# MAE 4700/5700: ANSYS Section

Fridays 1:25-2:15 pm

Rajesh Bhaskaran

Cornell University

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

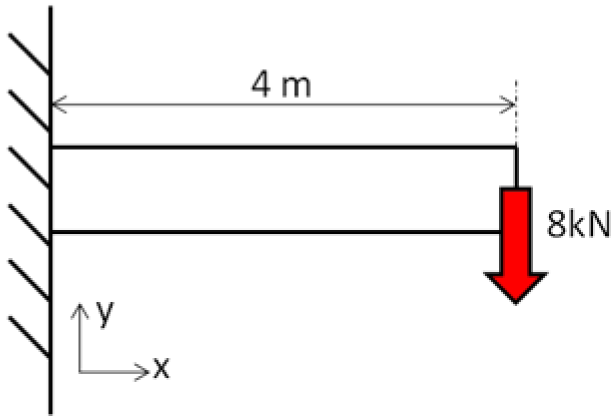
# ANSYS Software

- 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



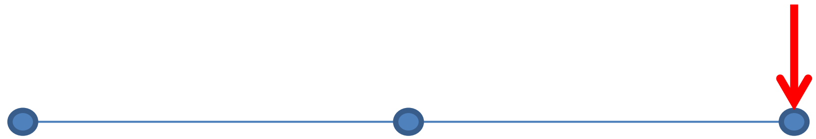

- Truss elements are available in ANSYS
  - Need to use scripting (advanced feature)
  - Not used widely in practice

# 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



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

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

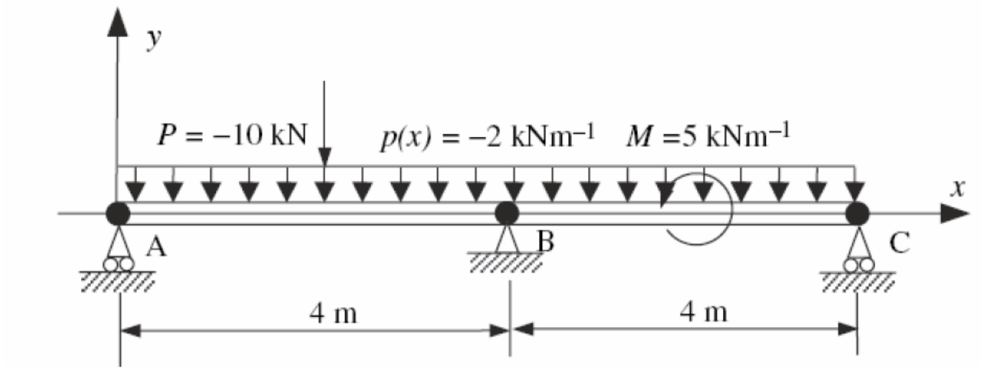
Reaction force at node 1
Reaction moment at node 1

# SECTION MEETING #2
# 9/7/2012

# Cantilever Beam

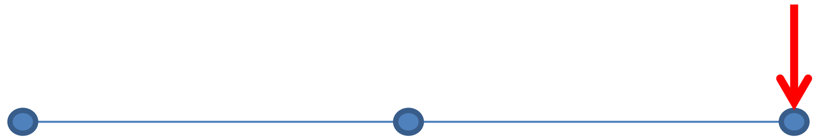
- First ANSYS exercise
  - Can do trusses but need to use scripting (advanced functionality)
  - Pin-jointed trusses rarely occur in practice
- ANSYS beam problem will appear in HW3

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



# Cantilever Beam: Degrees of Freedom

- Consider 2-element mesh



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

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

Reaction force at node 1
Reaction moment at node 1

# Cantilever Beam
# Beam element stiffness matrix

| | |
|---|---|
| Offset Type | Centroid |
| Model Type | Beam |
| **Material** | |
| Assignment | Go Big Red |
| Nonlinear Effects | Yes |
| Thermal Strain Effects | Yes |

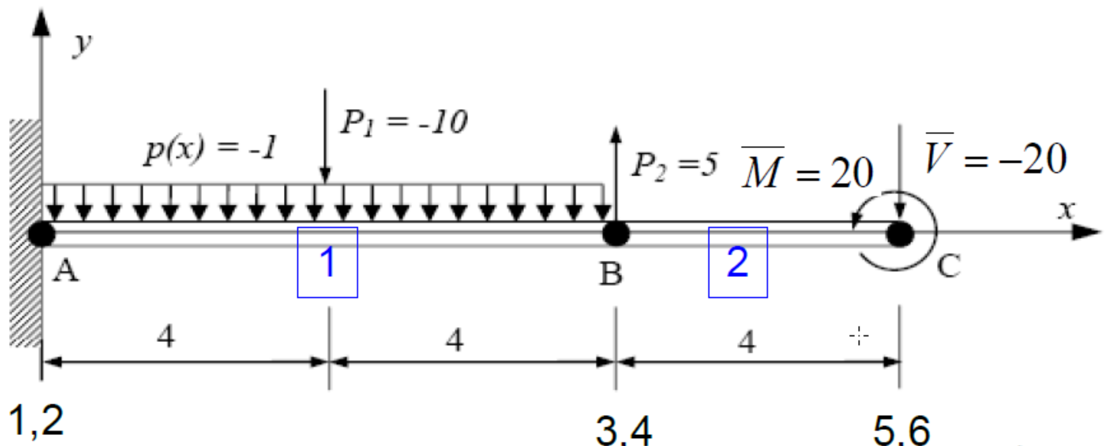
$K^e$ : see page 24 of *Lecture 4: Finite element formulation of beam problems*

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

| **Details of Line Body** | |
|---|---|
| Body | Line Body |
| Faces | 0 |
| Edges | 1 |
| Vertices | 2 |
| Cross Section | Rect1 |
| Offset Type | Centroid |

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

# Solve Step
# Beam Lecture, Page 43

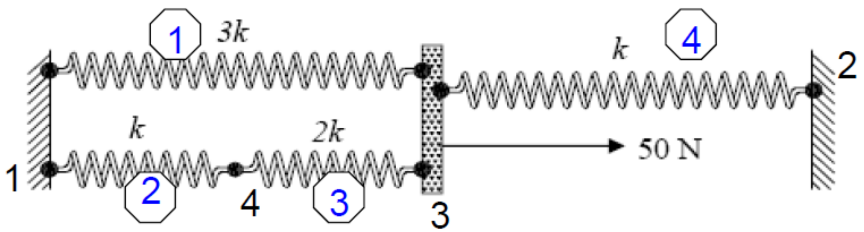



$$10^3 \left[\begin{array}{cc:cccc} 0.23 & 0.94 & -0.23 & 0.94 & 0 & 0 \\ 0.94 & 5.00 & -0.94 & 2.50 & 0 & 0 \\ \hdashline -0.23 & -0.94 & 2.11 & 2.81 & -1.88 & 3.75 \\ 0.94 & 2.50 & 2.81 & 15.00 & -3.75 & 5.00 \\ 0 & 0 & -1.88 & -3.75 & 1.88 & -3.75 \\ 0 & 0 & 3.75 & 5.00 & -3.75 & 10.00 \end{array}\right] \left\{\begin{array}{c} u_{y1}=0 \\ \theta_{y1}=0 \\ \hdashline u_{y2} \\ \theta_{y2} \\ u_{y3} \\ \theta_{y3} \end{array}\right\} = \left\{\begin{array}{c} -9+R_{u1} \\ -15.3+R_{\theta 1} \\ \hdashline -4 \\ 15.3 \\ -20 \\ 20 \end{array}\right\}$$

# 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: $d_1 = d_2 = 0$

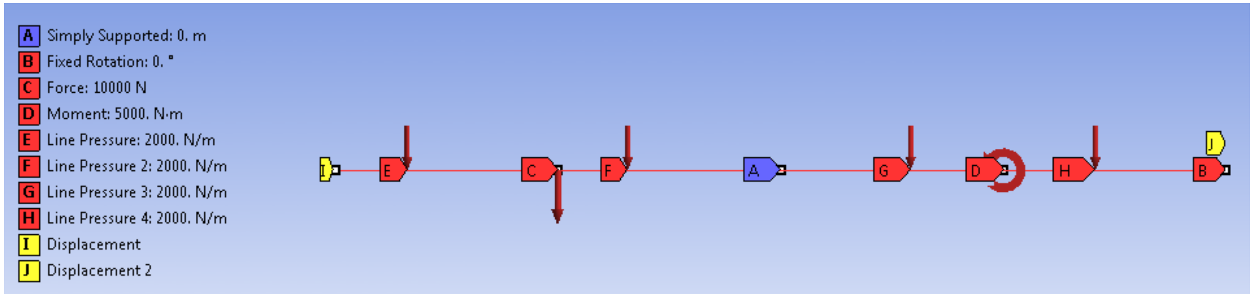
$$k\begin{bmatrix} 6 & -2 \\ -2 & 3 \end{bmatrix}\begin{Bmatrix} d_3 \\ d_4 \end{Bmatrix} = \begin{Bmatrix} 50N \\ 0 \end{Bmatrix} \Rightarrow \begin{Bmatrix} d_3 \\ d_4 \end{Bmatrix} = \frac{1}{k}\begin{Bmatrix} 10.7143 \\ 7.1429 \end{Bmatrix} N$$

# 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 $\theta_z$ free
- *Frictionless support* in this case will set $u_y$ to zero but leaves $u_x$ and $\theta_z$ free

# HW3 Tips

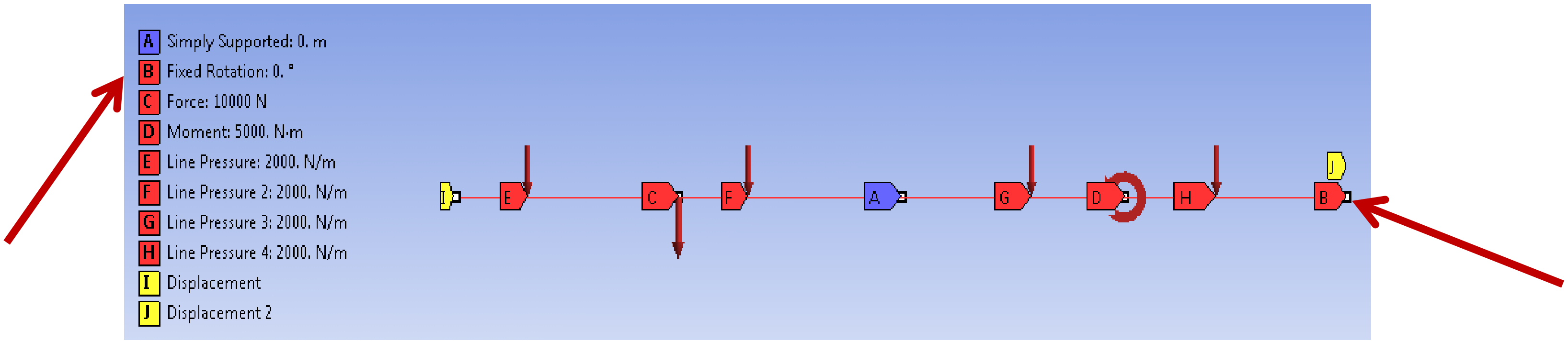
- You have to add a BC by fixing the rotations about x and y axes as in the snapshot below. Otherwise you might get a solver pivot error. This is because ANSYS is using a 3D beam element with these additional rotations as dof's.



# 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*

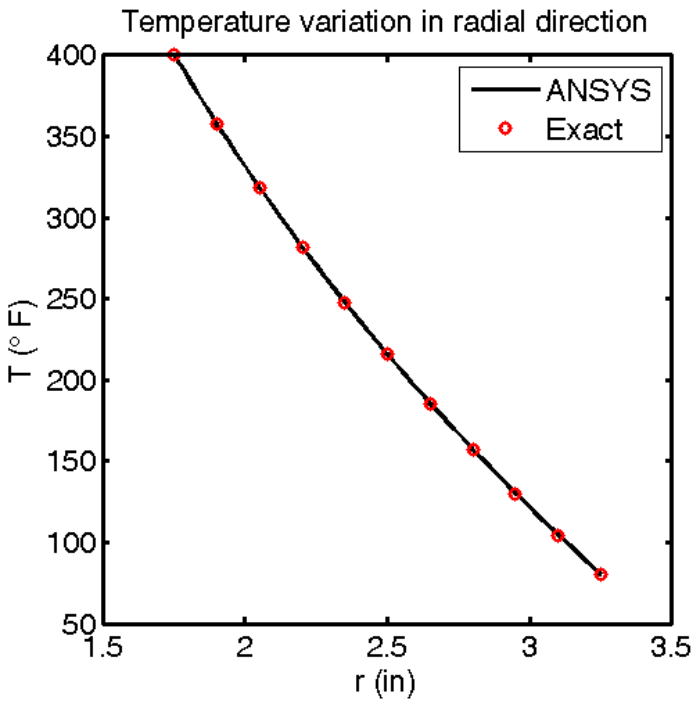
# SECTION MEETING #5
# 9/28/2012

# 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)

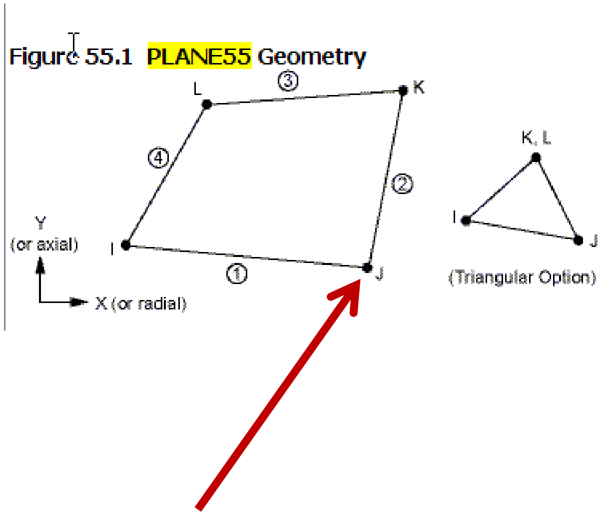
Comparison of ANSYS Result
with Exact Solution



# 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

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

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

# 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.

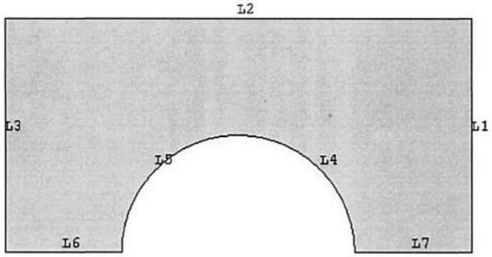
- INCLUDE SUCH INFORMATION IN YOUR PROJECT PRESENTATION/REPORT

# Mapped Meshing

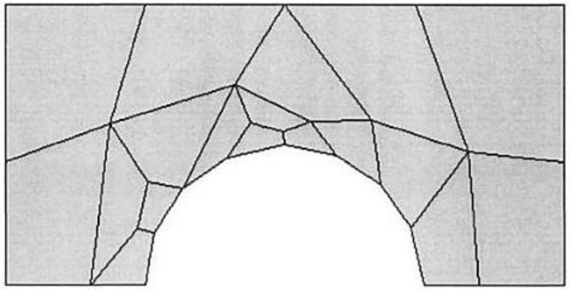
- Valid in 2D and 3D
- Generates regular meshes that generally lead to increased accuracy
- Can be used only in “regular” regions
- For 2D, works on areas with 3 or 4 sides
  - 4 sides: Opposite sides have equal number of divisions
  - 3 sides: All sides must have an equal, even number of divisions

# Mapped Meshing

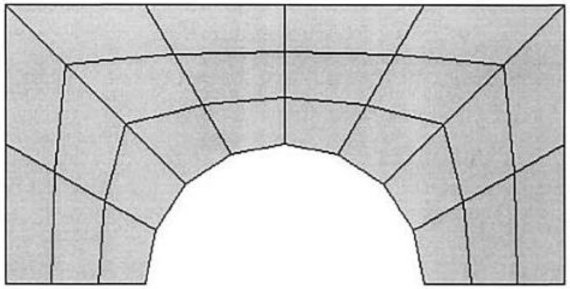
- Example from “The finite element method and applications in engineering using ANSYS” by Madenci & Guven




Free Mesh



Mapped Mesh

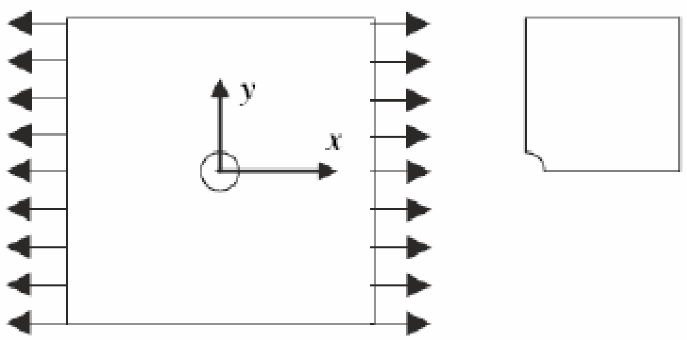


# Mapped Meshing

- 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

# SECTION MEETING #6
# 10/5/2012

# Updated Office Hours (Reminder)

- 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

# Swanson Lab (163 Rhodes)

- 16 Dell T5500 workstations
  - 2 quad-core Intel Xeon 2.26GHz processors
  - 30 GB of RAM
  - Extensive software suite
    - ANSYS including FLUENT
    - MATLAB
    - CAD software: Solidworks, Creo/ProE
- Suitable for large finite-element models (> ~200k elements)
- Contact me if you have access problems

# Plan for Today

- Sources for project ideas
  - Verification manual in ANSYS
  - Slides from ANSYS Tech Days
- Project approach
  - Novice vs. expert framework
  - Incremental build-up of FEA model
- Meshes for HW 6
- Plate with a hole in ANSYS (continuation)

# Project Ideas: Verification Manual

- Search for “verification manual” in ANSYS help
  - Look through list of verification problems
  - Example:
    - *VM14  Large Deflection Eccentric Compression of a Column*
  - Can also sort by element number
    - Choose *Index by Element Number*

# Project Ideas: Slides from ANSYS Tech Days

- ANSYS Tech Days was held on campus in March 2012
  - Several presentations from ANSYS engineers on industrial applications
  - These slides have been sent to you via dropbox
- **Stay away from fluid flow applications**
    - Uses finite-volume solver (FLUENT or CFX), not finite-element solver
    - Not appropriate for MAE 4700/5700

# ANSYS in Structural Mechanics

Nonlinear Analysis

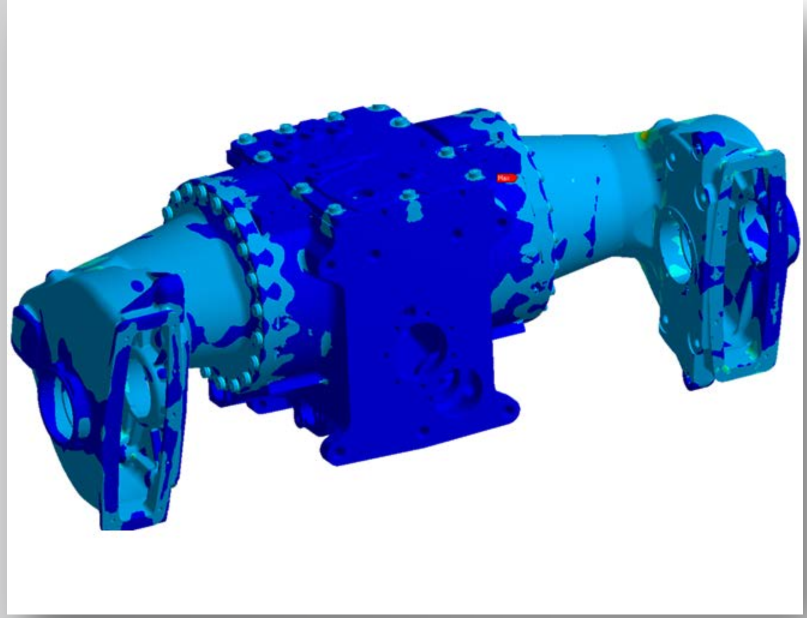
Fracture Mechanics

Linear Static

Instabilities

Acoustics

Contact



Explicit

Dynamics – Modal

Dynamics – Spectral

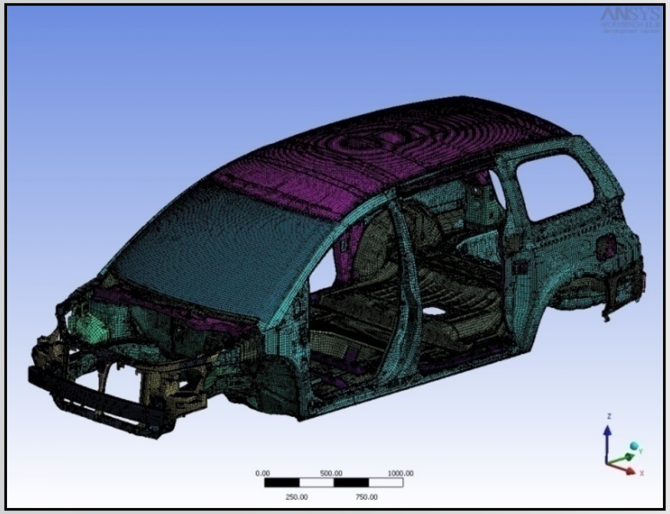
Dynamics – Harmonic

Dynamics – Transient

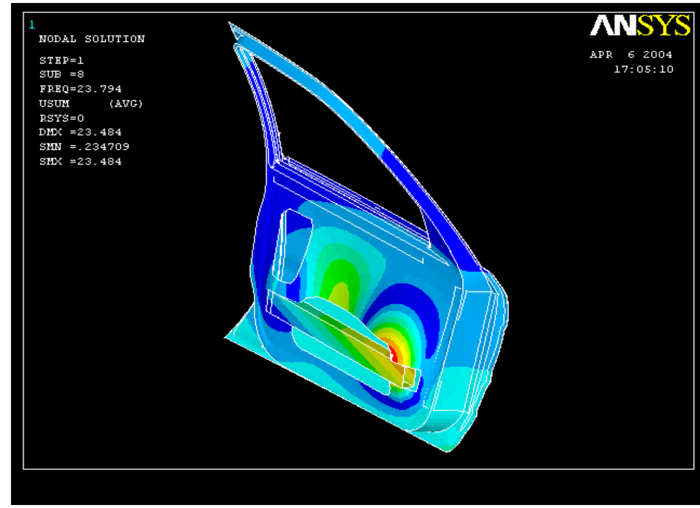
Rigid/Flexible Mechanisms

# BIW & Component Modeling

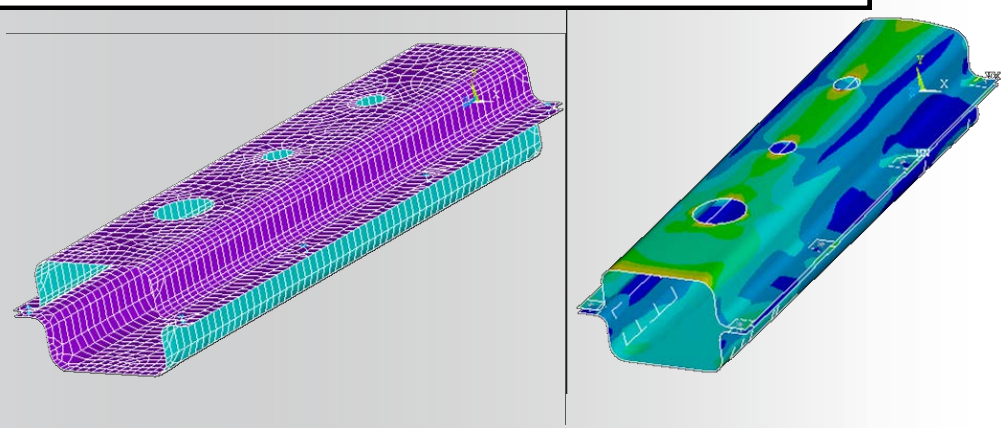
## Comprehensive Structural capability



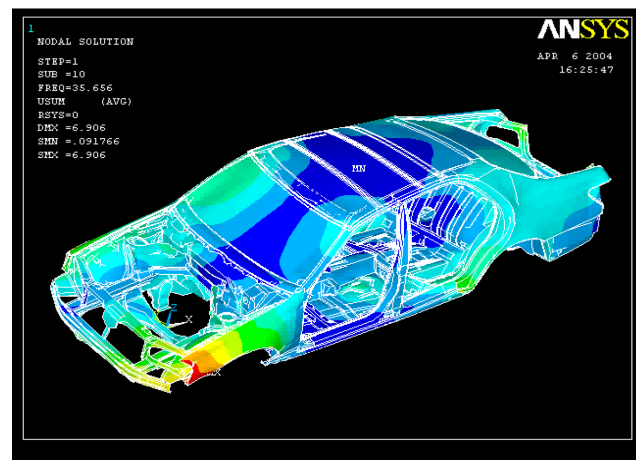
Efficient Mid surfacing, Automatic Contact Detection




Non linear analysis of Door Assembly



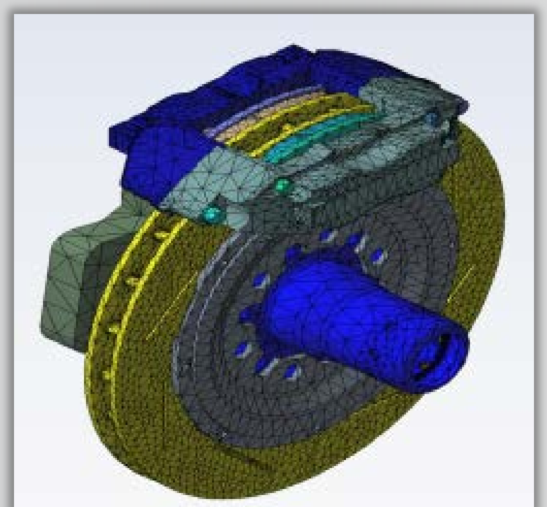
Spot Welds

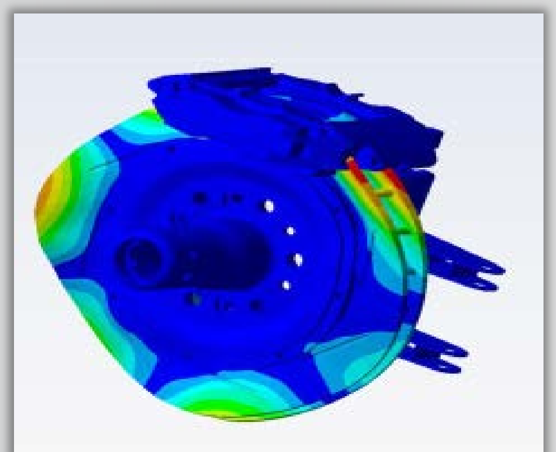



Dynamic Structural analysis

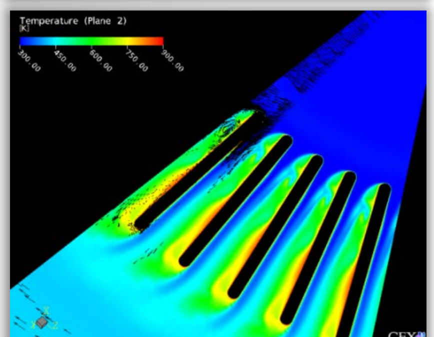
# Brakes Examples

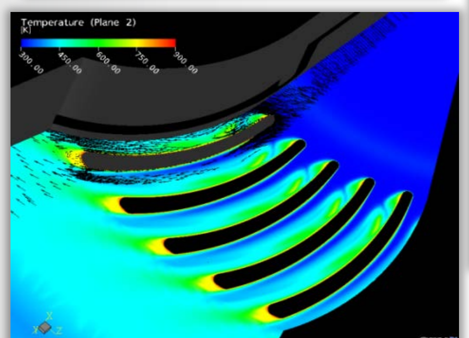
## Estimation of Unstable Modes

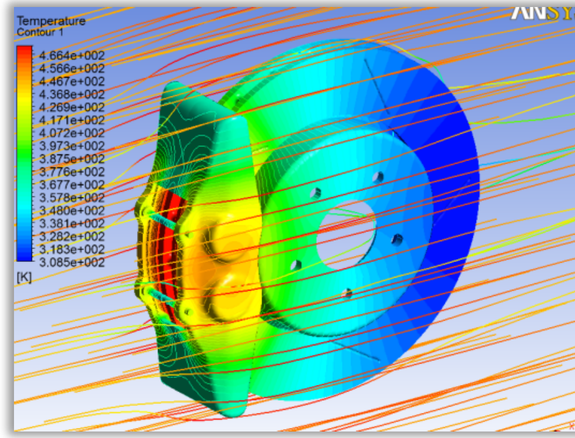


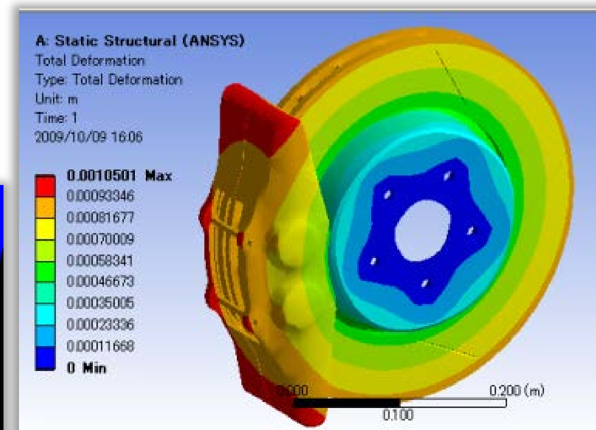


## Thermal management



# Application Area - Orthopaedics

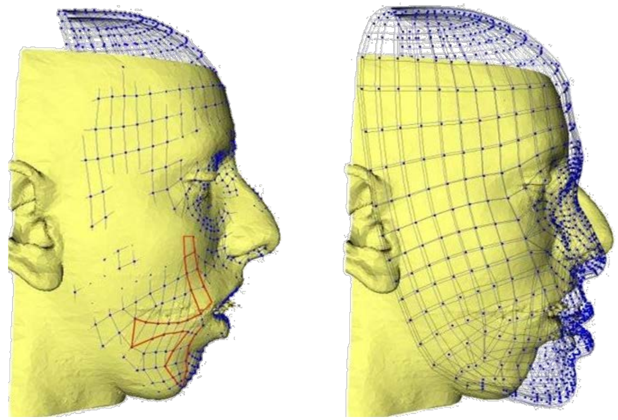
## Prosthetics

- Hip, spine, knee, elbow, shoulder, wrist
- Cement injection

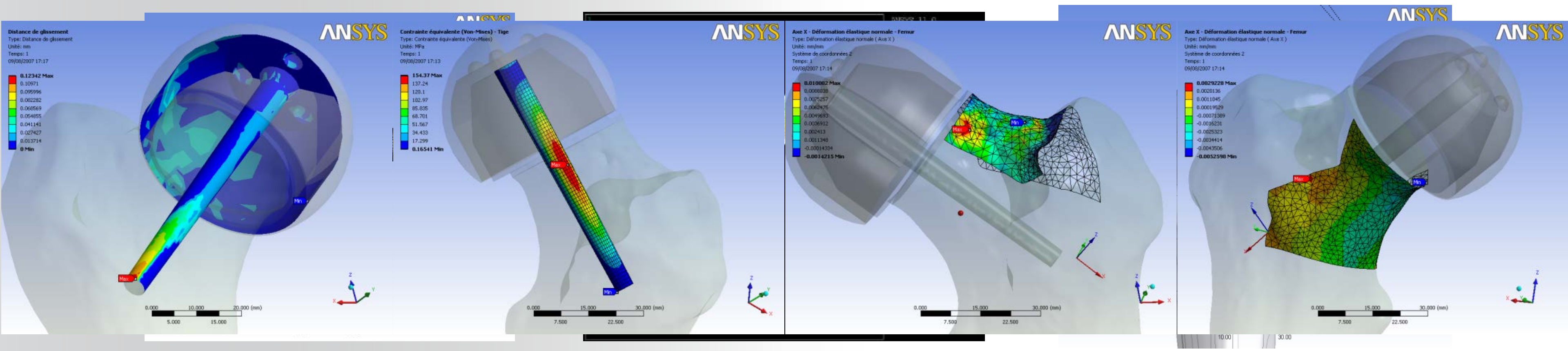
## Soft tissues

- Maxillofacial (re)modeling
- Advanced prostheses (arms, legs)
- Tumor withdrawal (electromagnetic)

## Surgical planning



Maxillofacial surgery modeling by TIMC-IMAG
(courtesy of TIMC-IMAG Laboratory, CNRS/UJF)



Sliding distance

Femur mesh

Von Mises stress in the prosthesis

Interpolated Ym

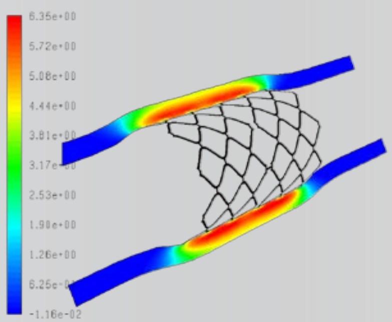
Trabecular tensile and compressive strain

Parametric model - 2 rotation parameters of the prosthesis

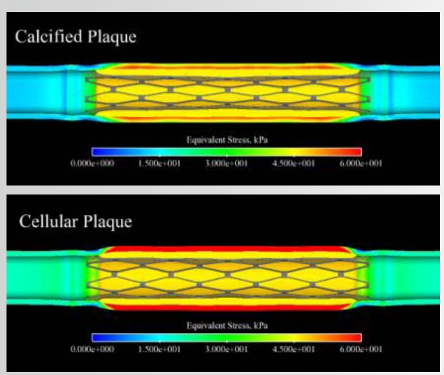
# Application Area - Cardiovascular

**Stents, heart valves, catheters, transcutaneous treatments, blood pumps, oxygenators, artificial hearts**

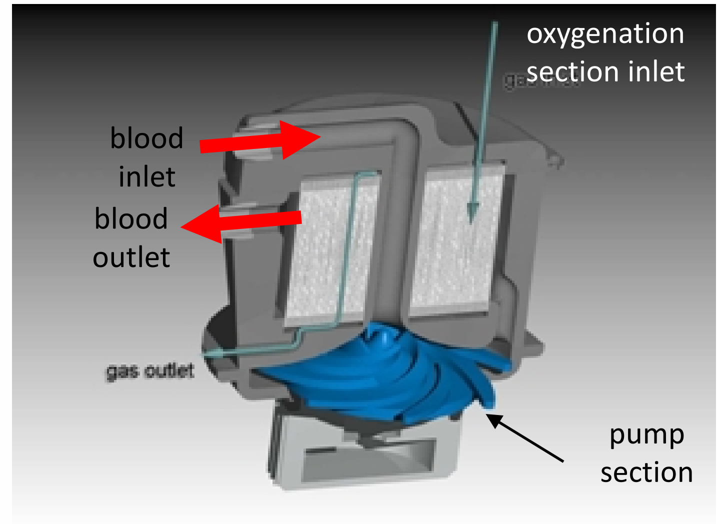
**Design, testing (mechanical testing, in-vivo testing)**

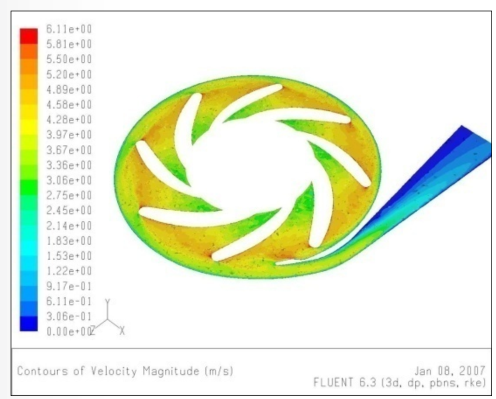


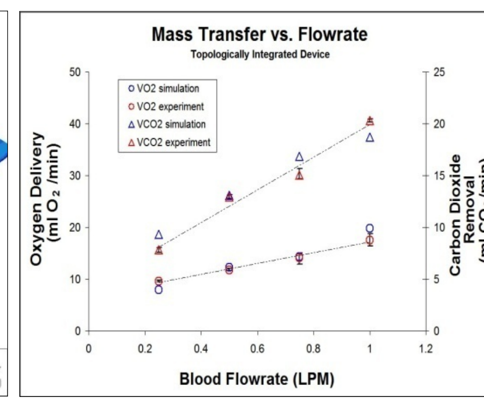
Drug eluting stent, species distribution in the artery wall




Arterial/Plaque Stresses for Calcified (top) and Cellular (lower) Plaque



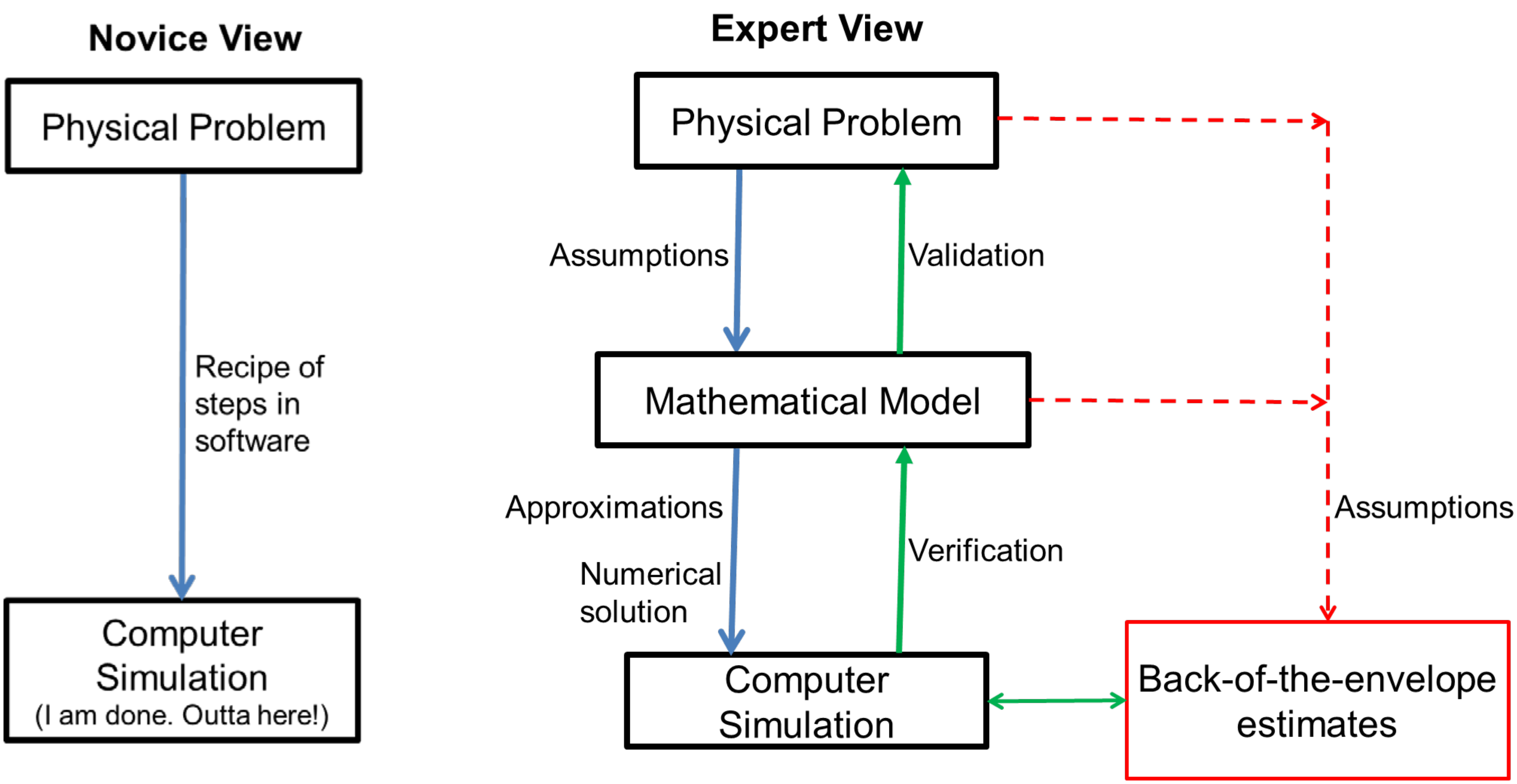

* Fill et al., ASAIO Journal (2008).

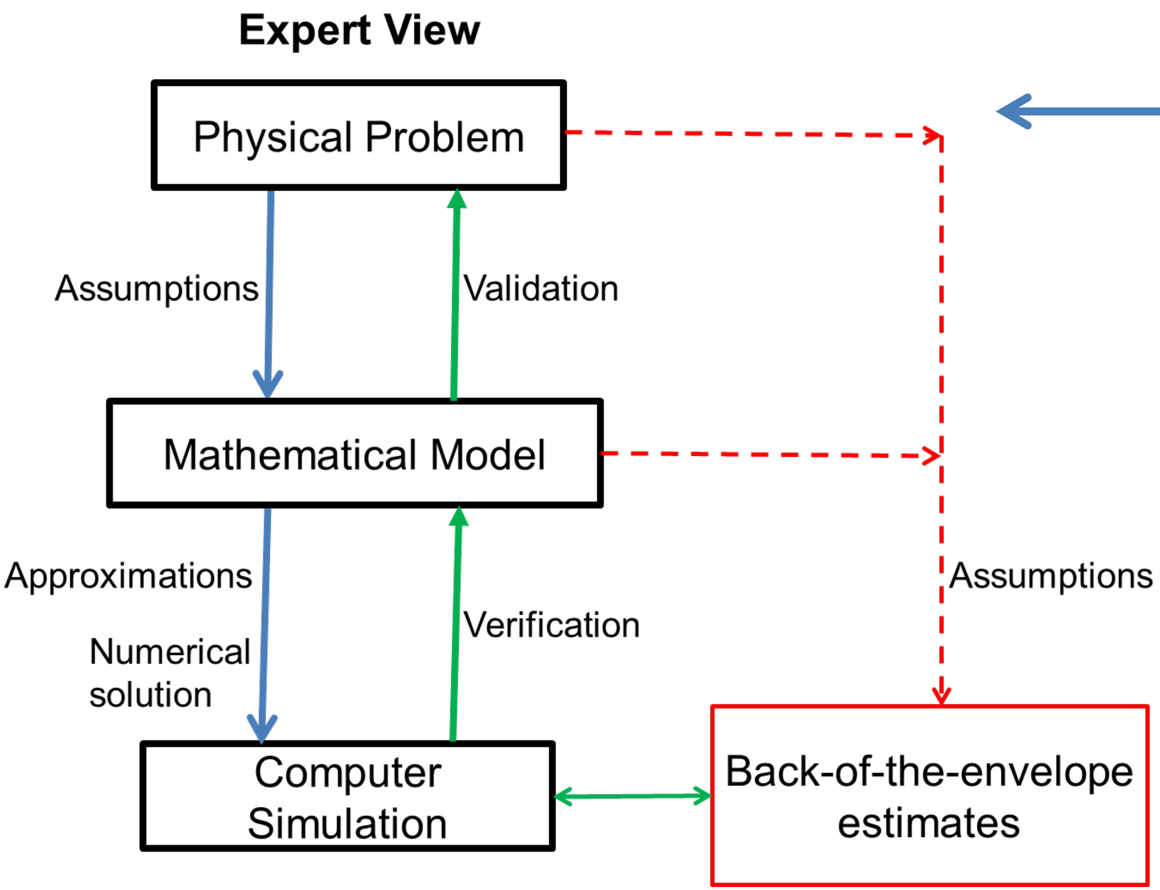
# Plan for Today

- Sources for project ideas
  - Verification manual in ANSYS
  - Slides from ANSYS Tech Days
- Project approach
  - Novice vs. expert framework
  - Incremental build-up of FEA model
- Meshes for HW 6
- Plate with a hole in ANSYS (continuation)

# Novice vs. Expert Framework



# What to Cover in Project Report and Presentation?

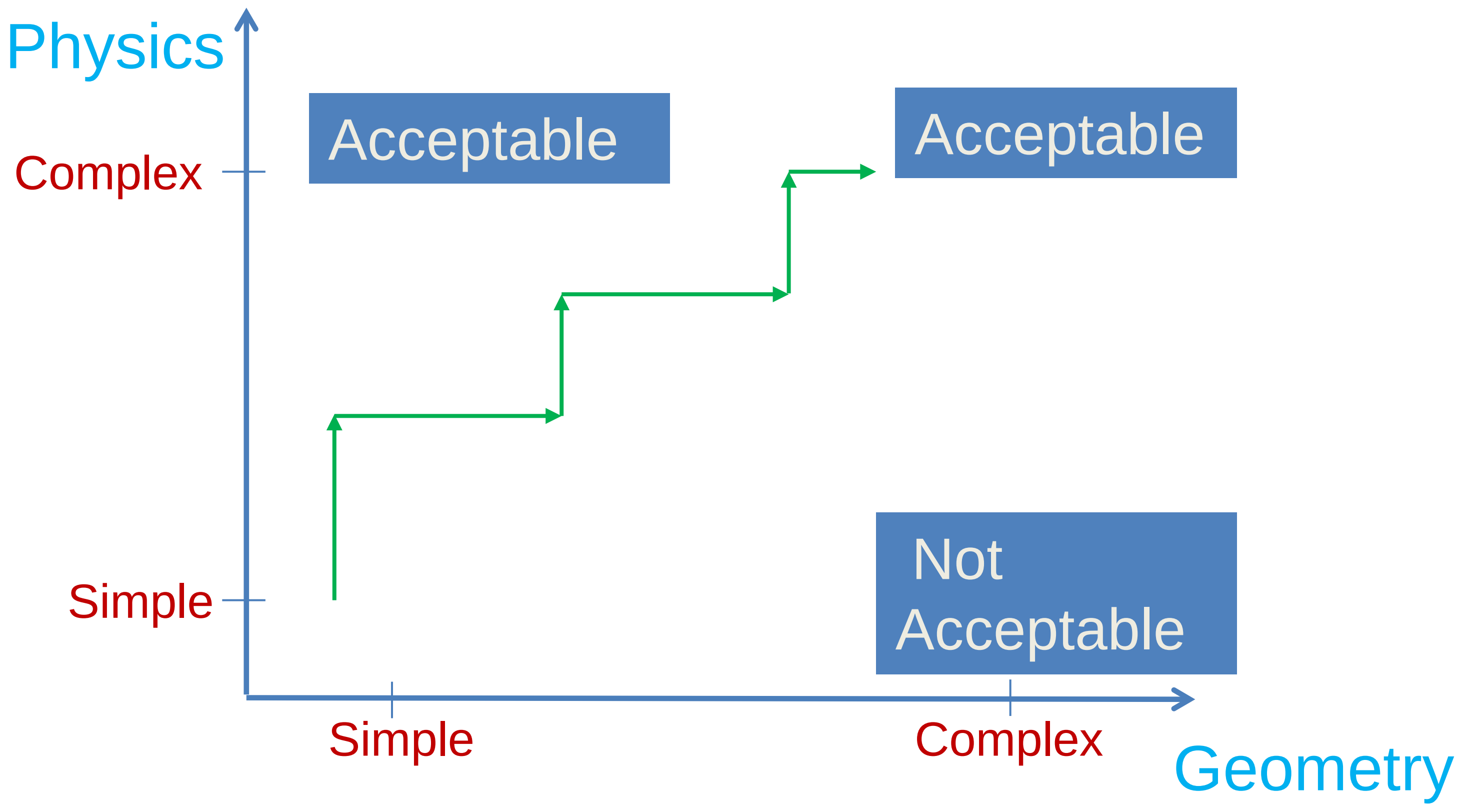



Cover all items in the “expert” framework

In the process, you will come across as an FEA expert!

More discussion in the coming weeks

# Incremental Build-Up of FEA Model



# Plan for Today

- Sources for project ideas
  - Verification manual in ANSYS
  - Slides from ANSYS Tech Days
- Project approach
  - Novice vs. expert framework
  - Incremental build-up of FEA model
- Meshes for HW 6
- Plate with a hole in ANSYS (continuation)

# Meshes for HW6

- HW6, problem 4, Convection heat transfer:
  - Download from
    - [*https://confluence.cornell.edu/display/SIMULATION/Files*](https://confluence.cornell.edu/display/SIMULATION/Files)
- HW6, problem 6: To be posted

# SECTION MEETING #7
# 10/12/2012

# Announcements

- ANSYS FEA expert visiting Oct. 17-19
  - Will present in class next week
  - Will provide free consultation for students on advanced FEA using ANSYS
- GE Energy summer internship opportunity
  - Learn to perform FEA (and CFD) on real engineering devices
  - Pay $20-25/hour
- Experiment
  - Get through class without texting, facebook, e-mail etc.

# ANSYS in Structural Mechanics

Nonlinear Analysis

Fracture Mechanics

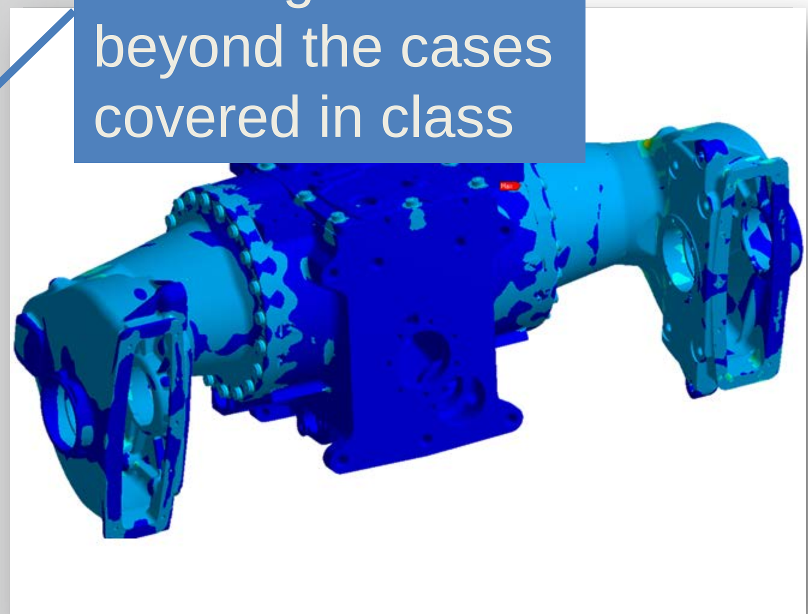
Linear Static

Instabilities

Acoustics

Contact

OK for project but should go well beyond the cases covered in class



Explicit

Dynamics – Modal

Dynamics – Spectral

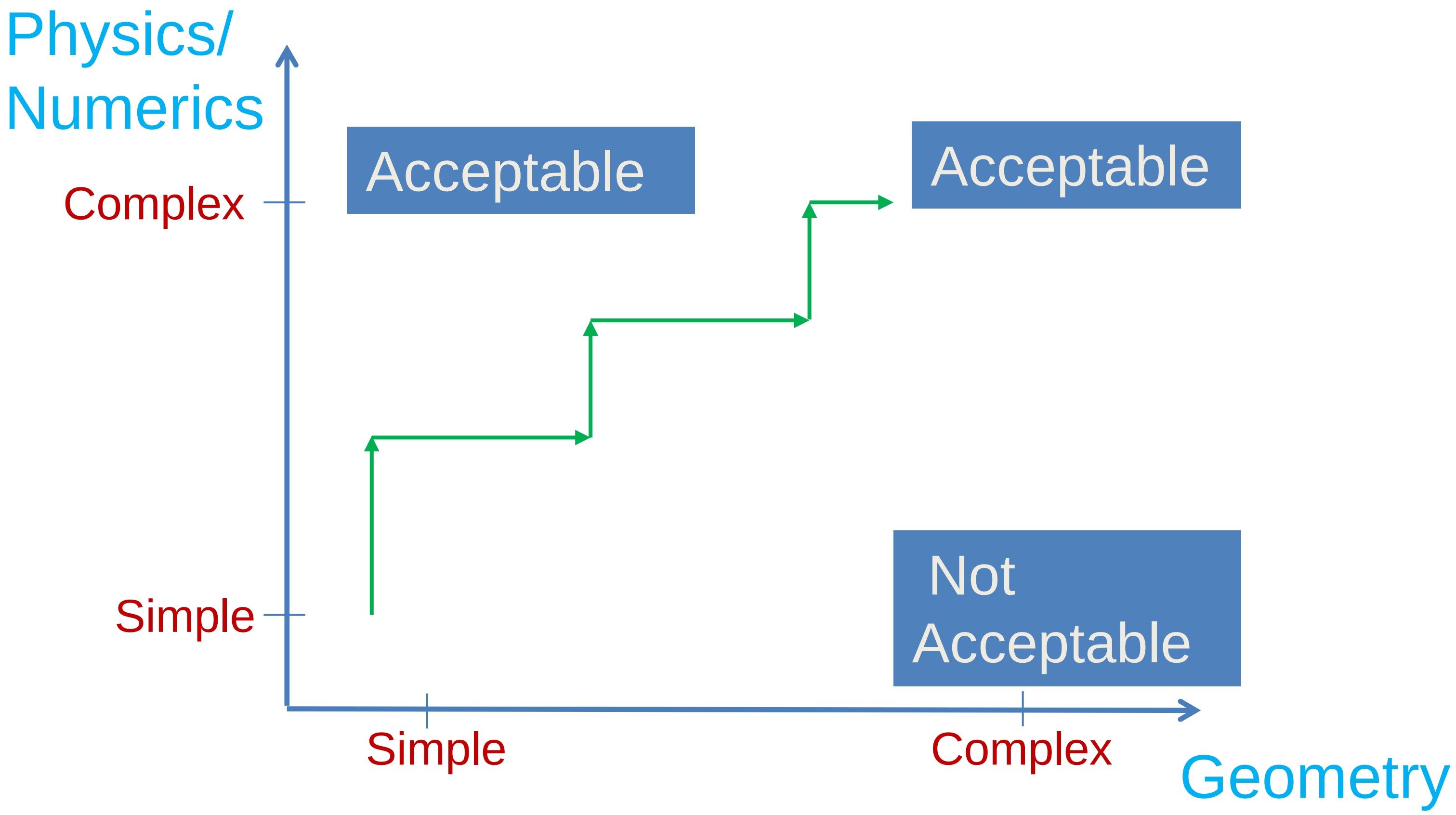
Dynamics – Harmonic

Dynamics – Transient

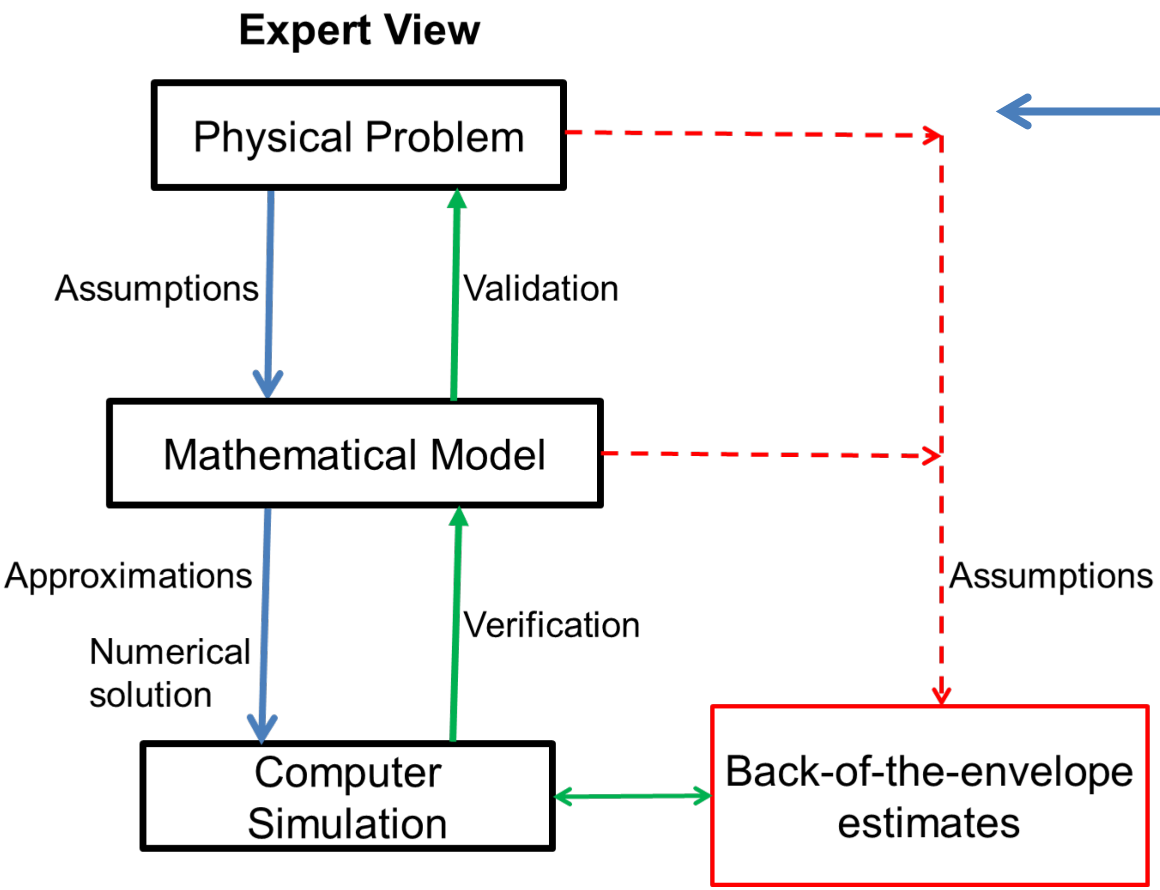
Rigid/Flexible Mechanisms

# Incremental Build-Up of FEA Model



# What to Cover in Project Report and Presentation?



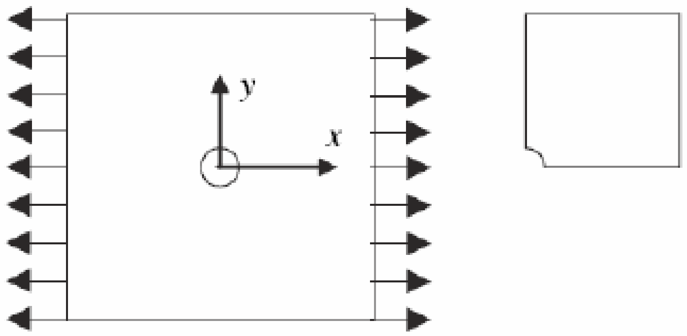

Cover all items in the “expert” framework

In the process, you will come across as an FEA expert!

More discussion in the coming weeks

# 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 $\sigma_0 = 100\ \text{N/cm}$.

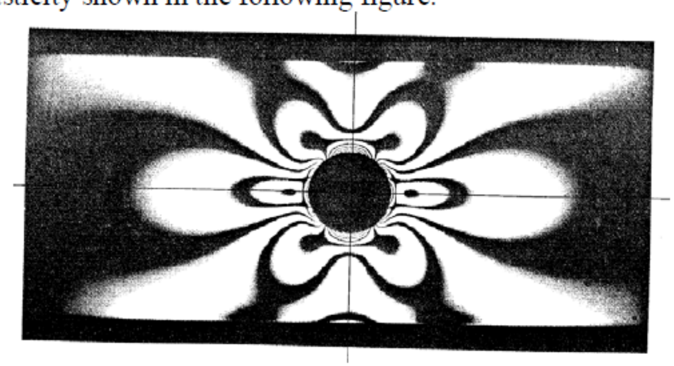
To check the quality of the finite element solution, compare your solution with that of an infinite plate. For a circular hole in an infinite plate subjected to a pure uniaxial tension or compression $\sigma_0$, at $\infty$, there exists an exact solution for the stresses (Timoshenko and Goodier, 1970):

$$\sigma_{rr} = \frac{\sigma_0}{2}\left\{\left[1-\left(\frac{a}{r}\right)^2\right]+\left[1-4\left(\frac{a}{r}\right)^2+3\left(\frac{a}{r}\right)^4\right]\cos 2\theta\right\}$$

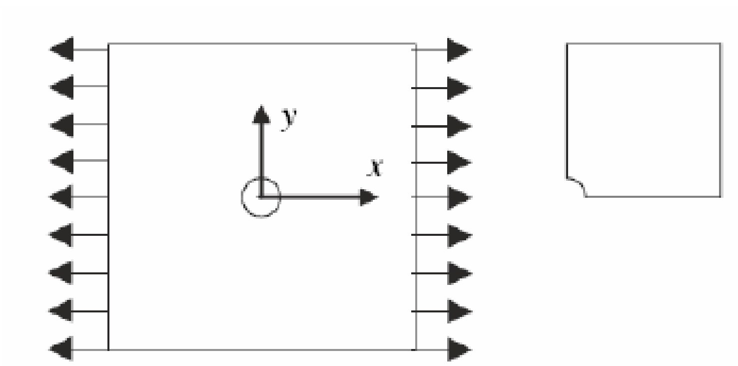
$$\sigma_{\theta\theta} = \frac{\sigma_0}{2}\left\{\left[1+\left(\frac{a}{r}\right)^2\right]-\left[1+3\left(\frac{a}{r}\right)^4\right]\cos 2\theta\right\}$$

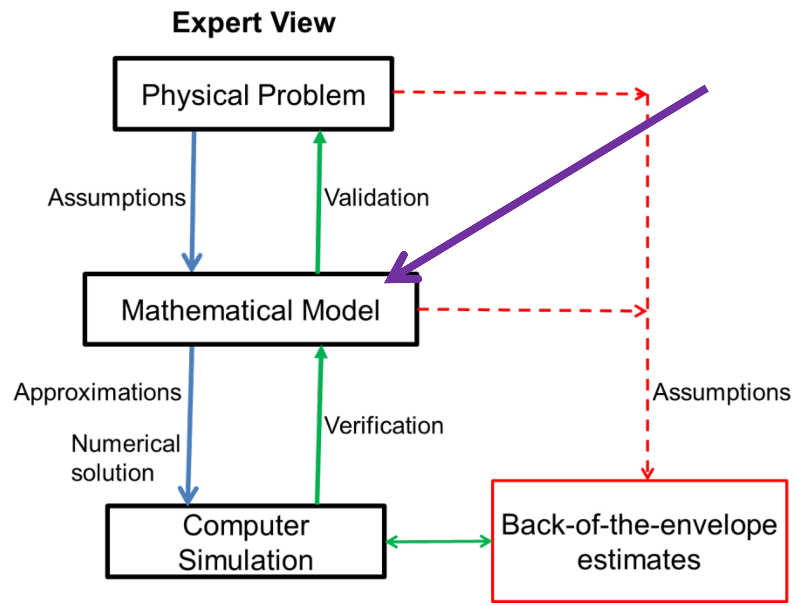
$$\sigma_{r\theta} = -\frac{\sigma_0}{2}\left[1+2\left(\frac{a}{r}\right)^2-3\left(\frac{a}{r}\right)^4\right]\sin 2\theta$$

a. Provide a plot showing the FE and exact $\sigma_{xx}$ stress distribution along the $y$-axis. What is the stress concentration factor?
b. Show the deformed configuration
c. Compute and plot the distribution of the finite element principal stress solution $(\sigma_1, \sigma_2)$ and the maximum shear stress solution.
d. Note that an interesting comparison can be made with the stress pattern obtained by photoelasticity in which fringes correspond to the maximum shear stress $\tau_{max} = 1/2(\sigma_1 - \sigma_2)$. Verify that the contour lines of $\tau_{max}$ obtained by the finite element approximation are quite similar to the fringe pattern obtained by the photoelasticity shown in the following figure.



# Plate with a Hole in Tension




- Boundary value problem
  - Governing equation
  - Domain
  - Boundary conditions