of curve segments to define
 >> the structure boundary

2

 >> Enter the X and Y coordinates of a starting
 >> point on the boundary

3.0 0.0
```

```
 >> ---------- Curve Segment No.1 ----------

 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

2

 >> To define the arc, select an option
 >> for entering parameters:
 >> Options:
 >>   1 - The arc will be defined by the
 >>       coordinates of a center point
 >>       and the angle of the arc
 >>   2 - The arc will be defined by the
 >>       coordinates of two additional
 >>       points of the arc

2

 >> NOTE: It is required that the angle of the arc
 >>       formed by three points must be equal or
 >>       less than 180 degrees.

 >> Enter the following parameters:
 >>   (1) The X and Y coordinates of the ending
 >>       point of the arc
 >>   (2) The X and Y coordinates of a third point
 >>       on the arc (except the starting point
 >>       and the ending point
 >>   (3) The number of points on the arc (excluding
 >>       the starting point)

-3.0 0.0 0.0 0.8 50

 >> ---------- Curve Segment No.2 ----------

 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

2

 >> To define the arc, select an option
 >> for entering parameters:
 >> Options:
 >>   1 - The arc will be defined by the
 >>       coordinates of a center point
 >>       and the angle of the arc
 >>   2 - The arc will be defined by the
 >>       coordinates of two additional
 >>       points of the arc
```

```
2

 >> NOTE: It is required that the angle of the arc
 >>       formed by three points must be equal or
 >>       less than 180 degrees.

 >> Enter the following parameters:
 >>   (1) The X and Y coordinates of the ending
 >>       point of the arc
 >>   (2) The X and Y coordinates of a third point
 >>       on the arc (except the starting point
 >>       and the ending point
 >>   (3) The number of points on the arc (excluding
 >>       the starting point)

3.0 0.0 0.0 -0.8 50


 ========== LAYERS OF SPECIAL MEDIA ==========

 >> Note: Media with special material properties
 >>       can be specified within the problem
 >>       domain enclosed by the rectangular
 >>       exterior boundary given above.


 >> How many layers of such media will be considered ?

1

 >> Now, please define each layer of special medium
 >> by specifying an close boundary which can be
 >> formed by any combination of the following curve
 >> segments in a COUNTERCLOCKWISE order:
 >>    (1) Straight lines;
 >>    (2) Arcs; and
 >>    (3) Arbitrary curves.


 >> :::: The Boundary of Special Medium No.1 ::::

 >> Enter the number of curve segments to define
 >> the interior boundary of this layer of medium

2

 >> Enter the X and Y coordinates of a starting
 >> point on the boundary

3.383785 0.0

 >> ---------- Curve Segment No.1 ----------
```

```
 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

2

 >> To define the arc, select an option
 >> for entering parameters:
 >> Options:
 >>   1 - The arc will be defined by the
 >>       coordinates of a center point
 >>       and the angle of the arc
 >>   2 - The arc will be defined by the
 >>       coordinates of two additional
 >>       points of the arc

2

 >> NOTE: It is required that the angle of the arc
 >>       formed by three points must be equal or
 >>       less than 180 degrees.

 >> Enter the following parameters:
 >>   (1) The X and Y coordinates of the ending
 >>       point of the arc
 >>   (2) The X and Y coordinates of a third point
 >>       on the arc (except the starting point
 >>       and the ending point
 >>   (3) The number of points on the arc (excluding
 >>       the starting point)

-3.383785 0.0 0.0 1.0 50

 >> ---------- Curve Segment No.2 ----------

 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

2

 >> To define the arc, select an option
 >> for entering parameters:
 >> Options:
 >>   1 - The arc will be defined by the
 >>       coordinates of a center point
 >>       and the angle of the arc
 >>   2 - The arc will be defined by the
 >>       coordinates of two additional
 >>       points of the arc

2
```

```
 >> NOTE: It is required that the angle of the arc
 >>       formed by three points must be equal or
 >>       less than 180 degrees.

 >> Enter the following parameters:
 >>   (1) The X and Y coordinates of the ending
 >>       point of the arc
 >>   (2) The X and Y coordinates of a third point
 >>       on the arc (except the starting point
 >>       and the ending point
 >>   (3) The number of points on the arc (excluding
 >>       the starting point)

3.383785 0.0 0.0 -1.0 50


 ======== ZONES OF HIGHER MESH DENSITY ========

 >> Note: You can also specify some zones with HIGHER
 >>       mesh density. In other words in addition to
 >>       the nodal points generated for the problem
 >>       domain, more nodal points will be generated
 >>       for these zones if you specify a smaller mesh
 >>       density factor later.

 >> How many such zones will be considered ?

1

 >> The zone boundary can also be formed by any
 >> combination of the following curve segments
 >> in a COUNTERCLOCKWISE order:
 >>    (1) Straight lines;
 >>    (2) Arcs; and
 >>    (3) Arbitrary curves.


 >> :::::::::: The Zone Boundary No.1 ::::::::::

 >> Enter the number of curve segments to define
 >> the zone boundary

2

 >> Enter the X and Y coordinates of a starting
 >> point on the boundary

3.383785 0.0

 >> ---------- Curve Segment No.1 ----------

 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
```

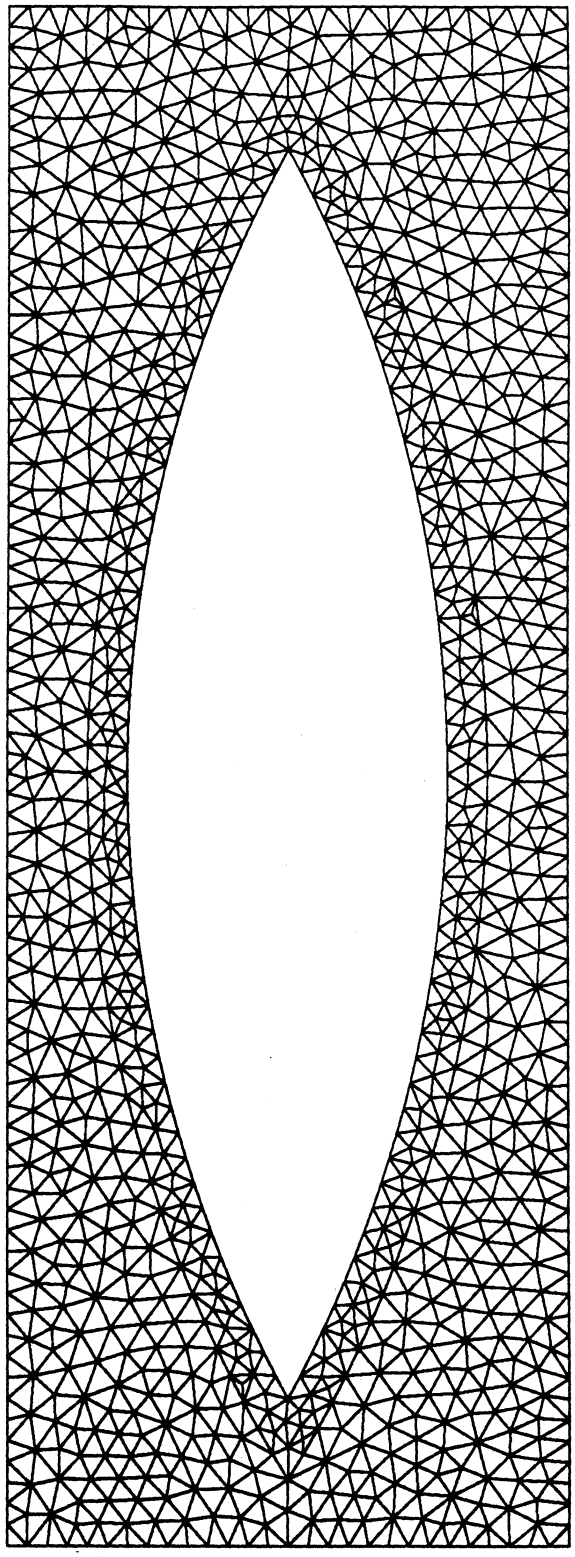
```
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

2

 >> To define the arc, select an option
 >> for entering parameters:
 >> Options:
 >>   1 - The arc will be defined by the
 >>       coordinates of a center point
 >>       and the angle of the arc
 >>   2 - The arc will be defined by the
 >>       coordinates of two additional
 >>       points of the arc

1

 >> Enter the following parameters:
 >>   (1) The X and Y coordinates of the center point
 >>   (2) The angle of the arc in degrees
 >>       (+) for counterclockwise
 >>       (-) for clockwise
 >>   (3) The number of points on the arc (excluding
 >>       the starting point)
 >>       Note: These points are for reference only,
 >>             and will not be counted as nodes.

0.0 -5.225 65.85535 50

 >> ---------- Curve Segment No.2 ----------

 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

2

 >> To define the arc, select an option
 >> for entering parameters:
 >> Options:
 >>   1 - The arc will be defined by the
 >>       coordinates of a center point
 >>       and the angle of the arc
 >>   2 - The arc will be defined by the
 >>       coordinates of two additional
 >>       points of the arc

1

 >> Enter the following parameters:
 >>   (1) The X and Y coordinates of the center point
 >>   (2) The angle of the arc in degrees
 >>       (+) for counterclockwise
```

```
 >>        (-) for clockwise
 >>    (3) The number of points on the arc (excluding
 >>        the starting point)
 >>        Note: These points are for reference only,
 >>              and will not be counted as nodes.

0.0 5.225 65.85535 50


 ========== SPECIFICATION OF MESH DENSITY ==========

 >> Note: A positive factor is required for the
 >>       problem domain and every speical zones.
 >>       This factor will be used as a reference
 >>       to control the density of nodal point
 >>       generation. Statistically, the average
 >>       size of element will be equal to the
 >>       density factor and the distance between
 >>       any two nodal points will not be less than
 >>       this factor.


 >> Enter a mesh density factor for the part of
 >> problem domain not included by speical zones

0.1

 >> Enter a mesh density factor for Zone No.1

0.07


 ========== TWO MATERIAL PROPERTIES==========


 >> ---- (1) RELATIVE PERMITTIVITY (EPS) ----

 >> Enter Re(EPS) and Im(EPS) for the part of
 >> problem domain not included by layers of
 >> special media

1.0 0.0

 >> Enter Re(EPS) and Im(EPS) for
 >> special medium No.1

3.0 0.0

 >> ---- (2) RELATIVE PERMEABILITY (MU) ----

 >> Enter Re(MU) and Im(MU) for the part of
 >> problem domain not included by layers of
 >> special media
```

```
1.0 0.0

 >> Enter Re(MU) and Im(MU) for
 >> special medium No.1

1.0 0.0

 >> NODE GENERATION ...

    Total number of nodes = 1161


 >> ELEMENT GENERATION ...

    Total number of elements = 2018

 >> MESH SMOOTHING ...

 >> Do you want to plot the nodal
 >> points ? (Yes/No)

y

 >> Do you want to print nodal numbers
 >> on the plot ? (Yes/No)

n

 >> Do you want to modify the plot ? (Yes/No)

n

 >> Do you want to plot the element
 >> mesh ? (Yes/No)

y

 >> Do you want to print element numbers
 >> on the plot ? (Yes/No)

n

 >> Do you want to modify the plot ? (Yes/No)

n

 >> Enter a file name to save data

example1.dat

 >> DATA FILE GENERATION ...


 E X E C U T I O N   T E R M I N A T E D
```

```
 The following files have been created :

 (1) example1.dat      - contains the finite element
                         data generated by this run

 (2) example1.echo     - contains the echo data of
                         the current session.

Fortran STOP
```

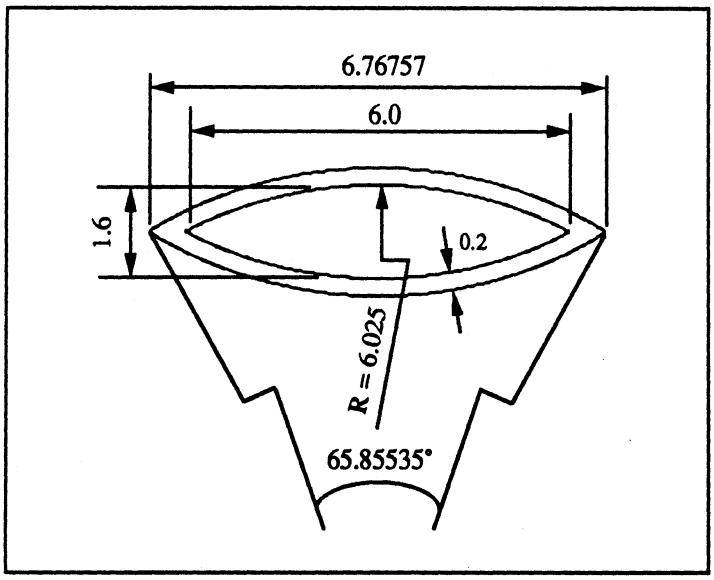
```
-----------------------------------------------------
             FILE          example1.echo
-----------------------------------------------------

-3.700000 -1.400000
3.700000 1.400000
0.1000000
2
3.000000 0.0000000
2
2
-3.000000 0.0000000
0.0000000 0.8000000
50
2
2
3.000000 0.0000000
0.0000000 -0.8000000
50
1
2
3.383785 0.0000000
2
2
-3.383785 0.0000000
0.0000000 1.000000
50
2
2
3.383785 0.0000000
0.0000000 -1.000000
50
1
2
3.383785 0.0000000
2
1
0.0000000 -5.225000
65.85535
50
2
1
0.0000000 5.225000
65.85535
50
0.1000000
7.0000000E-02
1.000000 0.0000000
3.000000 0.0000000
1.000000 0.0000000
1.000000 0.0000000
```



Appendix II

Results for Example 2

Problem Geometry:



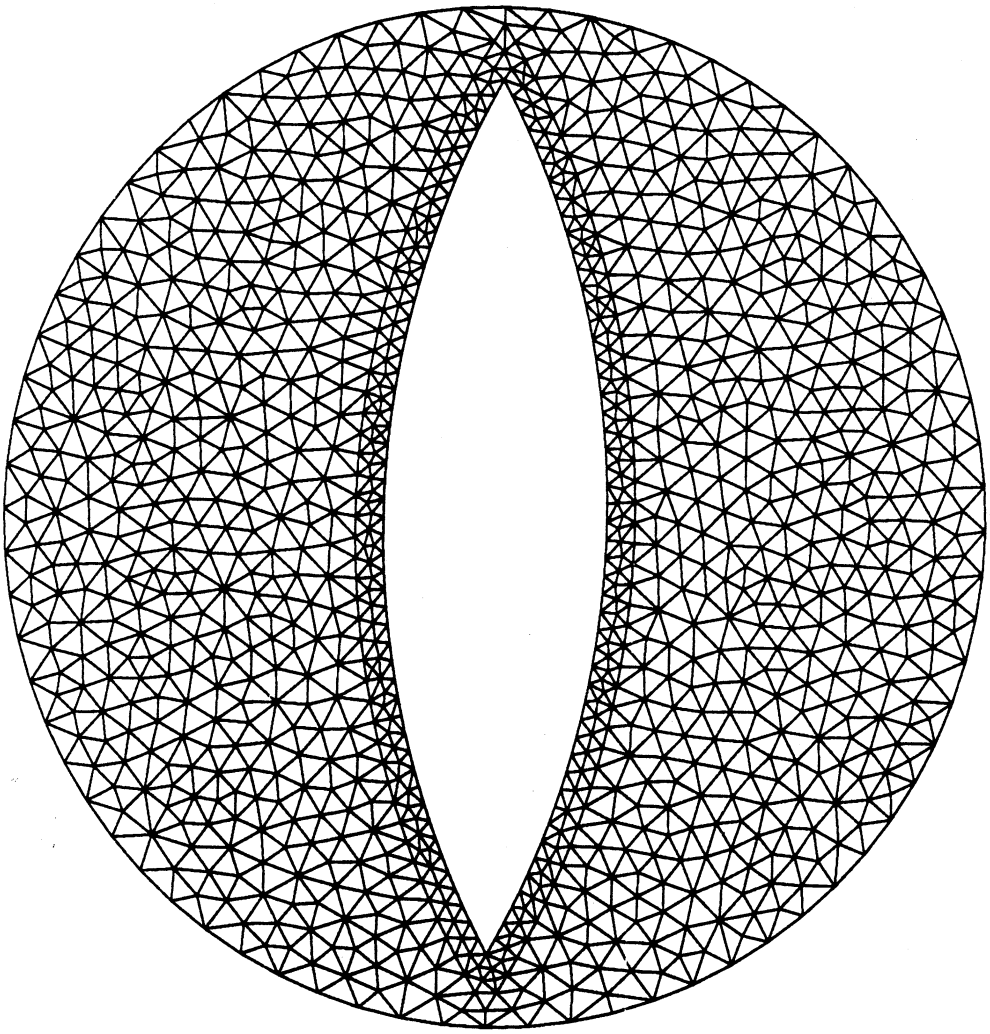
Shape of the Domain: Circular

```
-----------------------------------------------------
         EXAMPLE 2    Circular Boundary Domain
-----------------------------------------------------

% automesh

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

             A TRIANGULAR MESH GENERATOR
             FOR FINITE ELEMENT ANALYSIS

                 Radiation Laboratory
                 Department of EECS
               The University of Michigan
                     Ann Arbor

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


 -------- C U R R E N T   P A R A M E T E R S --------

        1. Input data device keyboard is : ON

        2. Echo mode of the program is   : OFF

        3. Exterior boundary is          : RECTANGLE

        4. Mesh generation of structures : NO

        5. Node generation index is      : 5

        6. Commence mesh generation

      To alter any of the above parameters type
      the corresponding item number or 6 to exit
      this menu or 0 to stop the program

1

 -------- C U R R E N T   P A R A M E T E R S --------

        1. Input data device keyboard is : OFF

        2. Echo mode of the program is   : OFF

        3. Exterior boundary is          : RECTANGLE

        4. Mesh generation of structures : NO

        5. Node generation index is      : 5

        6. Commence mesh generation
```

```
      To alter any of the above parameters type
      the corresponding item number or 6 to exit
      this menu or 0 to stop the program

3

 -------- C U R R E N T   P A R A M E T E R S --------

        1. Input data device keyboard is : OFF

        2. Echo mode of the program is   : OFF

        3. Exterior boundary is          : CIRCLE

        4. Mesh generation of structures : NO

        5. Node generation index is      : 5

        6. Commence mesh generation

      To alter any of the above parameters type
      the corresponding item number or 6 to exit
      this menu or 0 to stop the program

6


 E X E C U T I O N   B E G I N S


 >> Please enter the file name containing
 >> the input directives

example2.echo

 >> DATA INPUT ....

 >> NODE GENERATION ...

    Total number of nodes = 1126

 >> ELEMENT GENERATION ...

    Total number of elements = 2080

 >> MESH SMOOTHING ...

 >> Do you want to plot the nodal
 >> points ? (Yes/No)

n

 >> Do you want to plot the element
 >> mesh ? (Yes/No)
```

```
y

 >> Do you want to print element numbers
 >> on the plot ? (Yes/No)

n

 >> Do you want to modify the plot ? (Yes/No)

n

 >> Enter a file name to save data

example2.dat

 >> DATA FILE GENERATION ...


 E X E C U T I O N   T E R M I N A T E D

 The following file has been created :

 example2.dat     - contains the finite element
                    data generated by this run


Fortran STOP
```

```
-----------------------------------------------------
              FILE          example2.echo
-----------------------------------------------------

 0.0 0.0
 3.5
 72
 2
 3.000000 0.0000000
 2
 2
 -3.000000 0.0000000
 0.0000000 0.8000000
 50
 2
 2
 3.000000 0.0000000
 0.0000000 -0.8000000
 50
 1
 2
 3.383785 0.0000000
 2
 2
 -3.383785 0.0000000
 0.0000000 1.000000
 50
 2
 2
 3.383785 0.0000000
 0.0000000 -1.000000
 50
 1
 2
 3.383785 0.0000000
 2
 1
 0.0000000 -5.225000
 65.85535
 50
 2
 1
 0.0000000 5.225000
 65.85535
 50
 0.1500000
 0.0800000
 1.000000 0.0000000
 3.000000 0.0000000
 1.000000 0.0000000
 1.000000 0.0000000
```

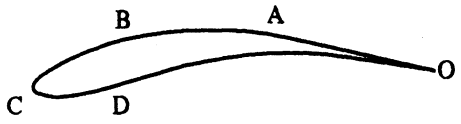


Appendix III

Results for Example 3

Problem Geometry:

The airfoil section is made by 5 curve segments.



Coordinates of Control Points

O = (0.0, 0.0)
A = (-1.932, 0.518)
B = (-5.555, 0.518)
C = (-6.867, -0.410)
D = (-5.408, -0.236)

OA - straight line
AB - circle of radius = 7.0 and of center at (-3.744, -6.244)
BC - polynomial parametric equation
CD - polynomial parametric equation
DO - circle of radius = 9.0 and of center at (-2.329, -8.693)

The polynomial parametric equation for curve BC: $(0 \le u \le 1)$

$$x(u) = 4.61149u^5 - 12.11403u^4 + 8.88606u^3 + 0.44275u^2 - 0.51440u - 6.86720$$
$$y(u) = 1.53278u^5 - 3.22680u^4 + 0.90615u^3 + 1.24623u^2 + 0.46927u - 0.41007$$

The polynomial parametric equation for curve BC: $(0 \le u \le 1)$

$$x(u) = -1.62131u^5 + 4.54389u^4 - 3.32901u^3 + 0.35663u^2 - 1.40984u - 5.404756$$
$$y(u) = -0.12563u^5 + 0.30612u^4 + 0.16113u^3 - 0.00343u^2 - 0.51216u - 0.23609$$

Shape of the Domain: Rectangular

```
-----------------------------------------------------
EXAMPLE 3    An Airfoil in Rectangular Boundary Domain
-----------------------------------------------------

% automesh

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

             A TRIANGULAR MESH GENERATOR
             FOR FINITE ELEMENT ANALYSIS

                 Radiation Laboratory
                  Department of EECS
               The University of Michigan
                     Ann Arbor

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


 --------C U R R E N T   P A R A M E T E R S --------

        1. Input data device keyboard is : ON

        2. Echo mode of the program is   : OFF

        3. Exterior boundary is          : RECTANGLE

        4. Mesh generation of structures : NO

        5. Node generation index is      : 5

        6. Commence mesh generation

      To alter any of the above parameters type
      the corresponding item number or 6 to exit
      this menu or 0 to stop the program

2

 -------- C U R R E N T   P A R A M E T E R S --------

        1. Input data device keyboard is : ON

        2. Echo mode of the program is   : ON

        3. Exterior boundary is          : RECTANGLE

        4. Mesh generation of structures : NO

        5. Node generation index is      : 5

        6. Commence mesh generation
```

```
      To alter any of the above parameters type
      the corresponding item number or 6 to exit
      this menu or 0 to stop the program

6


 E X E C U T I O N   B E G I N S


 >> Enter a file name to save the echo data

example3.echo


 ========== EXTERIOR BOUNDARY==========

 >> Enter the X and Y coordinates of two
 >> diagonal corner points to define the
 >> rectangular exterior boundary

 --> Corner No.1:

-7.5 -0.7

 --> Corner No.2:

0.7 1.0

 >> What is the nodal spacing on the boundary ?

0.1


 ========== BOUNDARY OF STRUCTURES ==========

 >> Note: The boundary of structures can be
 >>       specified by any combination of
 >>       the following curve segments in
 >>       a COUNTERCLOCKWISE order:
 >>       (1) Straight lines;
 >>       (2) Arcs; and
 >>       (3) Arbitrary curves.

 >> Enter the number of curve segments to define
 >> the structure boundary

4

 >> Enter the X and Y coordinates of a starting
 >> point on the boundary

-1.932 0.518
```

```
 >> ---------- Curve Segment No.1 ----------

 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

2

 >> To define the arc, select an option
 >> for entering parameters:
 >> Options:
 >>   1 - The arc will be defined by the
 >>       coordinates of a center point
 >>       and the angle of the arc
 >>   2 - The arc will be defined by the
 >>       coordinates of two additional
 >>       points of the arc

1

 >> Enter the following parameters:
 >>   (1) The X and Y coordinates of the center point
 >>   (2) The angle of the arc in degrees
 >>       (+) for counterclockwise
 >>       (-) for clockwise
 >>   (3) The number of points on the arc (excluding
 >>       the starting point)

-3.744 -6.244 29.99412 20

 >> ---------- Curve Segment No.2 ----------

 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

3

 >> To define the arbitrary segment, enter the
 >> number of nodal points on the segment

20

 >> Enter the X and Y coordinates of each nodal
 >> point on the arbitrary curve segment:


 --> Point No.1:

 -5.718571 0.4702906

 --> Point No.2:
```

-5.896511 0.4074460

--> Point No.3:

-6.095940 0.3227407

--> Point No.4:

-6.308450 0.2168588

--> Point No.5:

-6.515973 9.5615506E-02

--> Point No.6:

-6.696310 -3.1882107E-02

--> Point No.7:

-6.828667 -0.1550748

--> Point No.8:

-6.899188 -0.2637901

--> Point No.9:

-6.906492 -0.3500820

--> Point No.10:

-6.867200 -0.4100700

--> Point No.11:

-6.790515 -0.4556865

--> Point No.12:

-6.681735 -0.4812943

--> Point No.13:

-6.543055 -0.4886304

--> Point No.14:

-6.381328 -0.4799125

--> Point No.15:

-6.206121 -0.4576797

```
 --> Point No.16:

 -6.027770 -0.4246402

 --> Point No.17:

 -5.855433 -0.3835219

 --> Point No.18:

 -5.695143 -0.3369205

 --> Point No.19:

 -5.547868 -0.2871498

 --> Point No.20:

 -5.407560 -0.2360900

 >> ---------- Curve Segment No.3 ----------

 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

2

 >> To define the arc, select an option
 >> for entering parameters:
 >> Options:
 >>   1 - The arc will be defined by the
 >>       coordinates of a center point
 >>       and the angle of the arc
 >>   2 - The arc will be defined by the
 >>       coordinates of two additional
 >>       points of the arc

1

 >> Enter the following parameters:
 >>   (1) The X and Y coordinates of the center point
 >>   (2) The angle of the arc in degrees
 >>       (+) for counterclockwise
 >>       (-) for clockwise
 >>   (3) The number of points on the arc (excluding
 >>       the starting point)

-2.329 -8.693 -35.00367 30

 >> ---------- Curve Segment No.4 ----------
```

```
 >> Enter 1 or 2 or 3 to select curve type:
 >>   1 --> a straight line
 >>   2 --> an arc
 >>   3 --> an arbitrary curve segment

1

 >> To define the straight line, enter the following parameters:
 >>   (1) The X and Y coordinates of an ending point
 >>   (2) The number of nodal points on the straight line
 >>       (excluding the starting point)

-1.932 0.518 10


 ========== LAYERS OF SPECIAL MEDIA ==========

 >> Note: Media with special material properties
 >>       can be specified within the problem
 >>       domain enclosed by the rectangular
 >>       exterior boundary given above.


 >> How many layers of such media will be considered ?

0


 ======== ZONES OF HIGHER MESH DENSITY ========

 >> Note: You can also specify some zones with HIGHER
 >>       mesh density. In other words in addition to
 >>       the nodal points generated for the problem
 >>       domain, more nodal points will be generated
 >>       for these zones if you specify a smaller mesh
 >>       density factor later.

 >> How many such zones will be considered ?

0


 ========== SPECIFICATION OF MESH DENSITY ==========

 >> Note: A positive factor is required for the
 >>       problem domain and every speical zones.
 >>       This factor will be used as a reference
 >>       to control the density of nodal point
 >>       generation. Statistically, the average
 >>       size of element will be equal to the
 >>       density factor and the distance between
 >>       any two nodal points will not be less than
 >>       this factor.
```

```
 >> Enter a mesh density factor

0.1

 ========== TWO MATERIAL PROPERTIES==========

 >> ---- (1) RELATIVE PERMITTIVITY (EPS) ----

 >> Enter Re(EPS) and Im(EPS)

1.0 0.0

 >> ---- (2) RELATIVE PERMEABILITY (MU) ----

 >> Enter Re(MU) and Im(MU)

1.0 0.0

 >> NODE GENERATION ...

    Total number of nodes = 935

 >> ELEMENT GENERATION ...

    Total number of elements = 1592

 >> MESH SMOOTHING ...

 >> Do you want to plot the nodal
 >> points ? (Yes/No)

y

 >> Do you want to print nodal numbers
 >> on the plot ? (Yes/No)

n

 >> Do you want to modify the plot ? (Yes/No)

n

 >> Do you want to plot the element
 >> mesh ? (Yes/No)

y

 >> Do you want to print element numbers
 >> on the plot ? (Yes/No)

n
```

```
 >> Do you want to modify the plot ? (Yes/No)
n
 >> Enter a file name to save data
example3.dat
 >> DATA FILE GENERATION ...

 E X E C U T I O N   T E R M I N A T E D

 The following files have been created :

 (1) example3.dat      - contains the finite element
                         data generated by this run

 (2) example3.echo     - contains the echo data of
                         the current session.

Fortran STOP
```

```
------------------------------------------------
       File       example3.echo
------------------------------------------------

 -7.5 -0.700000
 0.7 1.0000
 0.1000000
 4
 -1.932 0.518
 2
 1
 -3.744000 -6.244000
 29.99412
 20
 3
 20
 -5.718571 0.4702906
 -5.896511 0.4074460
 -6.095940 0.3227407
 -6.308450 0.2168588
 -6.515973 9.5615506E-02
 -6.696310 -3.1882107E-02
 -6.828667 -0.1550748
 -6.899188 -0.2637901
 -6.906492 -0.3500820
 -6.867200 -0.4100700
 -6.790515 -0.4556865
 -6.681735 -0.4812943
 -6.543055 -0.4886304
 -6.381328 -0.4799125
 -6.206121 -0.4576797
 -6.027770 -0.4246402
 -5.855433 -0.3835219
 -5.695143 -0.3369205
 -5.547868 -0.2871498
 -5.407560 -0.2360900
 2
 1
 -2.329000 -8.693000
 -35.00367
 30
 1
 -1.932000 0.5180000
 10
 0
 0
 0.1000000
 1.000000 0.0000000
 1.000000 0.0000000
```