

#### FE Modeling Example Using ADINA

Invoke the ADINA user interface (AUI) by typing:

> cd /tmp
> add adina
> aui7.3

and press the ADINA-IN button in the ADINA system window.

#### Defining model control data

*Master degrees of freedom*: From the ADINA-IN menu, choose Control->Degrees of Freedom and pull out the X-Translation, X-Rotation, Y-Rotation, and Z-Rotation buttons and press OK. Note that in the ADINA system, 2D problems are entered in the YZ-plane.

The Degrees of Freedom window should look like Figure 2:

Master Degrees of Freedom:
🔟 X-Translation 📕 Y-Translation 📕 Z-Translation
X-Rotation Y-Rotation Z-Rotation
☐ Fluid Potential Degree of Freedom
Ovalization/Warping DOF at Pipe Element Nodes: None =
Default Number of DOF Associated with Shell Midsurface Nodes:
OK Reset Cancel Help

Figure 2

# **Defining Model Geometry**

Gener	Load	Cie	ar Der Row	Ins Row	
	Label Number	X1	X2	X3	System
	1				
	2			10.0	
	3		10.0	I	
Displa	y in Global Carte	sian Coordinate S	ystem		

From the menu, choose Geometry->Points... and enter the values in Figure 3:



Empty fields are taken as zero by ADINA.

From the menu, choose Geometry->Surfaces->Define, press Add... and press OK in the popup box. Enter the following values in the Vertices table and press OK:

Add Delete Copy Select with Pointer
Surface Number: 1
Surface Type: Vertex =
Vertices:
Point 1: 1
Point 2: 2
Point 3: 3
Point 4: 3
Optional Edge Numbers:
Edge 1:
Edge 2:
Edge 3:
OK Cancel Help

Figure 4

Note that we entered the same node for the third and fourth corners of the surface to create a triangle.

<u>F</u> ile	<u>E</u> dit	View	<u>D</u> isplay	<u>C</u> ontrol	Geometry	<u>A</u> DINA-M	Model	FE- <u>R</u> ej	presentation	List	H	elp
Analys	sis Type		s	tatics	-	No. ' 🗆 I	SI <b>Issors</b>	: 1	ADINA-	IN [	ADINA	-
AD	INA	TIME R	1.000								Z Y	
Reger	nerati	on su	ccessful	ly compl	eted.							-1-
				<b>F</b> -								7

At this point, the ADINA-IN window should look like Figure 5:

Figure 5

# **Defining and Applying Boundary Conditions**

From menu, choose Model->Boundary Conditions->Fixity->Apply (or click on Apply Fixity button in the Modeling toolbox), and enter the following data and press OK:

Apply Fixity Editor:
Applied to:
Save Discard
Default Fixity: ALL Define
Generate Clear Del Row Ins Row
P Line Fixity
3 ALL
I



In cases where you don't know the line or node number, you can click on the P button and then click on the line or point of interest in the ADINA-IN window to find out its number.

To check the boundary conditions, from the menu choose Display->Show Boundary Conditions->All. The ADINA-IN window should look like Figure 7:

<u>F</u> ile	Edit	View	Display	<u>C</u> ontrol	Geometry	<u>A</u> DINA-M	Model	FE- <u>R</u> ep	resentation	List	<u>H</u> elp
Analys	is Typ	e:	S	tatics	-	No. 1	FSI <sup>ISSOPS</sup>	: 1	ADINA-	IN 🔤	adina 🖃
A	DIN	JA .	TJME	1.000						z	Г
		ć				В				U B -	2 U <sub>3</sub>
Meshj	olot M	ESHPL	OTOOOO1 :	successf	ully creat	ed.					ב[]ح

Figure 7

#### **Defining and Applying Loads**

To apply the gravitational body force, from the menu, choose Model->Loading->Apply (or click on the Apply Load button in the Modeling toolbox). Add load application number 1 as in Figure 8 (press Add... and and then press OK in the popup window). Choose Mass Proportional for the Load Type, define the Load Label as shown in Figure 9 and make sure that Site Type reads Model. Then press OK.

Add   Delete   Copy     Save   Discard
Load application Number: 1
Loading:
Load Case:
Load Type: Mass Proportional =
Load Label: 1 Define
Deformation Dependent: Default =
Pressure Load Specification: Total (Normal)
Application Site:
Site Type: Model 🖃
Site Label: D P C X
Body Label: 0 P
Auxiliary Point: 0
OK Cancel Help
Figure 8
Add   Delete   Copy     Save   Discard

Jave Discalu
Loading/Mass-Proportional Number: 1
Load: 9.81
Direction: X: 0.0
Y: 0.0
Z:1.0
Apply
OK Cancel Help

Figure 9

Since the water pressure changes with height, we first need to define a spatial function by choosing Geometry->Spatial Functions->Line, add Line Function Number 1, choose Type of data variation as Linear, and enter the values shown in Figure 10.

Add Delete Copy	
Save Discard	
Line Function Number: 1	
Values at Specific Points:	
at $u = 0$ , 1	
at u = 0.5.	
at u = 1, 0	
Number of Points: 3	
Function Values:	
Generate Load Write	
í Value	
OK Cancel Help	

Figure 10

Now we can apply the water pressure by choosing Model->Loading->Apply (or click on the Apply Load button in the Modeling toolbox), add Load application number 2 as shown in Figure 11, choose Load Type as Pressure, define Load Label 1 as shown in Figure 12, define Site Type as Line, Site Label as 1, and Spatial Data Variation Label as 1. Then press OK.

Add Delete Copy
Save Discard
Load application Number: 2
Luau Type: Pressure
Load Label: 1 Define
Deformation Dependent: Default =
Pressure Load Specification: Total (Normal)
Application Site:
Site Type: Line 🔤
Site Label: 1
Body Label: 0 P
Auxiliary Point: 0 P
Line Load: In Plane
Spatial Variation:
Spatial Data Variation Label: 1
OK Cancel Help

Figure 11

Add Delete Copy
Save Discard
Loading/Pressure Number:
Pressure: 98.1
Apply
OK Cancel Help

Figure 12

To display the water pressure, from the menu choose Display->Load Plot->Use Default (or click on the Load Plot button in the General toolbox). The ADINA-IN plot window should look like Figure 13.



Figure 13

# **Defining the Material**

Choose Model->Materials->Elastic->Isotropic, add Elastic Isotropic Material Number 1 and enter the Elastic modulus, Poisson's ratio, and density of the material as shown in Fig.14 and press OK.

Add         Delete         Copy           Save         Discard
Elastic Isotropic Material Number: 1
Young's modulus, E : 2800000.0
Poisson's ratio, v: 0.2
Density, p : 2.4
Coef. of Thermal Expansion (Beam elements only), $\alpha$ :
OK Cancel Help

Figure 14

### **Defining the Elements**

Choose FE-Representation->Element Groups->Define, add Element Group Number 1, select Element Type as 2-D Solid, Type of 2D Solid Element as Plane Strain, and press OK (Figure 15).

Add Delete Copy				
Save Discard				
Element Group Number: 1				
Element Type: 2-D Solid =				
Type of 2D Solid Element: Plane Strain				
Auxiliary Point Defined with: 🔶 Point 💠 Note				
Auxiliary Point/Node: 0 P				
Kinematic Formulation:				
Displacements: Default = Strains: Default =				
Updated Lagrangian Formulation: Default				
Default Material: 1				
Numerical Integration Order: Default =				
Degenerated 8-noded Elements are Spatially Isotropic				
Interpolation Formulation:				
Type: Default				
Pressure DOF:				
OK Cancel Help				

Figure 15

To generate meshes, choose FE-Representation->Mesh->Density->Surface, and select Mode of Subdivision on Lines as Use Number of Divisions. For this example, a 2x2 mesh density is used. Finally enter Surface Label as 1 and press OK (Figure 16).

Copy     Select with Pointer       Save     Discard   Default
Number: 1
Number of Subdivisions in u-direction:       2         Number of Subdivisions in v-direction:       2         Length Ratio of Edges in u-direction (Last/First):       1.0         Length Ratio of Element Edges in v-direction (Last/First):       1.0         Progression:       Geometric         Apply to the Following Surfaces:       1
Generate Load Write Clear Del Row Ins Row P Surface Label I I I
OK Cancel Help



To plot the mesh, choose FE-Representation->Mesh->Create->Surface, choose Number of Nodes per Element and Preferred Cell Shape (3 node triangular element is selected for this example), enter Surface Label as 1, and press OK (Figure 17).

Substructure: 0
Notal Specifications:
Number of Nodes per Element: 3 =
Pattern: Automatic =
Preferred Cell Shape: Triangular =
Tyme of Mash Ganaration : A Bula_Based Manning A Erga_Form Mash
KI
Generate Clear Del Row Ins Row
P Surface Label
I
<u></u>
OK Apply Cancel Help

Figure 17



After mesh generation and plotting, the ADINA-IN window looks like in Figure 18.



#### Generating the ADINA-IN Data File

From the menu, choose File->Datafile, enter example.dat for the data file name, and press OK.

### Saving the ADINA-IN Database

From the menu, choose File->Close, and enter example.idb for the file name and press OK.

#### **Running ADINA**

When ADINA-IN window is closed, chose ADINA from the ADINA system window. From the menu, choose Job->Start and select example.dat data file. When the analysis is complete, choose File->Close and return to ADINA system window.

#### **Plotting the Deformed Shape**

Choose ADINA-PLOT from the ADINA system window. From menu, choose File->Load Porthole, and select example.port file. To plot the deformed shape choose Display->Geometry/Mesh Plot->Use Default. To adjust the magnification and to plot the original shape, choose Display->Geometry/Mesh Plot->Modify. Press the Model Depiction button, press the "Display the Original Mesh" button, press the Length button in the Displacement Display Options box and set the Max. Displacement Length to 5,

Add   Delete   Copy     Save   Discard				
Model Depiction Name: MESHPL0T00001				
Model Display Options:				
Display the Geometry				
Display the Original Mesh				
Display the Deformed Mesh				
Displacement Display Options:				
Specified by: 💠 Magnification 🔶 Length				
Magnification Factor: 1.0				
Max. Displacement Length:				
5 Unit: PERCENT =				
OK Cancel Help				

Figure 19

then press OK twice to close both dialog boxes.



The graphics window should now look like Figure 20.

Figure 20

# Listing the tip deflection

Choose List-Extreme Values->Zone. In the lower half of the dialog box, use the scroll bar to display the Variables box. For Variable 1, choose Y-Displacement from the right hand drop-down window list. Then press Apply. Displacement of the tip will be listed in the top window as shown in Figure 21.

ADINA: AUI version Licensed from ADI Finite element pro Scanning for abso. Variable Y-DISPLA 2.20797E-04, time **** End of list.	n 7.2.2, 1 November NA R&D, Inc. ogram ADINA, respons Lute maximum in zone JEMENT: e 1.00000E+00, node	1999: *** NO HEADING se range type load-ste # WHOLE_MODEL: 3	DEFINED ***
Zone Name: WHOLE,	_MODEL	▼	
Response Range:	ULT DEFAULT		
Variables:			
📕 Variable 1:	Displacement 🗖	Y-DISPLACEMENT	V
Variable 2:	Displacement 🗖	A-DI SPLACEMENT	<b>V</b>
Variable 3:	Displacement 🗖	A-DI SALACEMENT	
Apply	Write	Cancel	Help

Figure 21