Topic 21

A Demonstrative Computer Session Using ADINA—Linear Analysis

Contents:

- **Use of the computer program ADINA for finite element analysis, discussion of data preparation, program solution, and display of results**
- **Capabilities of ADINA**
- **Computer laboratory demonstration—Part I**
- **Linear analysis of a plate with a hole for the stress concentration factor**
- **Data input preparation and mesh generation**
- **Solution of the model**
- **Study and evaluation of results using plots of stresses, stress jumps, and pressure bands**

Textbook:

Appendix

References:

The use of the ADINA program is described and sample solutions are given in

Bathe, K. J., "Finite Elements in CAD — and ADINA," *Nuclear Engineering and Design,* to appear.

ADINA, ADINAT, ADINA-IN, and ADINA-PLOT Users Manuals, ADINA Verification Manual, and ADINA Theory and Modeling Guide, ADINA Engineering, Inc., Watertown, MA 02172, U.S.A.

Proceedings of the ADINA Conferences, (Bathe, K. J., ed.)
Computers & Structures
13, 5–6, 1981
17, 5–6, 1983
21, 1–2, 1985

References:
(continued)

The use of pressure band plots to evaluate meshes is discussed in

Sussman, T., and K. J. Bathe, "Studies of Finite Element Procedures—Stress Band Plots and the Evaluation of Finite Element Meshes," *Engineering Computations*, to appear.

A FINITE ELEMENT ANALYSIS — LINEAR SOLUTION

- We have presented a considerable amount of theory and example solution results in the lectures.
- The objective in the next two lectures is to show how an actual finite element analysis is performed on the computer.

Transparency 21-1

- We cannot discuss in detail all the aspects of the analysis, but shall summarize and demonstrate on the computer the major steps of the analysis, and concentrate on
  - — possible difficulties
  - — possible pitfalls
  - — general recommendations

Transparency 21-2

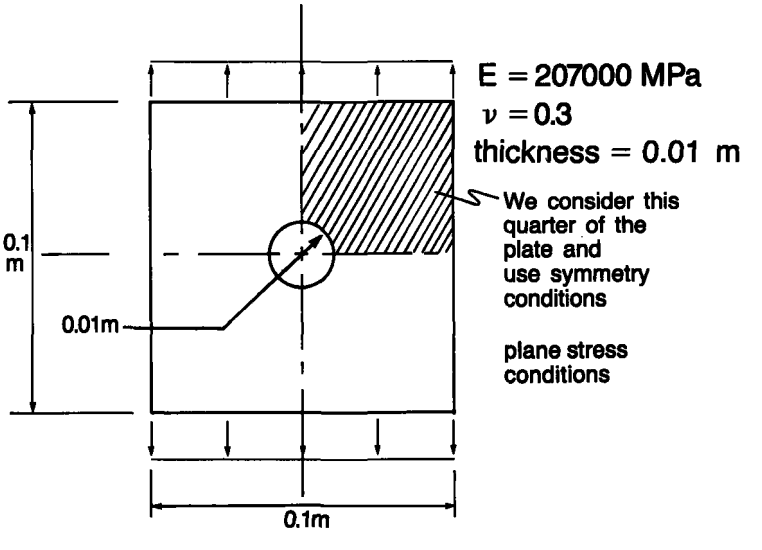
Transparency 21-3

We will use as the example problem the plate with a hole already considered earlier, and perform linear and nonlinear analyses

— elastic analysis to obtain the stress concentration factor
— elasto-plastic analysis to estimate the limit load
— an analysis to investigate the effect of a shaft in the plate hole

Transparency 21-4

Plate with hole: Schematic drawing

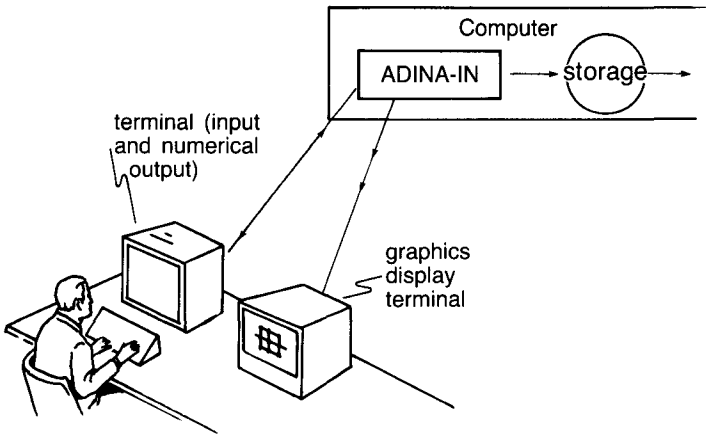


- The first step for a finite element analysis is to select a computer program. We use the ADINA system.

ADINA-IN	to prepare, generate the finite element data
ADINA	to solve the finite element model
ADINA-PLOT	to display numerically or graphically the solution results

Transparency 21-5

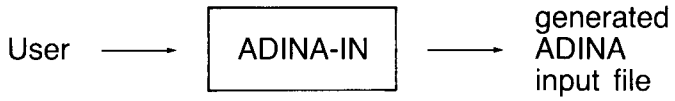
Schematically:



User work-station

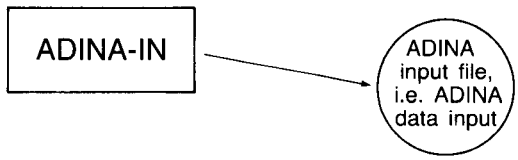
Transparency 21-6

Transparency 21-7

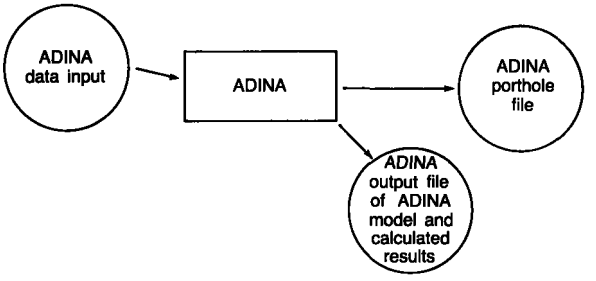


- User types into terminal ADINA-IN commands interactively or for batch mode processing. User checks input and generated data on graphics display terminal.

Transparency 21-8

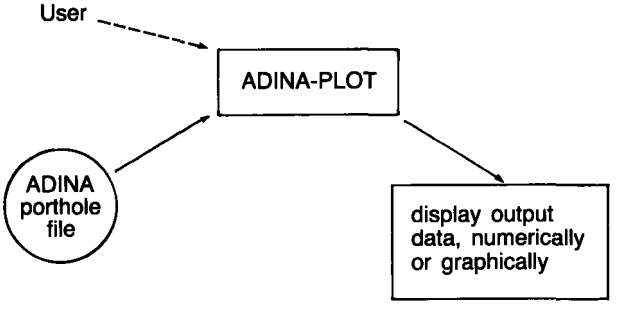


- ADINA-IN generates the input data for ADINA.
- The input data is checked internally in ADINA-IN for errors and consistency and is displayed as per request by the user.
- The degree of freedom numbers are generated (for a minimum bandwidth).



- User runs ADINA to calculate the response of the finite element model. ADINA writes the model data and calculated results on an output file and stores the model data and calculated results on the porthole file.

Transparency 21-9



- User runs ADINA-PLOT to access the output data and display selected results; displacements, stresses, mode shapes, maxima, . . .

Transparency 21-10

Transparency 21-11

A brief overview of ADINA

- Static and dynamic solutions
- Linear and nonlinear analysis
- Small and very large finite element models can be solved.

The formulations, finite elements and numerical procedures used in the program have largely been discussed in this course.

Transparency 21-12

DISPLACEMENT ASSUMPTIONS

- Infinitesimally small displacements
- Large displacements/large rotations but small strains
- Large deformations/large strains

MATERIAL MODELS

Isotropic Linear Elastic

Orthotropic Linear Elastic

Isotropic Thermo-Elastic

Curve Description Model for Analysis of Geological Materials

Concrete Model

Transparency 21-13

MATERIAL MODELS

Isothermal Plasticity Models

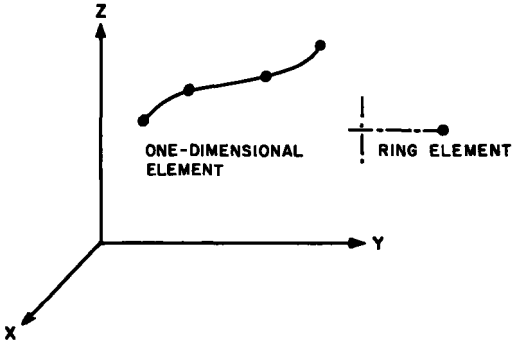
Thermo-Elastic-Plastic and Creep Models

Nonlinear Elastic, Incompressible Models

User-Supplied Models

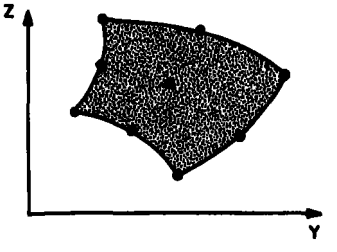
Transparency 21-14

Transparency 21-15

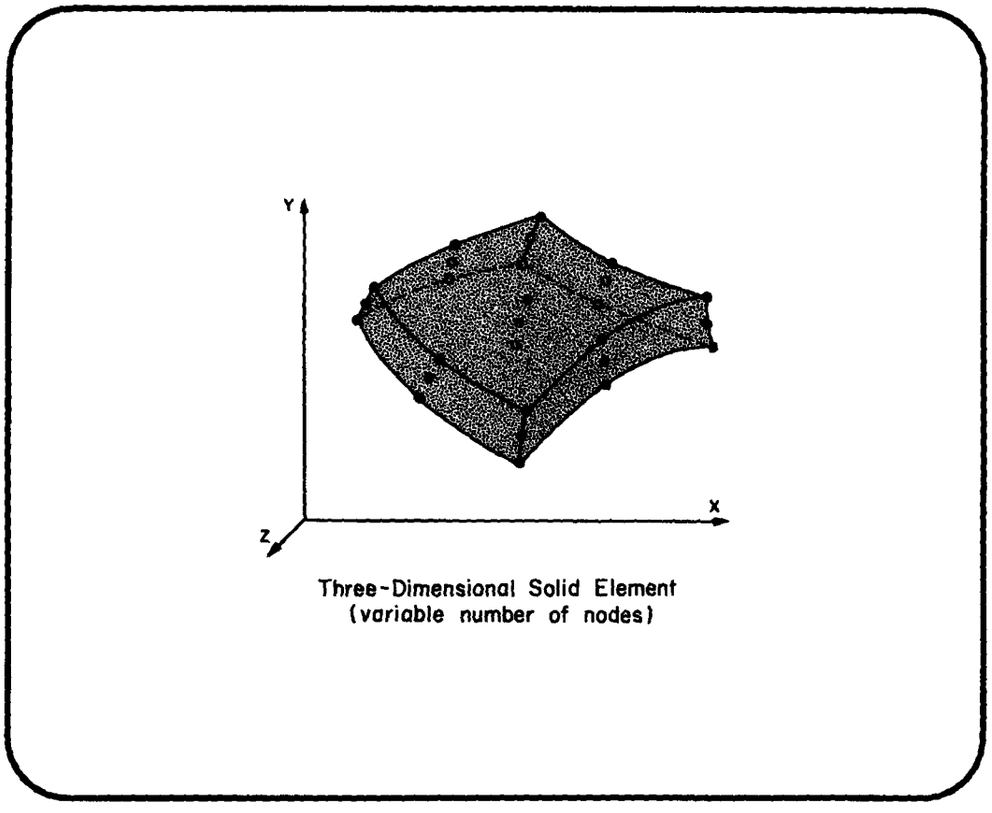


Truss and Cable Element
(2, 3, or 4 nodes)

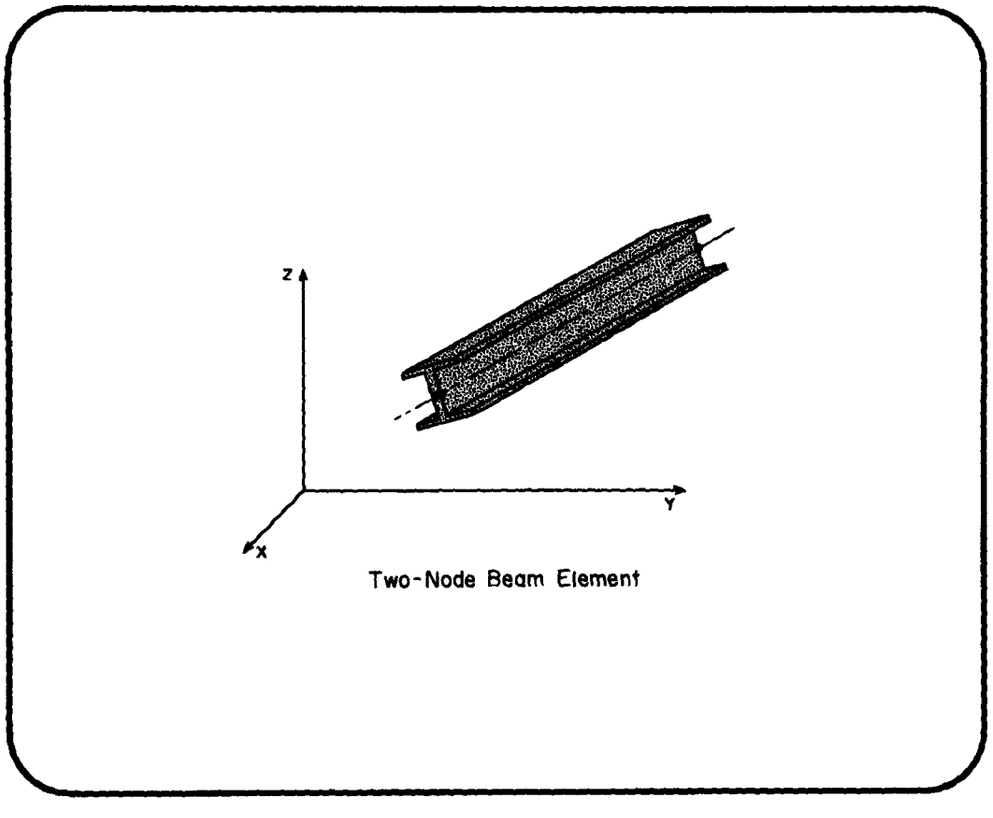
Transparency 21-16



Two-Dimensional Solid Element
(variable number of nodes)

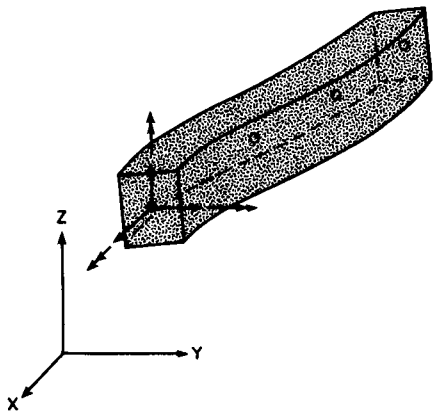


Transparency 21-17



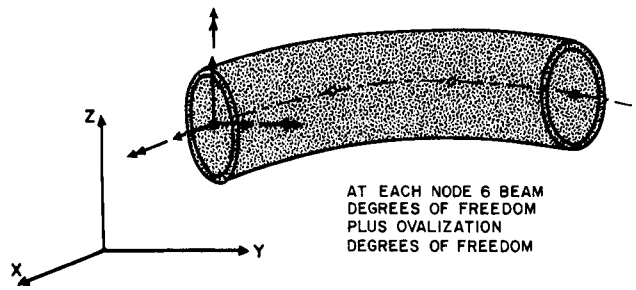
Transparency 21-18

Transparency 21-19

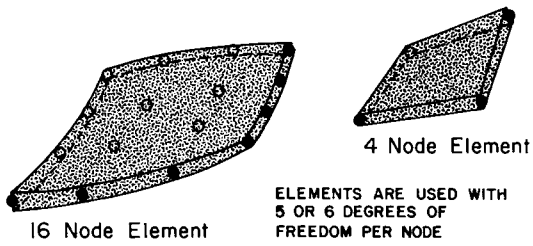


Isoparametric Beam Element
(2, 3, 4 nodes)

Transparency 21-20



Pipe Element with Ovalization



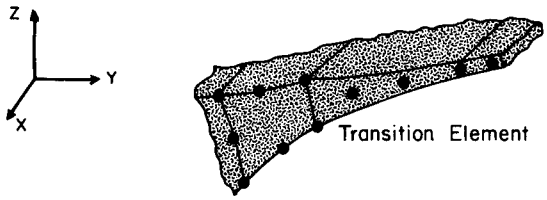


Plate and Shell Elements

Transparency 21-21

A SUMMARY OF IMPORTANT OBSERVATIONS

- We need to check the finite element data input carefully
  - — prior to the actual response solution run, and
  - — after the response solution has been obtained by studying whether the desired boundary conditions are satisfied, whether the displacement and stress solution is reasonable (for the desired analysis).

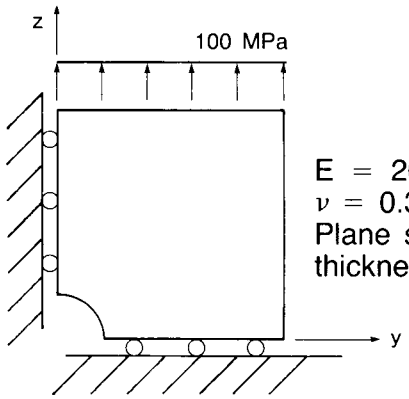
Transparency 21-22

Transparency 21-23

- We need to carefully evaluate and interpret the calculated response
  - — study in detail the calculated displacements and stresses along certain lines, study stress jumps
  - — stress averaging, stress smoothing should only be done after the above careful evaluation

Transparency 21-24

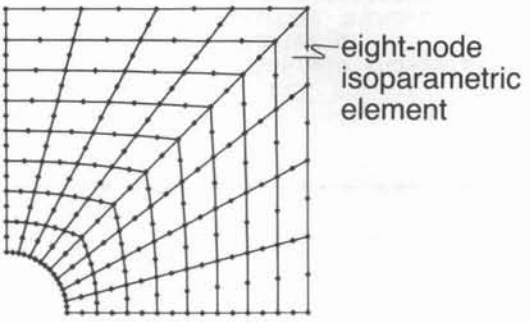
Data for Construction of 64 Element Mesh:



E = 207,000 MPa
ν = 0.3
Plane stress
thickness = 0.01 m

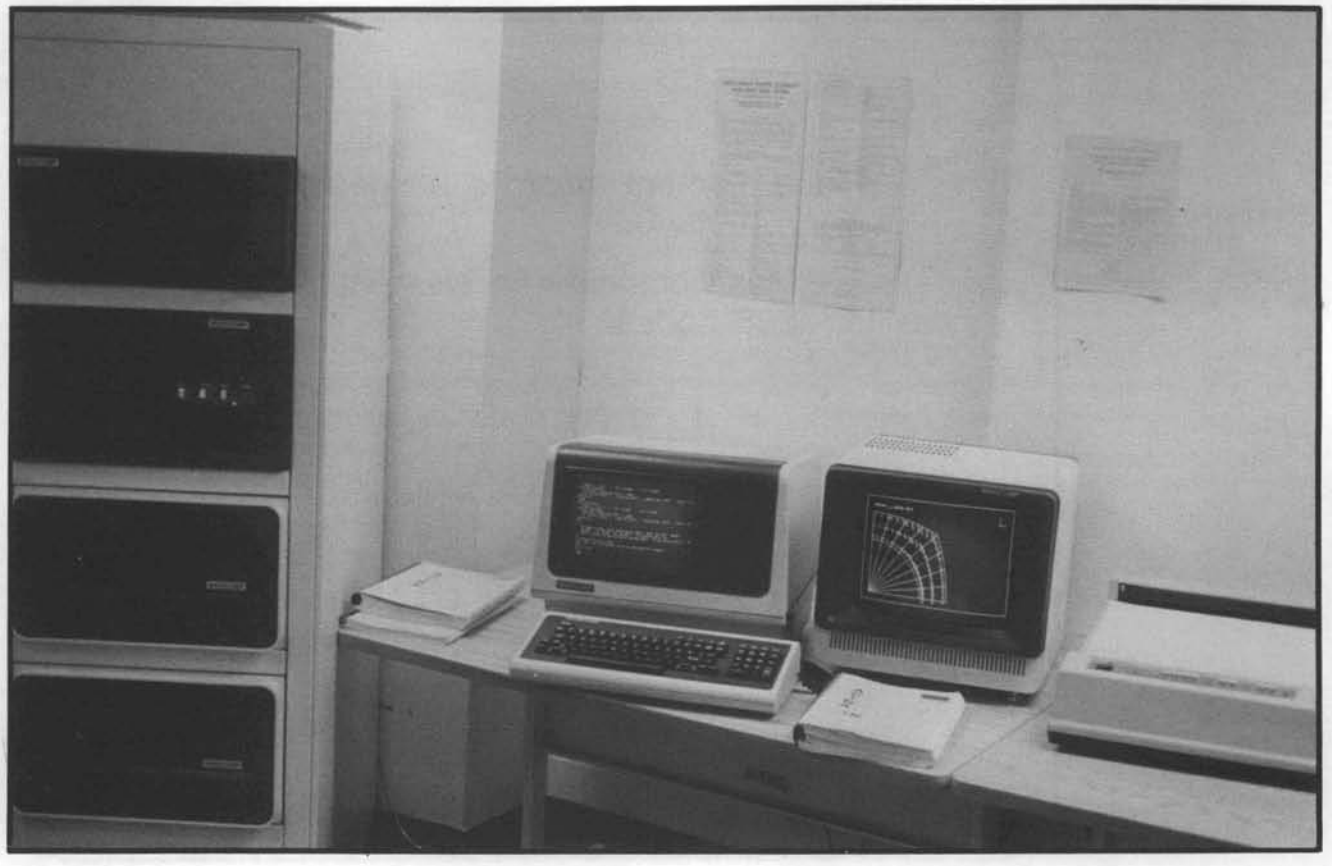
Finite element mesh to be generated using ADINA-IN:

- Mesh contains 64 elements, 288 nodes.



Transparency 21-25

Demonstration Photograph 21-1
Finite Element Research Group Laboratory computer configuration



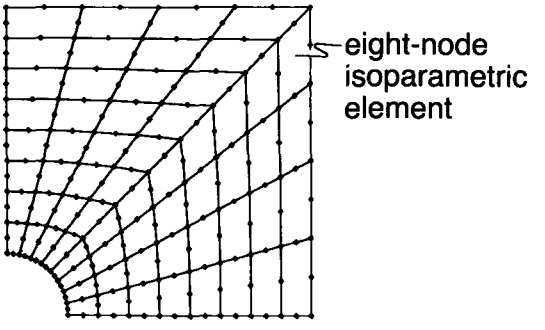
ADINA Demonstration 21-1
Input data

```
QUARTER PLATE WITH HOLE - 64 ELEMENTS
  2261001110  1    0    1    1 1.0000000
C***  MASTER CONTROL
99999    0    0    1    0    0    1   50 30.
C***  3  LOAD CONTROL
    0    4    0    0    0    0    0    0
C***  4  MASS AND DAMPING CONTROL
    0    0    0    0        .0        .0
C***  5  EIGENVALUE SOLUTION CONTROL
    0    0    0    0    0    0
C***  6  TIME INTEGRATION METHOD CONTROL
    0   20.500000000.25000000    0    0
C***  7  INCREMENTAL SOLUTION CONTROL
    1    1    210 15.001000000.010000000.05
C***  8  PRINT-OUT CONTROL
    1    1    1    1    1    1    0
```

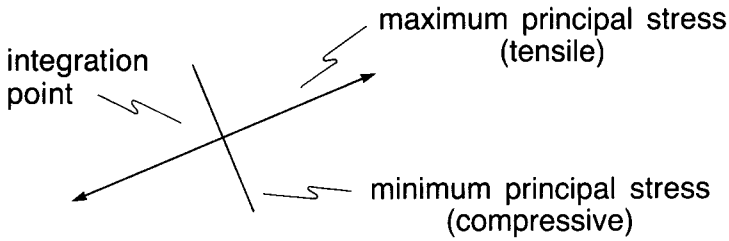
Transparency 21-26
(Repeat 21-25)

Finite element mesh to be generated using ADINA-IN:

- Mesh contains 64 elements, 288 nodes.

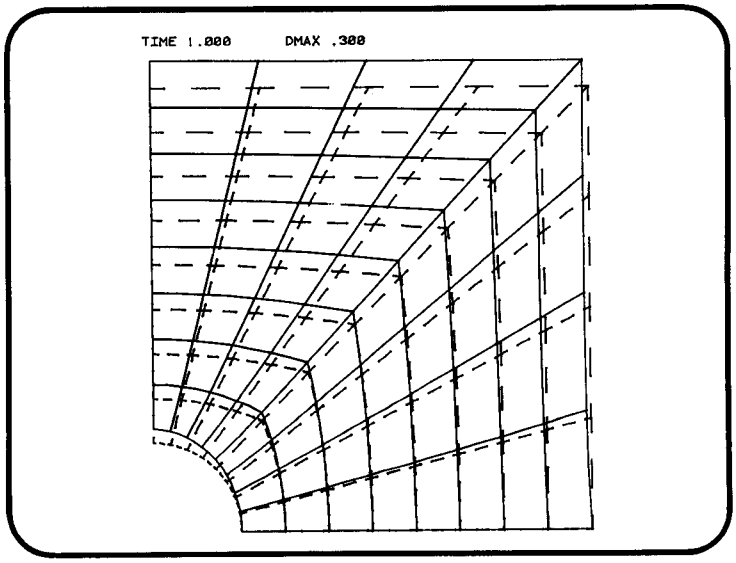


Stress vector output: Example



The length of the line is proportional to the magnitude of the stress.

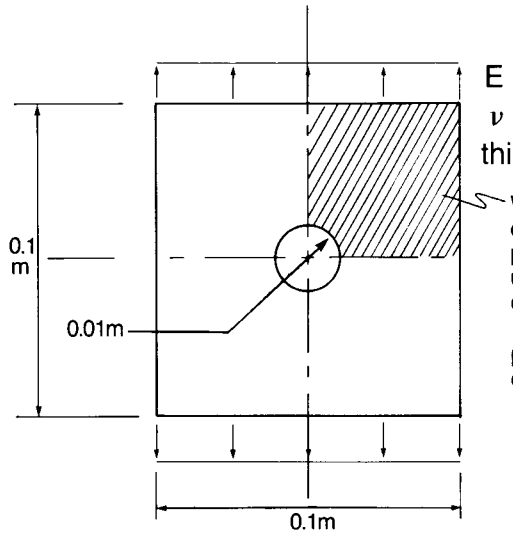
Transparency 21-27



ADINA Demonstration 21-2
Deformed mesh plot

Transparency 21-28

Plate with hole: Schematic drawing



E = 207000 MPa
ν = 0.3
thickness = 0.01 m

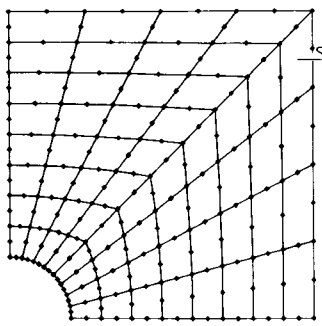
We consider this quarter of the plate and use symmetry conditions

plane stress conditions

Transparency 21-29
(Repeat 21-25)

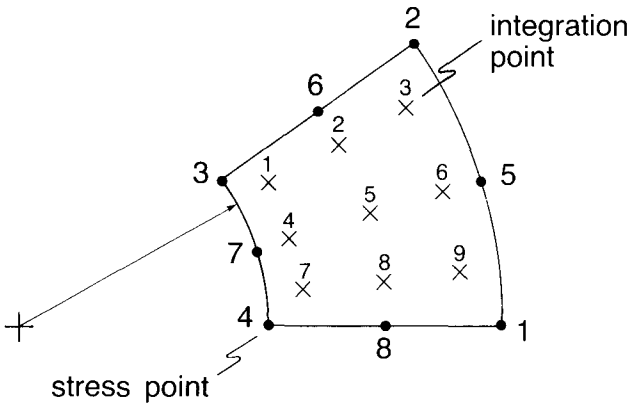
Finite element mesh to be generated using ADINA-IN:

- Mesh contains 64 elements, 288 nodes.



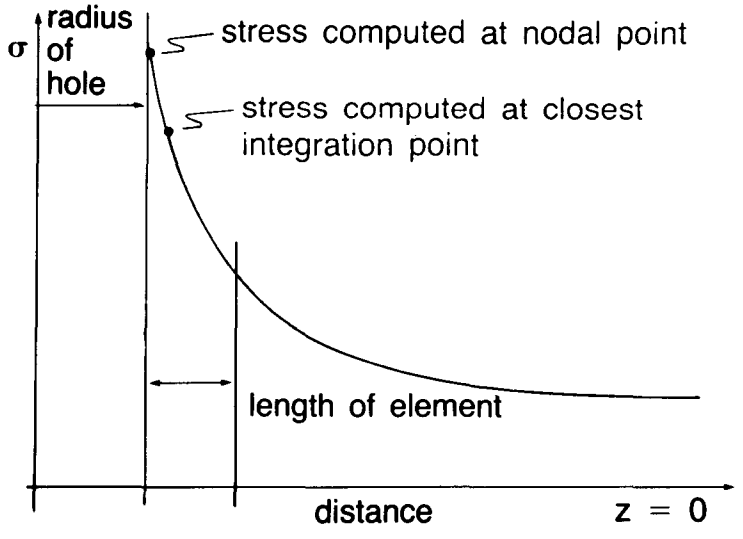
eight-node isoparametric element

Stress point numbers and integration point numbers for element 57



Transparency 21-30

Behavior of stresses near the stress concentration:



Transparency 21-31

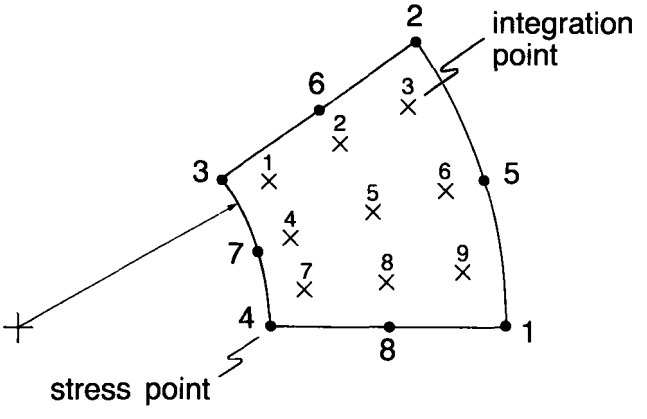
Transparency 21-32

Maximum principal stress calculation:

$$\sigma_1 = \frac{\sigma_{yy} + \sigma_{zz}}{2} + \sqrt{\frac{(\sigma_{yy} - \sigma_{zz})^2}{4} + \sigma_{yz}^2}$$

Transparency 21-33
(Repeat 21-30)

Stress point numbers and integration point numbers for element 57

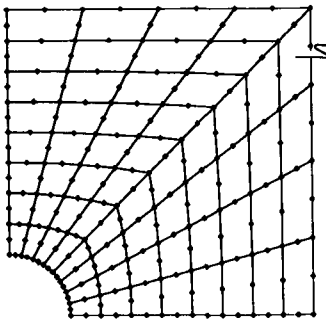


```
   RESULTANT = SMAX      ARITHMETIC EXPRESSION:

   (TYY+TZZ)/TWO+SQRT((TYY-TZZ)*(TYY-TZZ)/FOUR+TYZ*TYZ)

   TYY      = YY-STRESS
   TZZ      = ZZ-STRESS
   TYZ      = YZ-STRESS
   TWO      =  2.00000
   FOUR     =  4.00000

EXTREME ELEMENT RESULTS PER ELEMENT GROUP FOR WHOLE MODEL

 INTERVAL  TSTART= 1.0000      TEND= 1.0000      SCANNED FOR ABSOLUTE MAXIMUM

ELEMENT GROUP NO =   1  (2-D SOLID)          LISTED RESULTS ARE MEASURED IN
                                             GLOBAL COORDINATE SYSTEM
     RESULTANT SMAX        ELEMENT  POINT       TIME       STEP

   0.345151E+03              57       4    0.10000E+01      1
```

ADINA Demonstration 21-3
Close-up of calculations

Finite element mesh to be generated using ADINA-IN:

- Mesh contains 64 elements, 288 nodes.



Transparency 21-34
(Repeat 21-25)

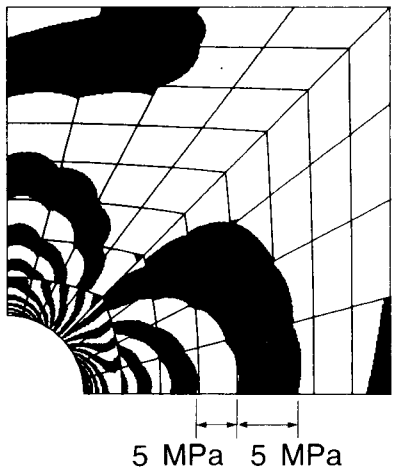
Transparency 21-35
(Repeat 2-33)

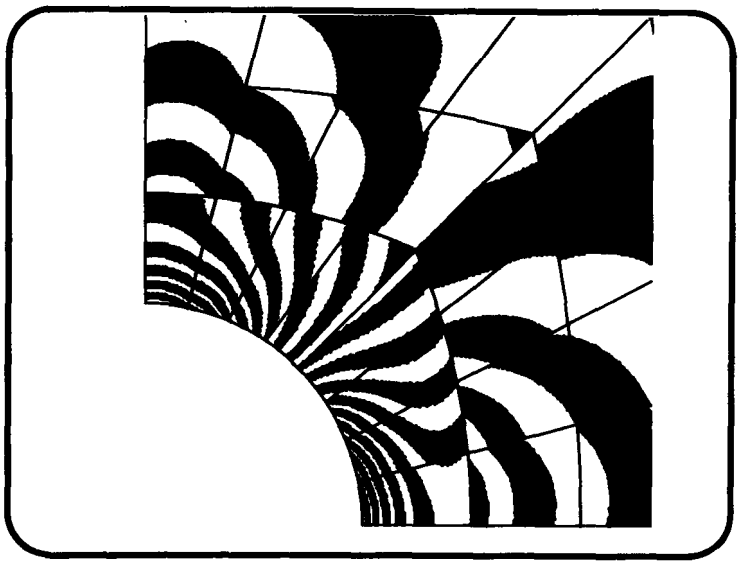
- To be confident that the stress discontinuities are small everywhere, we should plot stress jumps along each line in the mesh.
- An alternative way of presenting stress discontinuities is by means of a pressure band plot:
  — Plot bands of constant pressure where

$$\text{pressure} = \frac{-(\tau_{xx} + \tau_{yy} + \tau_{zz})}{3}$$

Transparency 21-36
(Repeat 2-35)

Sixty-four element mesh: Pressure band plot





ADINA Demonstration 21-4
Close-up of pressure bands

A SUMMARY OF IMPORTANT OBSERVATIONS

- We need to check the finite element data input carefully
  - — prior to the actual response solution run, and
  - — after the response solution has been obtained by studying whether the desired boundary conditions are satisfied, whether the displacement and stress solution is reasonable (for the desired analysis).

Transparency 21-37
(Repeat 21-22)

Transparency 21-38
(Repeat 21-23)

- We need to carefully evaluate and interpret the calculated response
  - — study in detail the calculated displacements and stresses along certain lines, study stress jumps
  - — stress averaging, stress smoothing should only be done after the above careful evaluation

Resource: Finite Element Procedures for Solids and Structures
Klaus-Jürgen Bathe