MAE 4700/5700: ANSYS Section

Fridays 1:25-2:15 pm

Rajesh Bhaskaran

Cornell University

Co-ordinates

- Dr. Rajesh Bhaskaran
  Swanson Director of Engineering Simulation
  Mechanical & Aerospace Engineering
- E-mail: bhaskaran@cornell.edu
- Office: 102 Rhodes Hall
- Office hours in the Swanson Lab (163 Rhodes)
  - TBA

Computer Labs with ANSYS

- CIT public labs
  - B7 Upson
  - 318 Phillips
  - ACCEL lab in Carpenter Hall
- 471 Rhodes
- Swanson Lab (163 Rhodes)
  - 16 workstations
  - 2 quad-core processors
  - 30 GB of RAM

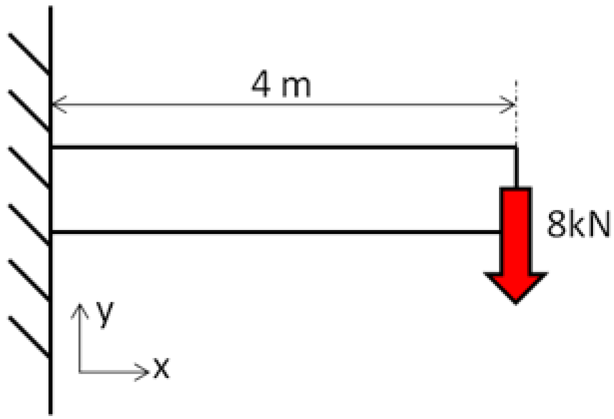
ANSYS Software

- Leading commercial FEA software
- Founded by Cornell alum Dr. John Swanson in 1970
- Can solve structural, thermal, flow and electro-magnetic problems
- Student version available for $25/year
  - Instructions to be provided by e-mail
  - Version 13
    - Labs are using Version 14. V14 files cannot be read into V13.

Friday Sections

- Purpose:
  - Learn to *apply* FEA to engineering problems using ANSYS
  - Prepare for project
- Plan:
  - Initially solve some HW problems using ANSYS
    - Compare MATLAB and ANSYS solutions
  - Move on to more complex problems.

ANSYS Exercise 1
Cantilever Beam



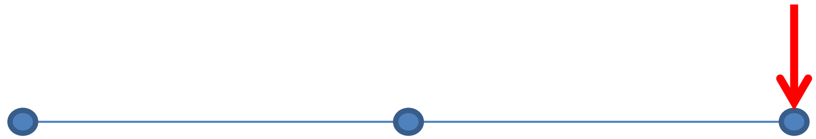
- Truss elements are available in ANSYS
  - Need to use scripting (advanced feature)
  - Not used widely in practice

ANSYS Exercise 1
Cantilever Beam

- Beams will appear in HW3
- One problem will be on ANSYS solution of cantilever beam
  - Save work from this section for submission with HW3

Cantilever Beam: Degrees of Freedom

- Consider 2-element mesh



$u_{y_1} = 0$ $\quad$ u_{y_2} $\quad$ u_{y_3}

$\theta_{z_1} = 0$ $\quad$ θ_{z_2} $\quad$ θ_{z_3}

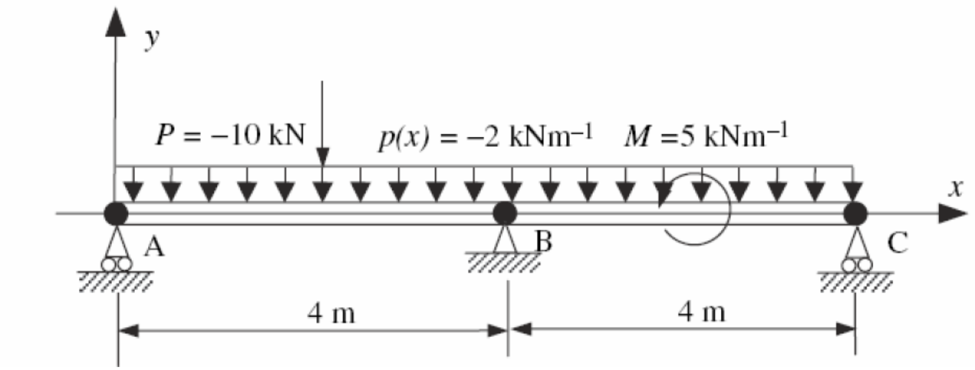
Reaction force at node 1
Reaction moment at node 1

SECTION MEETING #2
9/7/2012

Cantilever Beam

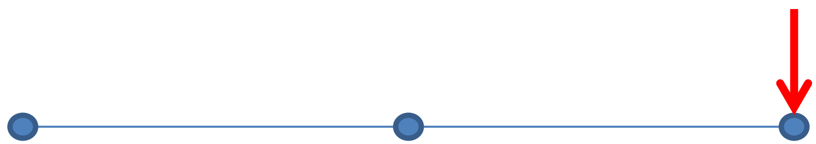
- First ANSYS exercise
  - Can do trusses but need to use scripting (advanced functionality)
  - Pin-jointed trusses rarely occur in practice
- ANSYS beam problem will appear in HW3

Problem 3 – Analysis of a two-span beam (MatLab and Ansys)



Cantilever Beam: Degrees of Freedom

- Consider 2-element mesh

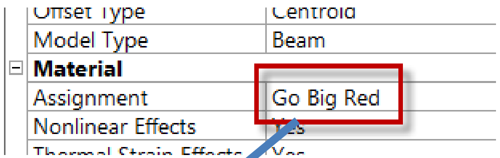


$u_{y_1} = 0$ $\quad$ u_{y_2} $\quad$ u_{y_3}

$\theta_{z_1} = 0$ $\quad$ θ_{z_2} $\quad$ θ_{z_3}

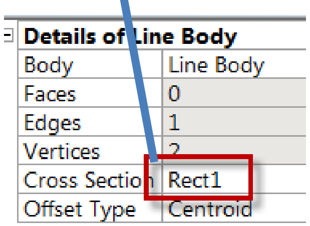
Reaction force at node 1
Reaction moment at node 1

Cantilever Beam
Beam element stiffness matrix



K^e : see page 24 of *Lecture 4: Finite element formulation of beam problems*

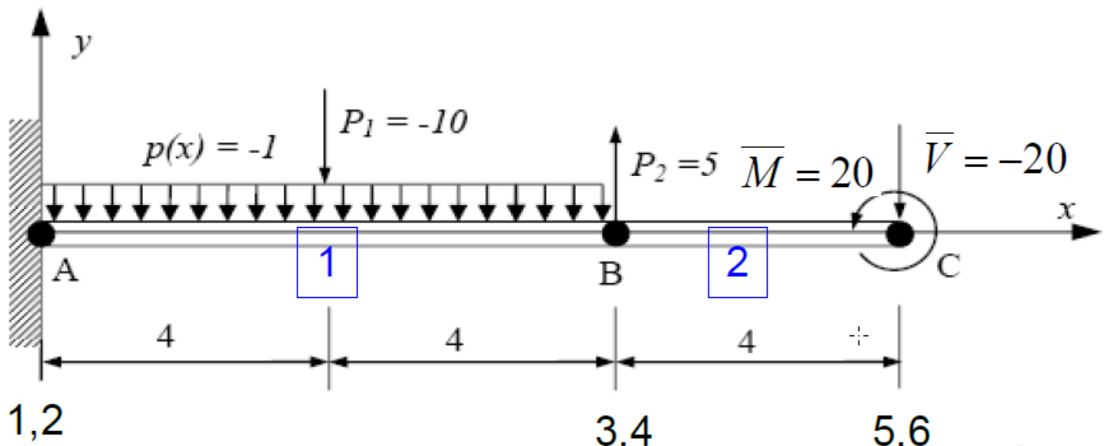
$$K^e = \frac{E^e I^e}{2L^e}\int_{-1}^{1}\begin{bmatrix} 36\xi^2 & 6\xi(3\xi-1)L^e & -36\xi^2 & 6\xi(3\xi+1)L^e \\ 6\xi(3\xi-1)L^e & (3\xi-1)^2 L^{e^2} & -6\xi(3\xi-1)L^e & (9\xi^2-1)L^{e^2} \\ -36\xi^2 & -6\xi(3\xi-1)L^e & 36\xi^2 & -6\xi(3\xi+1)L^e \\ 6\xi(3\xi+1)L & (9\xi^2-1)L^{e^2} & -6\xi(3\xi+1)L^e & (3\xi+1)^2 L^{e^2} \end{bmatrix} d\xi = \frac{E^e I^e}{L^{e^3}}\begin{bmatrix} 12 & 6L^e & -12 & 6L^e \\ 6L^e & 4L^{e^2} & -6L^e & 2L^{e^2} \\ -12 & -6L^e & 12 & -6L^e \\ 6L^e & 2L^{e^2} & -6L^e & 4L^{e^2} \end{bmatrix}$$



Poisson ratio doesn't appear in *K^e*

Cornell University

Solve Step
Beam Lecture, Page 43

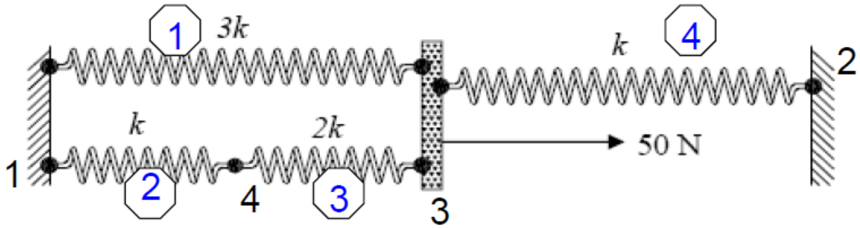


$$
10^3 \begin{bmatrix} 0.23 & 0.94 & -0.23 & 0.94 & 0 & 0 \\ 0.94 & 5.00 & -0.94 & 2.50 & 0 & 0 \\ -0.23 & -0.94 & 2.11 & 2.81 & -1.88 & 3.75 \\ 0.94 & 2.50 & 2.81 & 15.00 & -3.75 & 5.00 \\ 0 & 0 & -1.88 & -3.75 & 1.88 & -3.75 \\ 0 & 0 & 3.75 & 5.00 & -3.75 & 10.00 \end{bmatrix} \begin{Bmatrix} u_{y1}=0 \\ \theta_{y1}=0 \\ u_{y2} \\ \theta_{y2} \\ u_{y3} \\ \theta_{y3} \end{Bmatrix} = \begin{Bmatrix} -9+R_{u1} \\ -15.3+R_{\vartheta 1} \\ -4 \\ 15.3 \\ -20 \\ 20 \end{Bmatrix}
$$

Bending Moment and Shear Force

- General FEM Procedure (see next slide):
  1. Calculate unknown degrees of freedom
  2. Calculate reactions at known degrees of freedom (for instance, at fixed nodes)
- ANSYS then uses reactions to calculate bending moment and shear force:
  – More accurate than differentiating displacement → $M^1 = EI \frac{d^2 u_y^1}{dx^2}$

Calculation of Reactions



See Lecture 2: Direct approach, page 32

$$k\left[\begin{array}{cc|cc} 4 & 0 & -3 & -1 \\ 0 & 1 & -1 & 0 \\ \hline -3 & -1 & 6 & -2 \\ -1 & 0 & -2 & 3 \end{array}\right]\begin{Bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \end{Bmatrix} = \begin{Bmatrix} r_1 \\ r_2 \\ 50N \\ 0 \end{Bmatrix}$$

We partition and apply BCs: $d_1 = d_2 = 0$

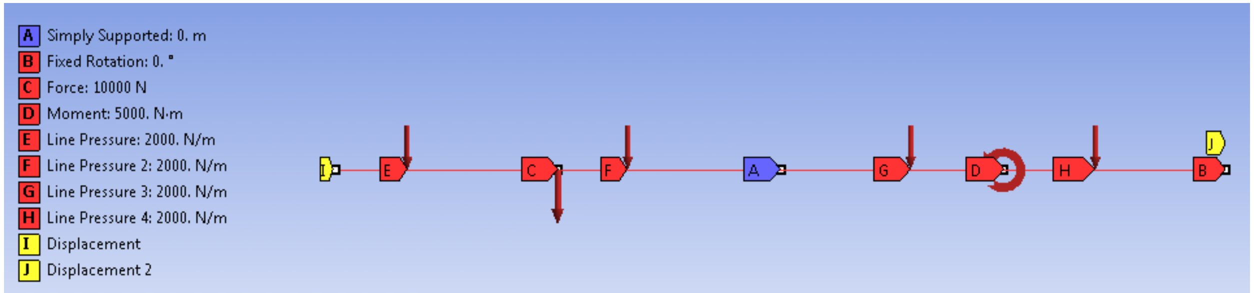
$$k\begin{bmatrix} 6 & -2 \\ -2 & 3 \end{bmatrix}\begin{Bmatrix} d_3 \\ d_4 \end{Bmatrix} = \begin{Bmatrix} 50N \\ 0 \end{Bmatrix} \Rightarrow \begin{Bmatrix} d_3 \\ d_4 \end{Bmatrix} = \frac{1}{k}\begin{Bmatrix} 10.7143 \\ 7.1429 \end{Bmatrix} N$$

HW3 Tips

- Model the geometry using four lines
- Apply distributed load using *Line Pressure*
- *Simply Supported* boundary condition *sets* u_x and u_y to zero but leaves θ_z free
- *Frictionless support* in this case will set u_y to zero but leaves u_x and θ_z free

HW3 Tips

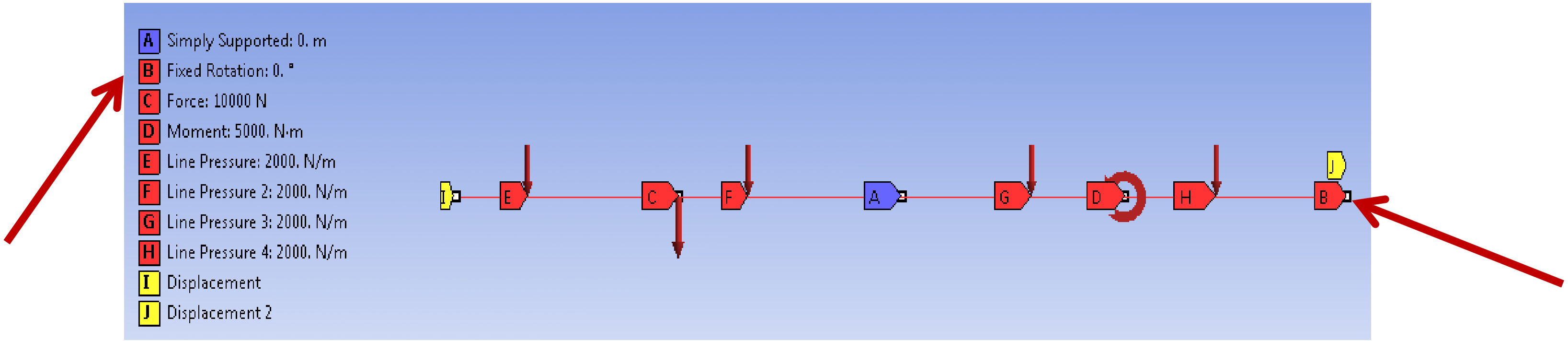
- You have to add a BC by fixing the rotations about x and y axes as in the snapshot below. Otherwise you might get a solver pivot error. This is because ANSYS is using a 3D beam element with these additional rotations as dof's.



SECTION MEETING #4
9/21/2012

HW3 Comments

- Need additional BC for beam problem (#2)
  - Fix the rotations about x and y axes
- Otherwise might get a solver pivot error
  - ANSYS is using a 3D beam element formulation with these additional rotations as dof's.



HW Comments

- Degrees of freedom in ANSYS' beam element

$$\begin{Bmatrix} u_x \\ u_y \\ \cancel{u_z} \\ \cancel{\theta_x} \\ \cancel{\theta_y} \\ \theta_z \end{Bmatrix}$$

Additional constraint ensures that these DOF's are identically zero

Otherwise might get rigid body motion (structure flying off)

HW3 Comments

- You generally apply loads to geometry
  - Transferred to nodes and elements during *Solve*
- Saving files
  - *File > Save*
    You need **both** *frame.wbpj* file and *frame_files* folder to restore project
  - *File > Archive*
    ANSYS saves entire project in one file *frame.wbpz*

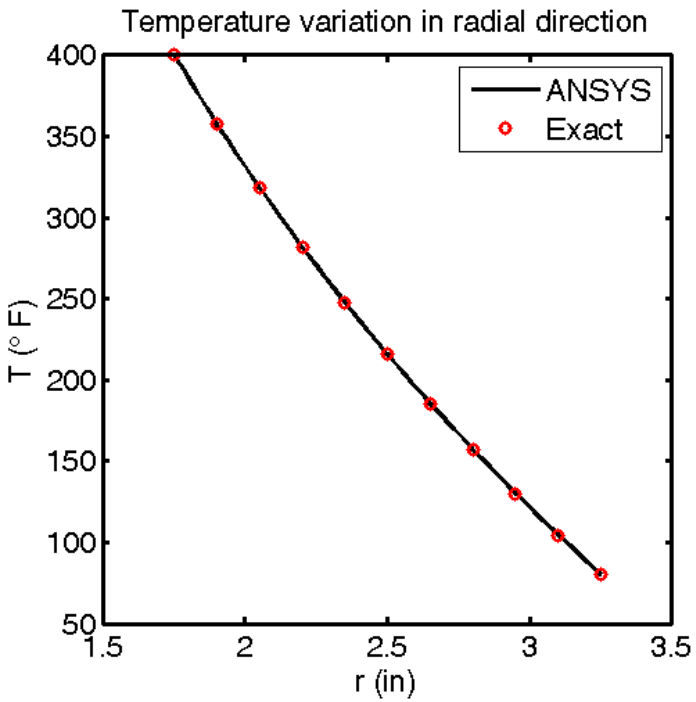
SECTION MEETING #5
9/28/2012

Updated Office Hours

- My office hours (held in Swanson Lab, 163 Rhodes):
  - M 3:30-4:30 pm
  - R 3:30-4:30 pm
  - F 2:30-3:30 pm
- Please come during these times for help with ANSYS modeling

Axisymmetric Heat Flow (HW4, #3)

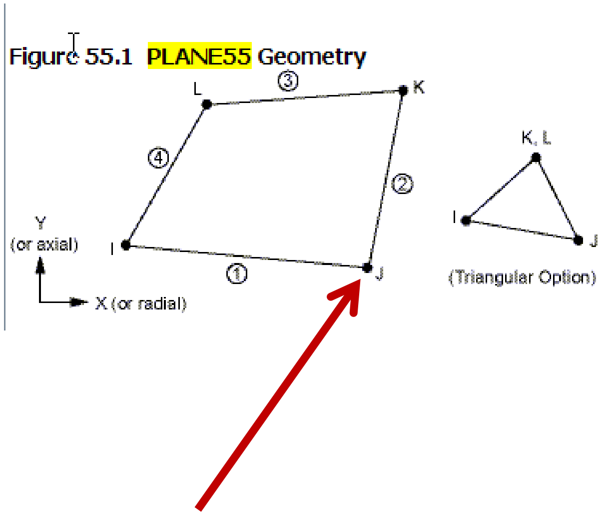
Comparison of ANSYS Result
with Exact Solution



Axisymmetric Heat Flow (HW4, #3)

- Element type used:
  - Look under *Solution > Solution Information*
    - *PLANE55*
  - Help page for *PLANE55*
    - 2D Conduction
    - Plane or Axisymmetric
    - Relevant material properties: KXX, KYY (=K in our case)

4 Nodes/Element



Single d.o.f. per node: Temperature

Axisymmetric Heat Flow (HW4, #3)

- Help page for *PLANE55*
  - Switch to go between plane and axisymmetric: *KEYOPT(3)*
  - *PLANE55 Assumptions and Restrictions*
    - Element must lie in an X-Y plane
    - Y-axis must be the axis of symmetry for axisymmetric analyses

Shape Function (follow link to Mechanical APDL Theory Reference)

$$T = \frac{1}{4}(T_I(1-s)\ .\ .\ .\ \text{(analogous to u)}$$

$$u = \frac{1}{4}(u_I(1-s)(1-t) + u_J(1+s)(1-t) + u_K(1+s)(1+t) + u_L(1-s)(1+t))$$

Lecture 6. page 7

$$u^e(x) = N_1^e d_1^e + N_2^e d_2^e$$

Axisymmetric Heat Flow (HW4, #3)

- Help page for PLANE55
  - A similar element with midside node capability is PLANE77

PLANE55

$$T = \frac{1}{4}(T_I(1-s)\ .\ .\ .\ (\text{analogous to } u)$$

$$u = \frac{1}{4}(u_I(1-s)(1-t) + u_J(1+s)(1-t) + u_K(1+s)(1+t) + u_L(1-s)(1+t))$$

PLANE77

$$T = \frac{1}{4}(T_I(1-s)\ .\ .\ .\ (\text{analogous to } u)$$

$$\begin{aligned} u = \frac{1}{4}(&u_I(1-s)(1-t)(-s-t-1) + u_J(1+s)(1-t)(s-t-1) \\ &+u_K(1+s)(1+t)(s+t-1) + u_L(1-s)(1+t)(-s+t-1)) \\ &+\frac{1}{2}(u_M(1-s^2)(1-t) + u_N(1+s)(1-t^2) \\ &+u_O(1-s^2)(1+t) + u_P(1-s)(1-t^2)) \end{aligned}$$

Important Takeaway

- Need to dig into the element manual to understand key things such as
  - Governing equations
  - Assumptions and restrictions
  - Element formulation
    - Number of nodes/element
    - Degrees of freedom at each node
    - Shape functions etc.

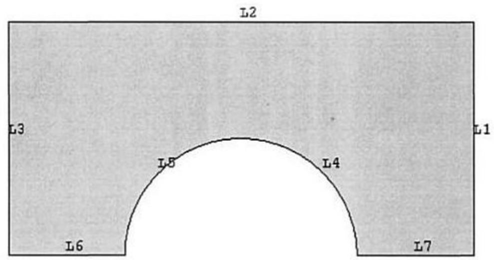
- INCLUDE SUCH INFORMATION IN YOUR PROJECT PRESENTATION/REPORT

Mapped Meshing

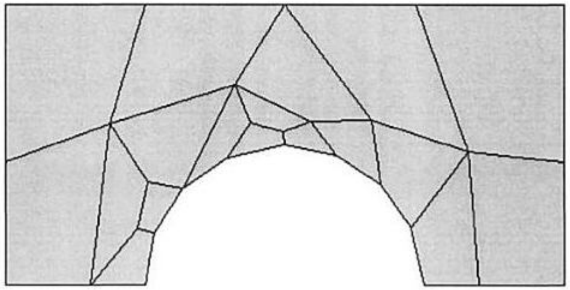
- Valid in 2D and 3D
- Generates regular meshes that generally lead to increased accuracy
- Can be used only in “regular” regions
- For 2D, works on areas with 3 or 4 sides
  - 4 sides: Opposite sides have equal number of divisions
  - 3 sides: All sides must have an equal, even number of divisions

Mapped Meshing

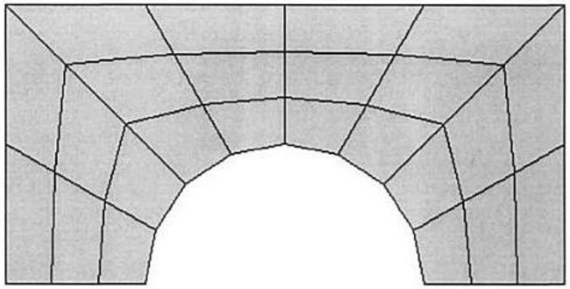
- Example from “The finite element method and applications in engineering using ANSYS” by Madenci & Guven



Free Mesh



Mapped Mesh

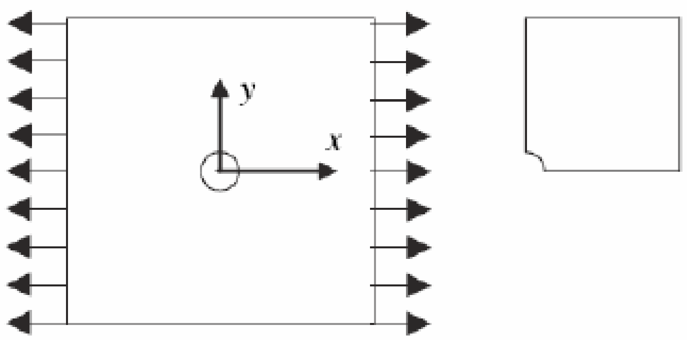


Mapped Meshing

- Why the “mapped” in name?
  - A four-sided area with equal number of divisions on the opposite edges can be mapped to a regular mesh on a square

Upcoming HW

Problem 4 – Thin plate with a hole in tension (MatLab)



Consider a tension problem involving a thin linearly elastic plate with a hole as shown in the figure. Suppose that the plate is a homogeneous isotropic elastic body.

The plate is of unit thickness and subject to tension in the horizontal direction. Because of symmetry in the model and loading, model only one quarter of the plate. Use ANSYS to generate the gird files using 4-node quadrilateral elements.

The plate is $20\ \text{cm} \times 20\ \text{cm}$ and the radius of the hole is $2.5\,\text{cm}$. Assume Young's modulus is $2.1 \times 10^7\ \text{N/cm}^2$ and Poisson's ration is 0.29. The uniform load applied is $\sigma_0 = 100\,\text{N/cm}$.

Plate with a Hole

- Degrees of freedom per node: ux, uy
- Element formulation:
  - Derived from weak form of elasticity equation
    - Solves the elasticity equation