FAST4 User Manual

September 2006

Chapter I

Chapter II

1.1 Introduction

FAST4 is a program that performs analysis of general axi-symmetric shell structure for both thick and thin shells. Through specifying the physical properties of a system including material properties and geometrical variations, the resulting mechanical changes on the system from a stimulus can be computed. Mechanical changes of the system such as stress, strain, and displacement data are included in an output file and may be used to visualize how the system responds to the stimulus. The effect of shear deformation and other thick shell corrections are included in the computation.

The applications of FAST4 are vast and include most systems having an axi-symmetric structure that can be subjected to a mechanical perturbation. Complicated biological systems such as the organ of corti movement due to basilar membrane oscillations, ear drum vibrations due to sound stimulation, cell membrane contractions and expansions have been accurately modeled and reconstructed. Single element simulations such as a soda can with an internal pressure load can also be modeled and observed with FAST4.

The source code is written in FORTRAN77 and compiled for the MAC and Windows operating systems. The accuracy of the linear result from FAST4 has been confirmed by comparisons with exact solutions of test examples. Further comparisons for more elaborate configurations have been made with BOSOR4, which is a finite difference program developed by D. Bushnell. BOSOR4 is a widely used program with an accuracy that depends on the defined mesh spacing. As such, in comparison with BOSOR or any of the many available finite element programs, FAST4 users find a substantial reduction in the preparation time for a given problem. FAST4 can be used for both preliminary design as well as validation of design.

1.2 Using FAST4 with MATLAB

The FAST4 program is an executable that processes an input file 'fstin'. The input file specifies the geometry, material properties, displacement and the load and generates an output text file 'fst4out'. The output file includes computed data on stress resultants, stress, strain, and displacement for each of the input files. The graphics program 'fst1graph' is a separate script which reads the output file 'fst4out' and has commands to allow visualization of the stress resultants, strain, and deformation of the structure in graphical form.

To facilitate ease of application of the FAST4 program, MATLAB functions were developed to work with the FAST4 input and output format to allow users to call the FAST4 program and immediately view the output in the desired graphical/movie format. The MATLAB functions must be located in the specified directory in MATLAB. The following section describes the functionality each MATLAB function and an example demonstrating the valid input parameters and the significance of the output parameters. A similar user interface has also been developed using Mathematica.

1.3 Description of MATLAB Functions

m-file	Function	Input parameters	Output Parameters
[cpmodel, cpdisp] = call_fast4('inputfile', 'outputfile')	Calls FAST4 program and reads results into MATLAB struct arrays	'inputfile' - a character string specifying the name of the input FAST4 data file 'outputfile' - a character string specifying the name of the FAST4 output file	[cpmodel, cpdisp] - cell struct arrays in MATLAB that contain the finite element model and displacement data read from the FAST4 output file
[fem,fedisp] = read_f4out('outputfile', gonly)	Reads a FAST4 output file and converts it to a FE model stored in struct arrays in MATLAB	'outputfile' - a character string array specifying the name of the FAST4 output file. If gonly=1 (default) only geometrical data (positions and displacements of the different elements of the finite element model) will be read. If gonly=0, the output contains also the material properties of the model, together with resultants, strain and stress data.	'fem' - A cell struct array containing information about the finite element model* fedisp - A cell struct array containing data about the displacements**
function res = plot_fem(fem, styl)	Generates a plot of a finite element model that was read from a FAST4 output file	'fem' - a cell struct array containing the FEM geometry data obtained from the outpt of the read_f4out function 'styl' - a string character defining the plot style (default '_')	Output res is the handle to the plot
Function cpmr = plot_cp(cpm, spix,	Plots a rotated CP model for		

siz, alphaBM, orig, iflag)	superposition to a confocal image by reading the geometry data from the FAST4 data file into a struct array that represents the CP model and applying rotation and rescaling to the CP coordinate data		
function M = femanim(fem, fdsp, maxamp, nstep)	Computes a movement of the FE model from geometry and displacement data	'fem' and 'fedsp' - cell struct arrays that contain the initial finite element model geometry and displacement data obtained from the output of read_f4out 'maxamp' - maximum amplitude weighing the displacements 'nstep' - number of steps in the animation	'M' - a movie array that is an input to the movie function
function fem2 = femove(fem1, disp, amp)	Computes a movement of the FE model from geometry and displacement data	'fem1' and 'disp - cell struct arrays containing the initial finite element geometry and displacement data obtained from the output file read from read_f4out 'amp' - amplitude factor weighing the displacements	'fem2' - cell struct array containing the deformed finite element model geometry

*Each cell element contains the following fields:

Elt	Element number
Type	Type of element
Npts	Number of points on the element
Geom.	nptpsx7-array containing the geometry data for the different points. The columns of this array correspond resp. to [s r z phi thick k1 k2]
prop	nptsx6-array containing the material properties of the points, with the following column organization: [s e et nu cother temp].

** Each element has the following fields:

elt, typ, npts	Same as above
Geom.	nptsx7 array containing the displacements themselves, with the following column organization: [s h v utan uth wnorm rot]
Res	nptsx7 array containing resultant data. Column organization [s Ms Ns Mth Nth Msth Nsth Qs Qth].
Strain	nptsx7 array containing strain data. Column organization as in fast4 output
stress	nptsx7 array containing stress data. Column organization as in fast4 output

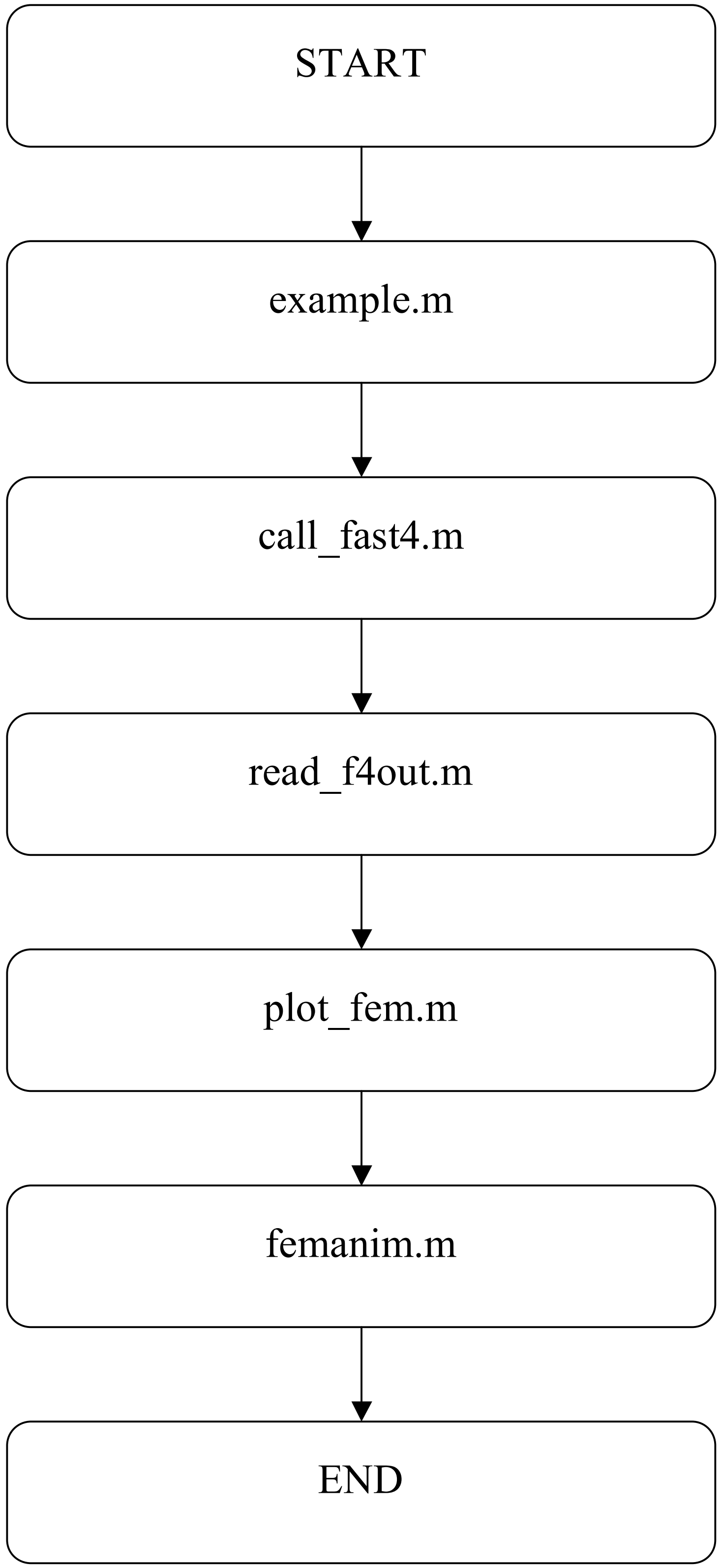

Figure 1.3.1 Matlab Function Flow chart

1.4 Setting Up MATLAB to Run FAST4 files

Having described the relevant MATLAB functions in the previous section, an example that utilizes the functions in an executable MATLAB m-file has been written to automatically read the FAST4 output file. The m-file will also automatically generate a plot of the modeled system as well as an animation of the system response to stimulation. An example of executing the above functions can be seen in the example.m file. This section will provide a step-by-step guide to setting up MATLAB directories to work with FAST4 and specifying relevant input and output file locations such that FAST4 can perform calculations on the input file and such that MATLAB can locate the output files to generate the relevant graphics.

To run FAST4 through MATLAB, the following steps need to be taken to ensure that MATLAB can locate the relevant files:

1. Change the home directory in MATLAB to the location of the folder Fast4Tool. See Figure 1.4.1.
2. Open call_fast4.m
3. In call_fast4.m, change the path of f4dir to the location of the folder Fast4Tool. See Figure 1.4.2.
4. Save call_fast4.m and close the file
5. Open example.m
6. Change the call_fast4 input parameters to the name of the input FAST4 file and the desired name of an output file.
7. Save example.m
8. Execute example.m

If the execution was successful, a graphical representation of the cochlea should be displayed and thereafter an animation is generated. The graphical representation is shown in Figure 1.5.4.

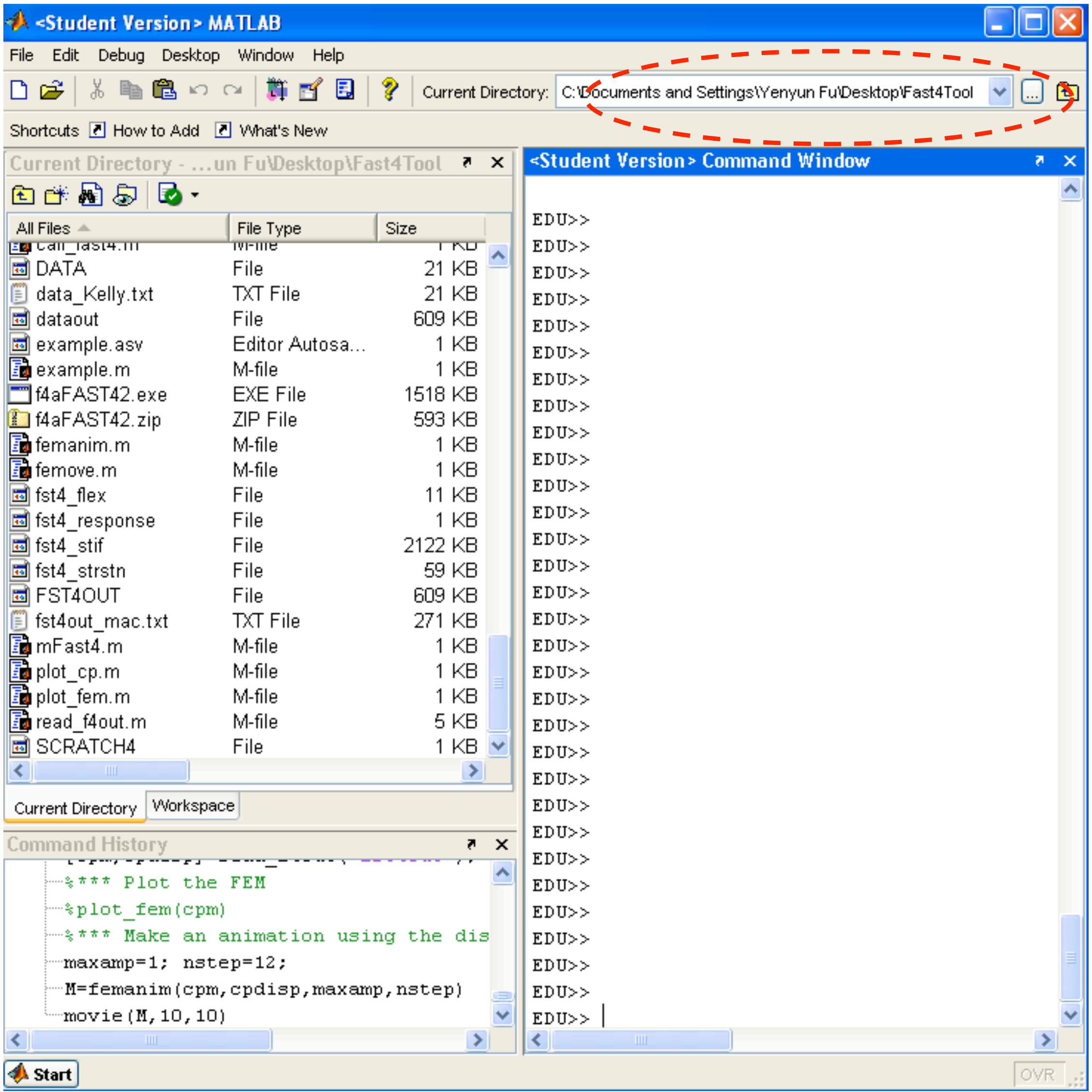

Figure 1.4.1 Setting the home directory in MATLAB

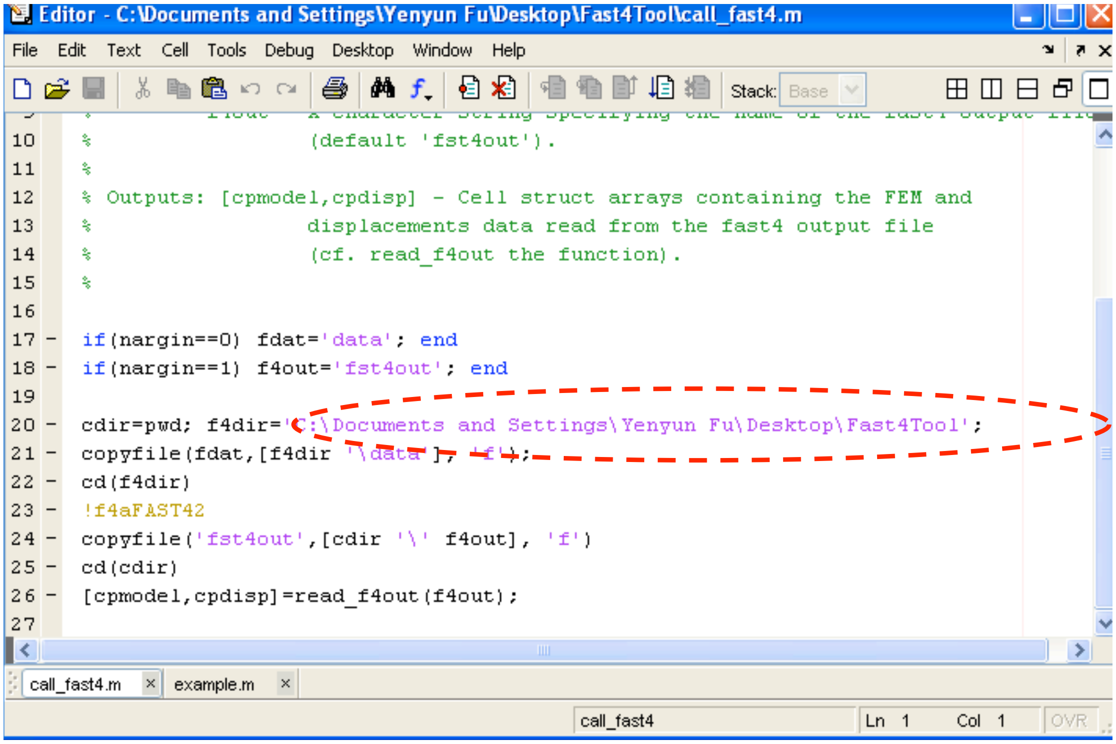

Figure 1.4.2 Setting the f4dir in call_fast4.m

1.5 Examples

1.5.1 Example 1 – Cylindrical vessel

In this example, a cylindrical vessel with hemispherical ends is considered. The pressure is acting on the exterior such that the distributed load in the normal direction acting on the OD designated by ‘dlnod’ is negative. When the membrane stress resultants are negative, a local buckling analysis is performed. The output variable ‘rho’ gives the ratio of stress resultant to that causing bifurcation instability. In this example, the value of ‘rho’ (circumferential) of -1.11 for the cylinder indicates that the bifurcation buckling is exceeded by 11% by the pressure of ‘dlnod’ = -200. The values of ‘rho’ for the spherical ends indicate that these are far from the stability limit for this pressure. The spherical shell with external pressure is, however, imperfection sensitive, so these values should be increased. In contrast, the cylinder with pressure is not imperfection sensitive so the ‘rho’ values do not require much empirical adjustment.

To run the example, enter the input file name ‘Example_cylinder.txt’ in example.m and follow the step-by-step instructions from the previous section. Upon successful execution, the output figure should be that shown in Figure 1.5.1.

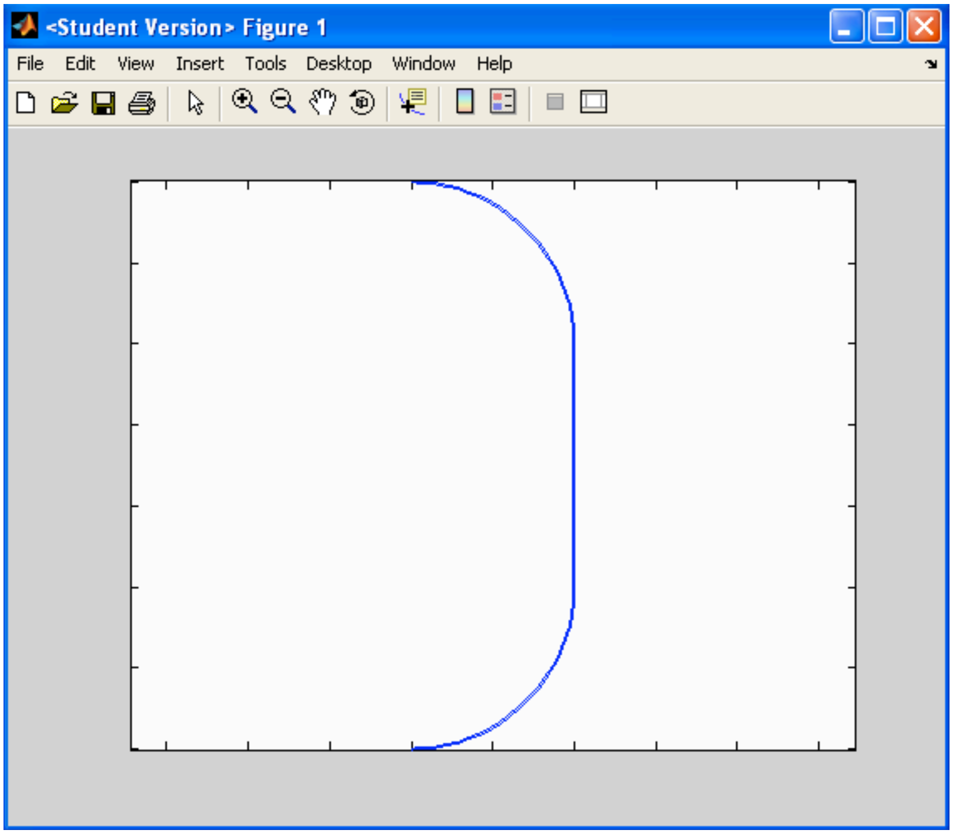

Figure 1.5.1 Graphical Output of Cylinder Example

1.5.2 Example 2 – Ear Drum

The ear drum consists of an organized collagen fiber structure with an angular placement in the ear canal. The cross section of the ear drum shows an umbo to tympanic annulus having an increasing slice thickness profile in the human ear. The overall eardrum thickness comprises of four layers. The outermost layer is the epithelial layer and the inner most layer is the submucosal layer. The inner two layers have collagen fibers that are oriented radially in one layer and circumferentially in the layer below. The inner layers provide scaffolding for the ear drum and thus predominantly determine the compliance of the membrane. The thickness variations and the layered structure of the ear drum is seen in Figure 1.5.2a below.

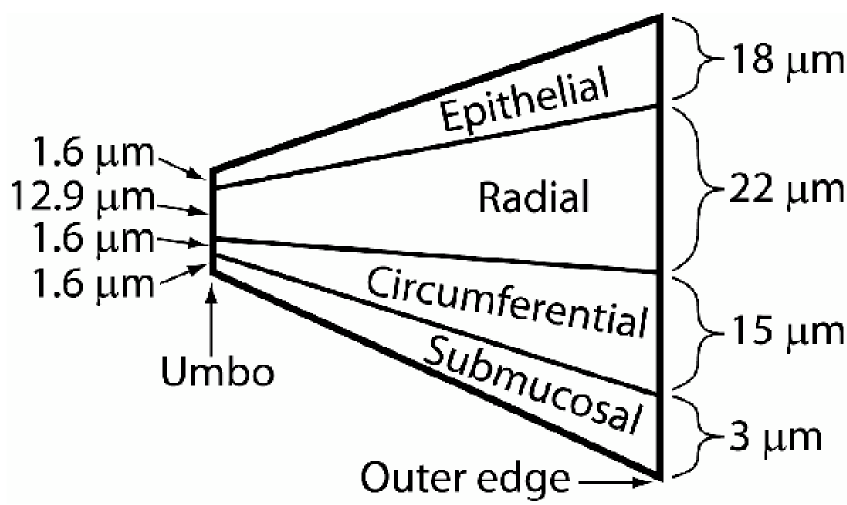

Figure 1.5.2a Cross section showing ear drum layers in humans

This example is a demonstration of modeling the ear drum with FAST4. The four-layer composite and varying thickness requires the geometrical data to be defined in multiple stations. Five conical region and five toroidal regions are used between the umbo and the tympanic annulus with each region having a thickness gradient. Higher precision may be obtained with more sections. Material properties for each segment is also specified individually taking into account thickness variations on material compliance. Execution of the file Example_TM should generate the following graphics of an ear drum slice.

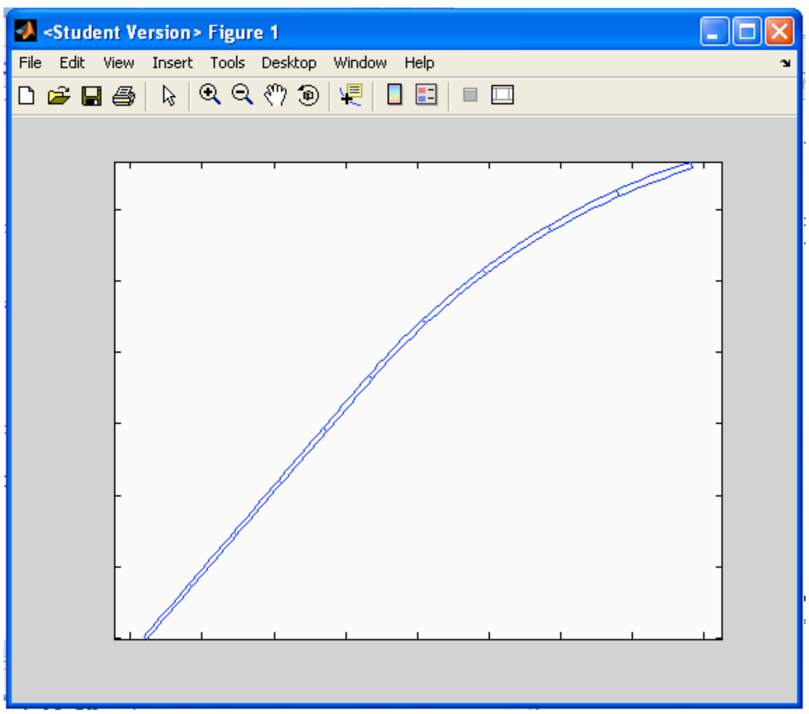

Figure 1.5.2b Graphical output showing one slice of a tympanic membrane

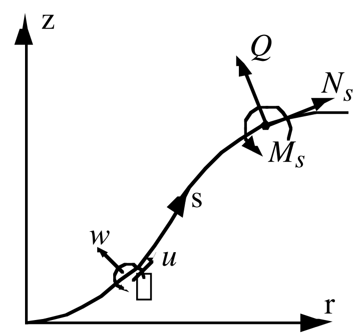

Figure 1.5.2c Graphical output showing one slice of a tympanic membrane

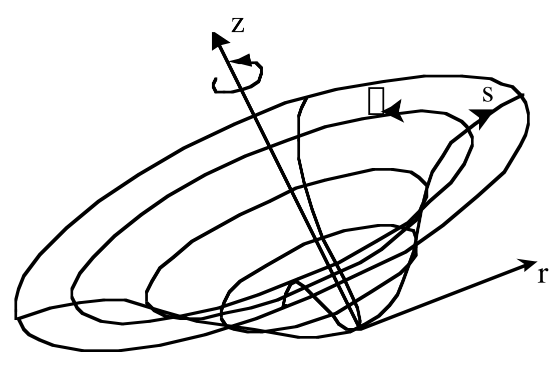

Figure 1.5.2d Axi-symmetric plot of the ear drum

1.5.3 Example 3 – Organ of Corti

Another example is a model of the cochlea which is a part of the inner ear. The cochlea is a snail-like structure divided into three fluid-filled chambers. Two chambers called the scala vestibuli and the scala tympani are for transmission of the acoustic energy to the third chamber which houses the sensitive organ of Corti that detects input pressure impulses and responds with electrical impulses which travel along the auditory nerve to the brain. The model includes details such as the primary and secondary spiral lamina, arcuate zone basilar membrane, pectinate zone basilar membrane, pillar cells, inner hair cell, recticular lamina, hensen cells, tectorial membrane, three rows of outer hair cells, cilia, deiter rods, and tip links. For an animation of the organ of Corti motion, run the file 'example_organofcorti.txt' as described in the previous sections. Upon successful execution, the motion of the organ of Corti will be simulated. Figure 1.5.4 shows the graphical output of the cochlea model as described.

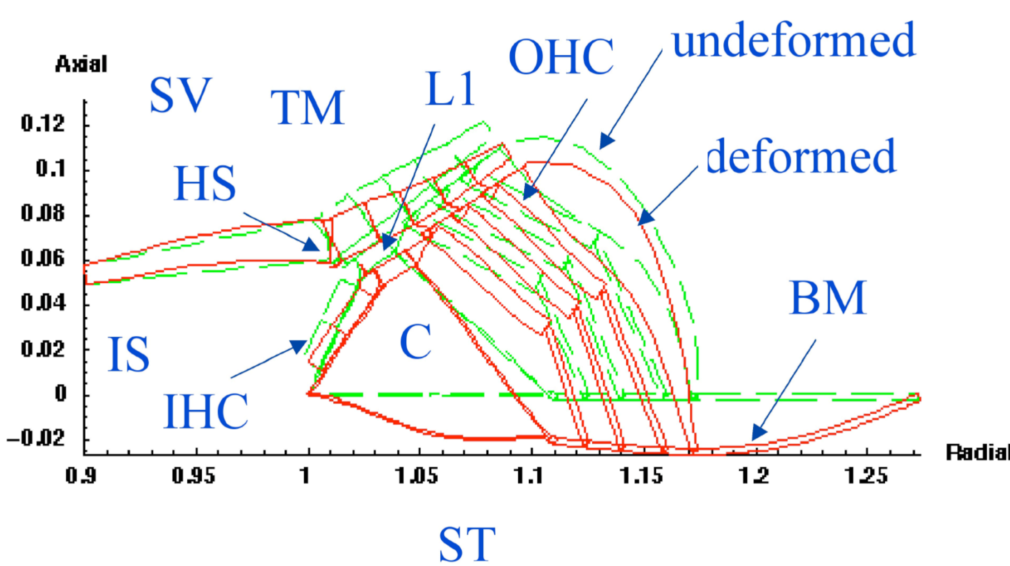

Figure 1.5.4 Graphical output of Organ of Corti

2.1 Writing the FAST4 Input file

2.1.1 Overview

In the FAST4 input file, an axi-symmetric system can be described by specifying each elemental shape that comprise a unique cross section of the axi-symmetric geometry. The cross section can be a combination of several elemental shapes having each starting point and connecting points specified. Each starting point forms a 'station' for each elemental shape that exists in the axi-symmetric system. The end point of the system is also the ending point of the outermost elemental shape. Thus, the last 'station' does not require an elemental shape to be specified. Thickness profiles for each elemental shape of the system are also specified with the geometric data associated with each station.

Material properties such as the Young's modulus, Poisson's ratio, elastic stiffness can also be specified in the FAST4 input file on a station by station basis for each elemental shape. Variations of material properties within the system and within each elemental shape can be specified. Physical stimulus that can be modeled on the system include line loads, line moments, distributed loads of a surface area/projected area, can be specified in various directions. The specific options with each type of force loading will be described in detail in the following sections. Furthermore, boundary conditions of each 'station' that connects each elemental shape can be specified in the FAST4 input file to fix axial, radial, and/or rotational displacements at boundary points of the system and intersection points of different elements. Discontinuities that may occur between connecting elements can also be described.

There is no limit to the number of comment lines that can be included in a FAST4 input file. A comment line is indicated by a letter ‘c’ in the first column of each comment line. The input file is initially read by a subroutine COMDEL and then written to the file 'datcom' with all the comment lines deleted. The following is a description of the format for the file 'datcom'. The original input file 'data' has the same code but may include the comment lines. The original input file is analyzed for consistency by COMDEL and error messages are written to an output file ‘fst4out’ and operation is halted upon discovery of a deficiency. It is important to note that line numbers in the error messages are references to line numbers in 'datcom' rather than in the original input file. The subroutine DATIN reads the geometry, properties, and loads from the file 'datcom'.

2.1.2 Input File Format

The first line of code that is not a comment must be the title of the simulation model. Following the title, the next set of lines of uncommented code includes the geometric specification of the axi-symmetric system. The geometric specification comprises multiple lines with each line specifying an individual element that makes up the cross section of the axi-symmetric system with each individual element having a different elemental shape. Each line of the geometric specification, with line number i includes the following parameters:

1. station number - i
2. station radial coordinate - R(i)
3. station axial coordinate - Z(i)

4. code for shell type TYPE(i) beginning at the station STA(i)

5. thickness of shell at beginning of element - TH1

6. thickness of shell at end of element - TH2 (may be omitted for constant thickness)

7. parameter for shell PARAM(i), if needed

For example, each line of the geometric specification may look like this:

i	R(i)	Z(i)	TYPE(i)	TH1	TH2	PARAM(i)
12	0.200E-04	0.000E+00	CYIND	0.320E-05		

The last entry of the geometric specification is the end point of the last element rather than the starting point of a new element. As such, the last line of the geometric specification should not have a TYPE indicator or a THICKNESS indicator. The end of the geometry specification is denoted with a line containing only '0' in column three after the last line of code that specifies the geometry. The geometric specification of the cylindrical vessel in example 1 having two end spherical regions and a tubular like center is shown:

```
c Geometry
c
c sta     r            z          type       param        th1
th2
c
   1  .000000e+00  .000000e+00 sphere              .1000e+01
   2  .100000e+03  .100000e+03 cylind              .1000e+01
   3  .100000e+03  .250000e+03 sphere              .1000e+01
   4  .000000e+00  .350000e+03 end
   0
c
```

The next set of un-commented code contains information regarding the material properties, loads, and boundary conditions. Each line contains:

1. Quantity type specifier
2. Magnitude of the quantity
3. Station where this quantity begins
4. Station where this quantity ends
5. Occurrence of station in geometry list

For example, each line of the geometric specification may look like this:

Quantity type	Magnitude	Starting station	Ending Station	Occurrence
e	0.370E09	1	2	

The starting and ending station entries may be omitted for constant value throughout the system. The material properties, external load specification and boundary conditions for the cylindrical vessel of example 1 may be specified as:

```
c Material properties
c
  e        .3000e+08
  et       .3000e+08
  nu       .3000e+00
c
c  Load: External pressure
c
  dlnod   -.2000e+03
c
c  Reference axial displacement set equal to zero at
sta = 3
c
```

2.1.3 Sign Conventions

```
z - Axis

 ^
 |          ←
 |             \       → Radial displacement -
 |              |         positive in r - direction
 |        \____/
 |                      ^
 |     Rotation -       |  Vertical displacement -
 |    positive for      |  positive in z - direction
 |    rotation from
 |   r – axis to z - axis
 |
 |                              The direction of increasing
 |    sta=93   id   sta=4        arc length s in one element
 |       ------------> s         is defined by the order of the
 |          | od                 stations in the geometric
 |          |                    input. Shown is a flat plate
 |          |                    beginning at sta=93 and ending
 |          |                    at sta=4.
 |          | n                  The positive outward normal is the
 |          |                    n-direction obtained by a -90 degree
 |          |                    rotation from the positive s-direction.
 |          \/
 |
 |    ^ s                    ^
 |    |                  od  | n
 |  id |→ n             s ←
 |    | od                  id
 |
 |         od | id
 |            |
 |        n ←|
 |            |
 |           \/ s
 |________________________________________>   r - Axis
```

The convention of positive rotation defined by the rotation from the r - axis to the z - axis holds whether positive r is chosen to the left or right, or whether positive z is chosen as up or down.

2.2 Description of Notations and Variables in FAST4

2.2.1 Shell Specifiers

Shell	Type	Parameter
Cone	cone	
Cylinder	Cylind	
Ellipsoid	Ellipe	Width/height
Hyperboloid	Hyperb	Width/height
Knuckle	Knuckl	Radius
Link	Link	
Paraboloid	Parab	
Plate	Plate	
Sphere	Sphere	
Toroid pos min	Torpmn	Radius
Toroid pos max	Torpmx	Radius
Toroid neg min	Tornmn	Radius
Toroid neg max	Tornmx	Radius

2.2.2 Description And Usage Of Shell Specifiers

Width/height – Ellipsoid	Ratio of the width to the height for the complete ellipse
Width/height – Hyperboloid	Slope of asymptote
Radius	Meriodional radius of curvature of toroids
Knuckle**	Toroidal element which can be inserted between two shells with only radius prescribed.
Link	Element that is rigid in the meridional direction and with zero circumferential stiffness
Torpmn	Toroidal element with a meridian having a circular arc in the positive rotation direction with minimum arc length
Tornmn	Toroidal element with a meridian having a circular arc in the negative rotation direction with minimum arc length
Torpmx***	Toroidal element with a meridian having a circular arc in the positive rotation direction with maximum arc length
Tornmx***	Toroidal element with a meridian having a circular arc in the negative rotation direction with maximum arc length

** The endpoints of a knuckle are computed to provide continuous meridional slope. Since this can be used to approximate a fillet, the input value of radius is the critical fillet radius, which, for example, is on the od of the intersection of a small cylindrical pipe and a spherical shell, or on the id of the intersection of a shallow spherical head on a cylindrical vessel. The meridional radius of curvature is then computed, appears in the output table of geometry, and is used for the subsequent calculations.

***WARNING: Torpmx and Tornmx are not recommended at this time. The best accuracy is obtained by using torpmn and tornmn with extra stations at the points where phi = 0 or pi.

2.2.3 Specifiers For Material Properties

Material Property	Specifier
Young's Modulus	E
Poisson's Ratio	Nu
Young's modulus in transverse direction	et
Elasti-stiffness of spring attachment at station with components constraining: moment per unit length (positive rot direction) per radian	Ke11
Elasti-stiffness of spring attachment at station with components constraining: radial force per unit length (outward)per unit displacement	Ke22
Elasti-stiffness of spring attachment at station with components constraining: coupling moment and radial force	Ke12
Elasti-stiffness of spring attachment at station with components constraining: axial force per unit length per unit displacement	Ke33
Mass per unit volume of shell	Mv
Weight per unit volume of shell	Wv
Weight per unit volume of fluid	Wvf
Coefficient of thermal expansion	Cother

2.2.4 Load Specifiers

Load	Specifier
Line load in positive radial direction	llr
Line load in positive axial direction	Llz
Line moment in direction of positive rotation	Lm
Distributed load per unite surface area in normal direction - Pressure on inner surface	Dlnid
Distributed load per unit surface area in normal direction - Pressure on outer surface	Dlnod
Distributed load per unit surface area in tangential direction	Dlt
Distributed load per unit surface area in radial direction	Dlr
Distributed load per unit surface area in axial direction	dlz
Distributed load per unit of projected area in axial direction (snow load)	Dls
Axial coordinate of fluid free surface (fluid in region z<zf)	Zf
Temperature	temp
number of "g's" of axial acceleration (in positive z-direction)	ng
Number of revolutions per minute of spinning about axis	rpm

2.2.5 Displacement Specifers

Displacement	Specifier
Discontinuity in radial displacement	Deldr
Discontinuity in axial displacement	Deldz
Discontinuity in rotation	Delrt
Radial displacement	Disr
Axial displacement	Disz
Rotation	disrt

2.2.6 Other Options

VARIATION IN PROPERTIES WITHIN AN ELEMENT

A variation in the properties e, et, nu, th, temp, cother can be prescribed within an element at a set of points, which may number from 2 to mmax. A cubicurve (lagrangian interpolation) is used to obtain the values at intermediate points. In the input file the key is the line with the parameter 'variat' set and the station number in the form previously discussed. This is followed by a formatted table of values for the quantities:

% distance e et nu th temp cother

If a column is blank, the previously defined constant value of the quantity is used. The loads may also be prescribed by the keyword 'varild' followed by a formatted table of values for the quantities:

% distance dlnid dlnod dlt dlr dlz dls

A four-point Lagrangian interpolation is used for values between the prescribed points. The results will be accurate for "smooth" distributions. For discontinuities in properties or loads, additional stations should be introduced such that the distribution is smooth between stations.

RERUN WITH PREVIOUS ELEMENT STIFFNESS

The computation will be repeated for a new load and/or a new constraint condition without recomputing the element stiffness when the parameter 'oldstf 'appears in the input file. With 'oldstf', the constraints and loads may be changed in the input file, but changing the geometry or material properties will result in nonsense.

RERUN WITH PREVIOUS CONSTRAINTS

The computation will be repeated for a new load without recomputing the element stiffness and without recomputing the constraint stiffness terms when the parameter 'oldcon' appears in the input file 'data'. With 'oldcon', oldstf is also assumed, and the loads in input file may be changed. However note that changing the geometry, material properties or constraints will result in nonsense.

LINEAR SOLUTION

The linear solution is computed when update does not appear in the input file ', in which case the program variable 'update' is set equal to 0. The geometry is computed from the coordinates of the stations given in the input file. Zero pre-stress is used. The output is written to the output file and consists of the following information:

1. Initial geometry
2. Displacement data
3. Stress resultants
4. Strain on od and id
5. Stress on od and id