

# Transducer Design and Modeling

42<sup>nd</sup> Annual UIA Symposium

Orlando Florida 2013

Jay Sheehan

JFS Engineering

# Transducer Design and Modeling

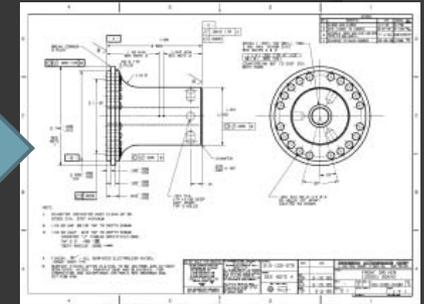
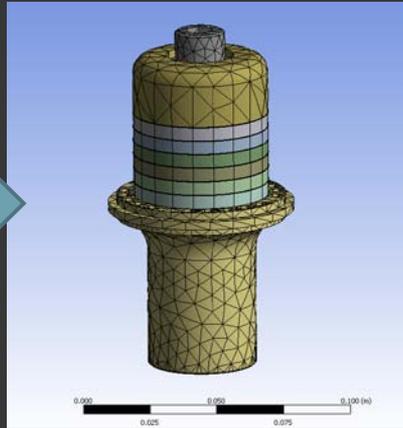
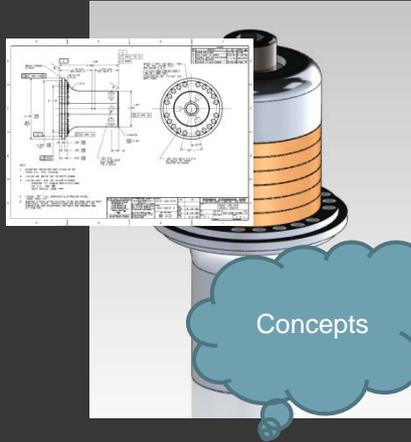
- Introduction
- ANSYS Workbench Introduction
  - The project format
  - Setting up different analysis – Static, Modal and Harmonic
  - Connection to the solid modeling package
  - Material library
  - ANSYS mechanical
  - Setup challenges
- The converter stack solid model
  - Components
  - Modeling Techniques
- Model example - modal analysis
  - Use of APDL programming language
  - Set up of piezoelectric materials / assignments
  - Theory – series and parallel analysis
  - Boundary conditions – series and parallel analysis
  - Examining results
- Model example – harmonic analysis
  - Use of APDL programming language
  - Boundary conditions – harmonic analysis
  - Examining results – single point and frequency sweep
- Model example – static analysis (pre-load)
  - Model setup options
  - Examining results

# Transducer Design and Modeling

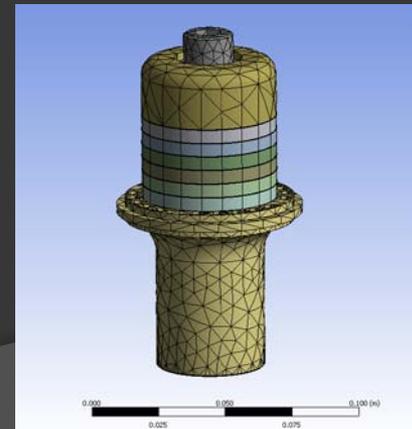
## - Introduction -

Recent advancements in finite element software and computing power have changed the methods to analyze ultrasonic transducers

*OLD Way*



*New Way*



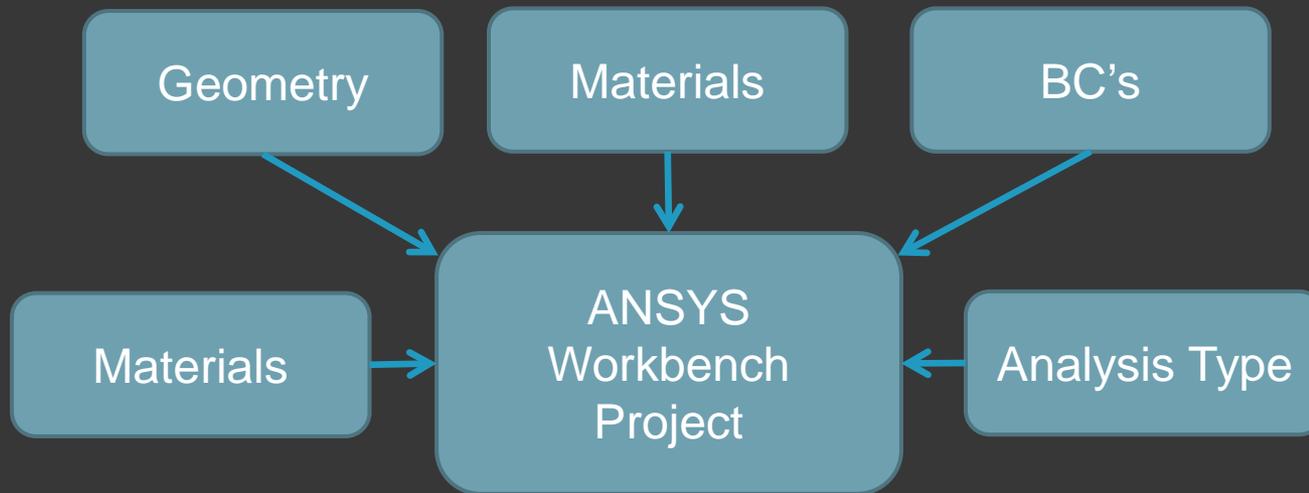
# Transducer Design and Modeling

## - Introduction -

ANSYS Workbench provides a seamless connection between the design data (solid model) and the analysis (FE model, analysis type, BC's)

- Advantages – Single point to connect different analysis, types, BC's, geometry

- Disadvantages (for US analysis) – Piezo materials **not** incorporated in standard library



***The disadvantages can be overcome through the use of ANSYS APDL coding***

# Transducer Design and Modeling

## - ANSYS Workbench, The Project Format -

ANSYS Workbench is set up in an object orientated project format  
"Drag and Drop"

The screenshot displays the ANSYS Workbench interface. On the left is the 'Toolbox' containing various analysis systems. The central 'Project Schematic' shows three analysis systems: 'UJA 20kHz Modal', 'UJA 20kHz Harmonic', and 'UJA 20kHz Preload'. Each system has a hierarchical list of steps: 1. Analysis System, 2. Engineering Data, 3. Geometry, 4. Model, 5. Setup, 6. Solution, and 7. Results. A blue callout box labeled 'Active Analysis' has arrows pointing to the 'Solution' step of each of the three analysis systems. Another blue callout box labeled 'Different Analysis Choices' has an arrow pointing to the 'Analysis Systems' section of the toolbox. On the right, the 'Properties of Project Schematic' panel is visible, showing a table with columns A and B, and rows for Property, Notes, and Value.

	A	B
1	Property	Value
2	Notes	
3	Notes	

# Transducer Design and Modeling

## - ANSYS Workbench, The Project Format -

Each analysis is self contained. Certain analysis elements can be connected (Materials, geometry)

The screenshot displays the ANSYS Workbench interface. The main window shows the Project Schematic with three analysis systems: A (UIA 20kHz Modal), B (UIA 20kHz Harmonic), and C (UIA 20kHz Preload). System A is highlighted with a blue arrow pointing to a detailed view of its components. The detailed view shows a list of components for system A:

Component ID	Component Name	Status
1	Modal	
2	Engineering Data	✓
3	Geometry	✓
4	Model	✓
5	Setup	✓
6	Solution	✓
7	Results	✓

The Properties of Project Schematic table on the right shows the following data:

	A	B
1	Property	Value
2	Notes	
3	Notes	

# Transducer Design and Modeling

## - ANSYS Workbench, Setting Up Different Analysis -

- Grab available analysis from the toolbox. Connect analysis elements as required
- This presentation will look at a modal, harmonic, and static preload analysis connected to common geometry and material
- Welding stack analysis – Stack Bolt thread Stress

The screenshot displays the ANSYS Workbench interface for a project named "UIA 20kHz". The Project Schematic shows three analysis systems (A, B, and C) connected to a common geometry and material. System A is a Modal analysis, System B is a Harmonic Response analysis, and System C is a Static Structural analysis. The connections are as follows:

- System A (UIA 20kHz Modal) is connected to Engineering Data, Geometry, Model, Setup, Solution, and Results.
- System B (UIA 20kHz Harmonic) is connected to Engineering Data, Geometry, Model, Setup, Solution, and Results.
- System C (UIA 20kHz Preload) is connected to Engineering Data, Geometry, Model, Setup, Solution, and Results.

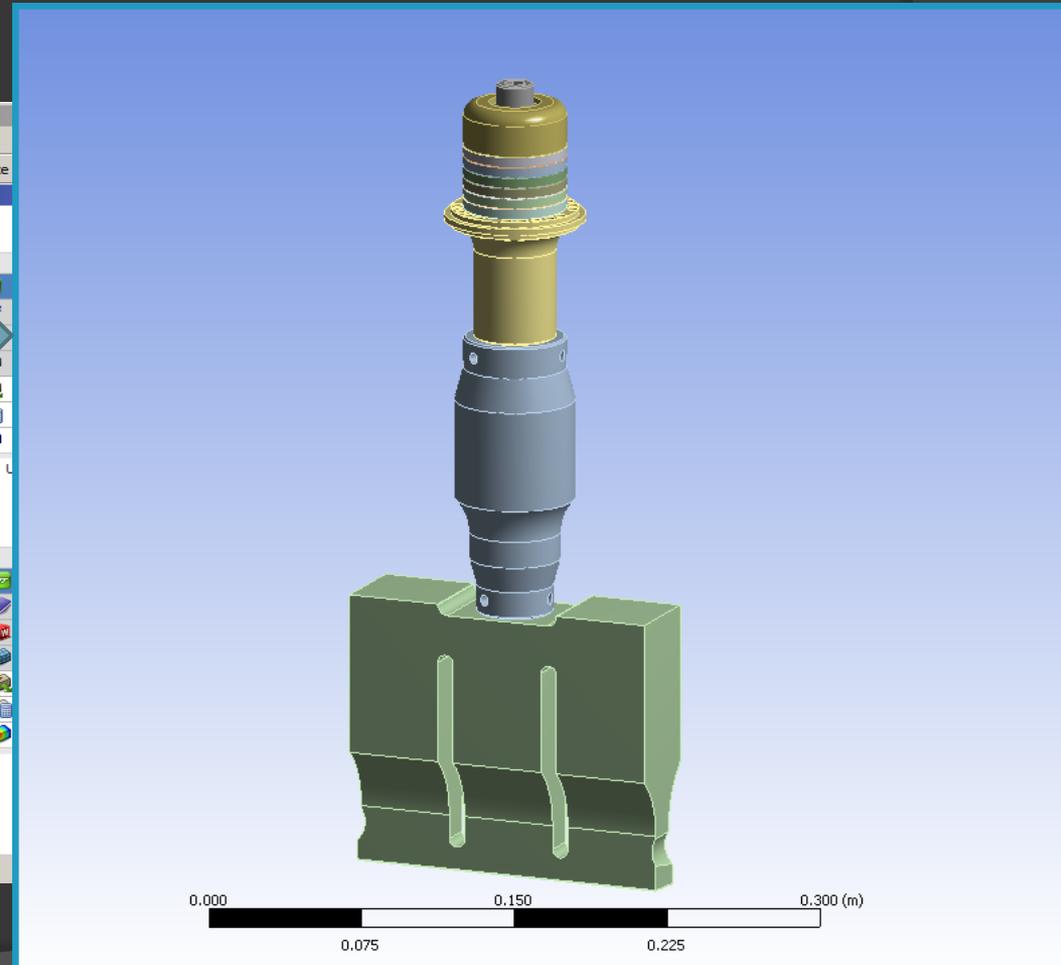
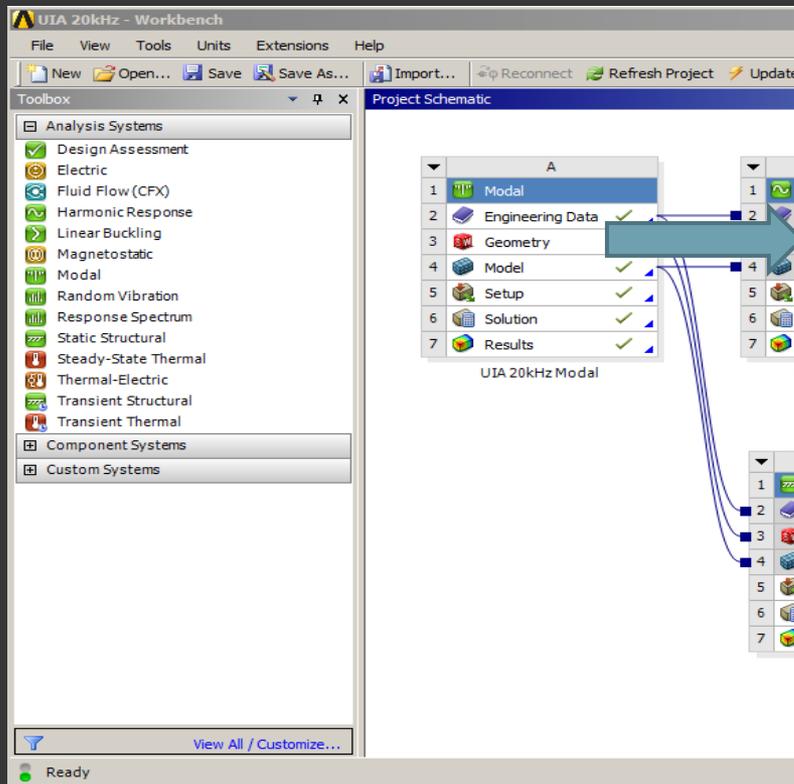
The Properties of Project Schematic table is shown on the right:

	A	B
1	Property	Value
2	Notes	
3	Notes	

# Transducer Design and Modeling

## - ANSYS Workbench, Connection to the Solid Modeling Package -

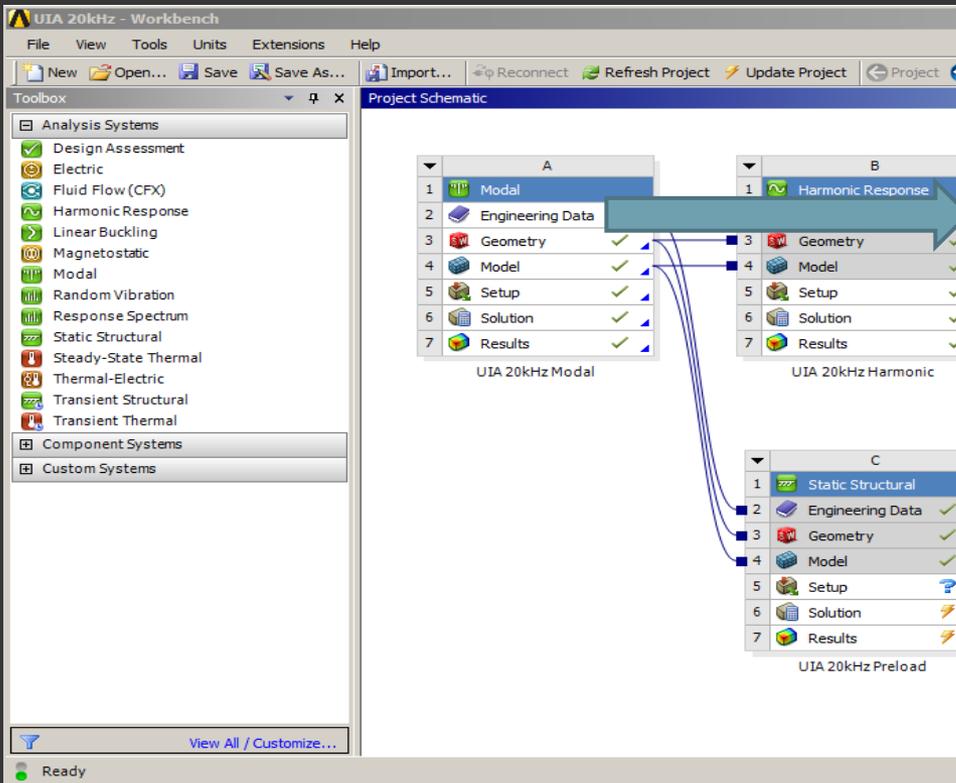
- All three analysis point to the same geometry in solid works
- Changes here affect all three analysis



# Transducer Design and Modeling

## - ANSYS Workbench, Material Library -

- All three analysis point to the same material library
- Changes here affect all three analysis
- Standard acoustic materials only – *does not include Piezo materials*



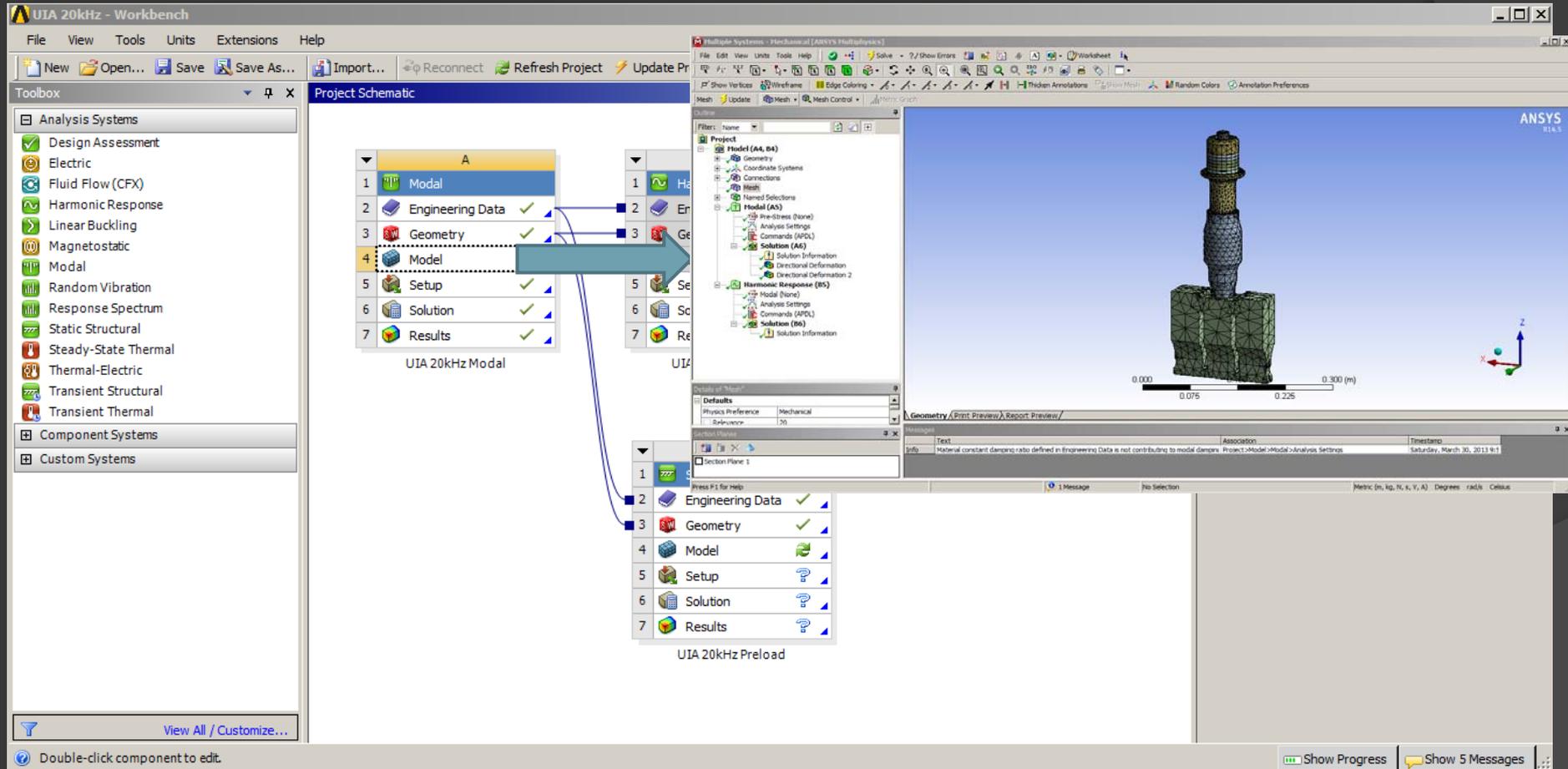
	A	B	C	D
1	Contents of Engineering Data	Source		Description
2	Material			
3	Al 7075-T6			
4	Ceramtec P8			
5	Nickle			
6	Steel			
7	Structural Steel			Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
8	TI 7-4 ATG			
=	Click here to add a new material			

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m <sup>-3</sup>		
3	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's Modulus and...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0.3			
10	Bulk Modulus	1.6667E+11	Pa		
11	Shear Modulus	7.6923E+10	Pa		
12	Alternating Stress Mean Stress	Tabular			
16	Strain-Life Parameters				
24	Tensile Yield Strength	2.5E+08	Pa		
25	Compressive Yield Strength	2.5E+08	Pa		
26	Tensile Ultimate Strength	4.6E+08	Pa		
27	Compressive Ultimate Strength	0	Pa		

# Transducer Design and Modeling

## - ANSYS Workbench, ANSYS Mechanical -

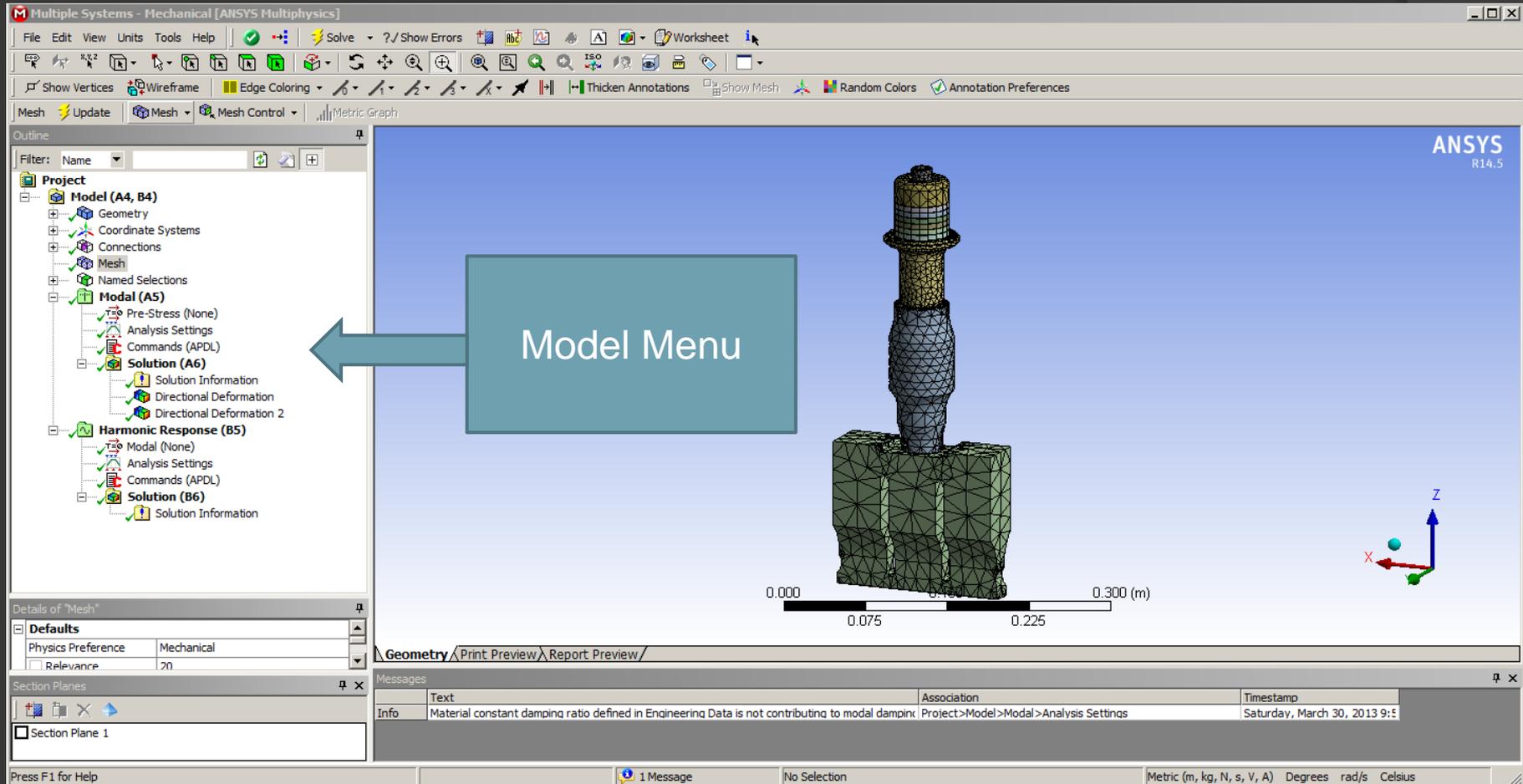
- ANSYS finite element interface
- Meshing, naming, apply BC's, run, examine results
- Modal and harmonic model connected, static preload separate



# Transducer Design and Modeling

## - ANSYS Workbench, ANSYS Mechanical -

- ANSYS finite element interface
- Meshing, naming, apply BC's, run, examine results
- Model menu



# Transducer Design and Modeling

## - ANSYS Workbench, Set Up Challenges for Ultrasonic Analysis -

- Piezo materials do not exist in standard model library
- Need to input into ANSYS using APDL programming language
- Use APDL for material definition, and boundary conditions, special piezo results

The screenshot displays the ANSYS Workbench interface for a mechanical analysis. The main window shows a 3D model of a transducer with a coordinate system (X, Y, Z) and dimensions of 0.300 (m) and 0.225. The Outline tree on the left is highlighted with a blue box, showing the following structure:

- Project
  - Model (A4, B4)
    - Geometry
    - Coordinate Systems
    - Connections
    - Mesh
    - Named Selections
    - Modal (A5)
      - Pre-Stress (None)
      - Analysis Settings
      - Commands (APDL)
      - Solution (A6)
        - Solution Information
        - Directional Deformation
        - Directional Deformation 2
    - Harmonic Response (B5)
      - Modal (None)
      - Analysis Settings
      - Commands (APDL)
      - Solution (B6)
        - Solution Information

A blue text box on the right contains the text "Input APDL Code for Modal and Harmonic". A blue arrow points from this text box to the "Commands (APDL)" nodes in the Outline tree, specifically to the "Commands (APDL)" node under "Modal (A5)" and "Harmonic Response (B5)".

ANSYS R14.5

Timestamp: Saturday, March 30, 2013 9:5

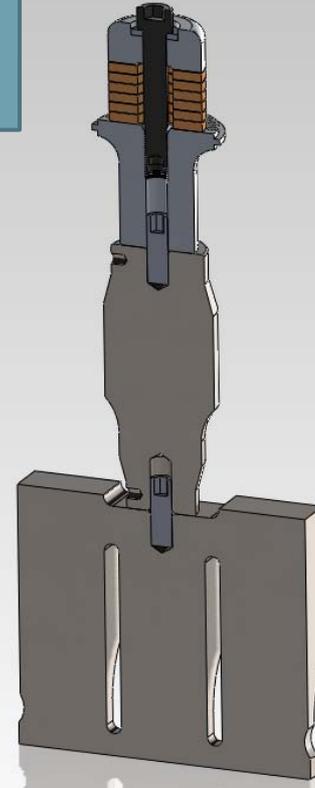
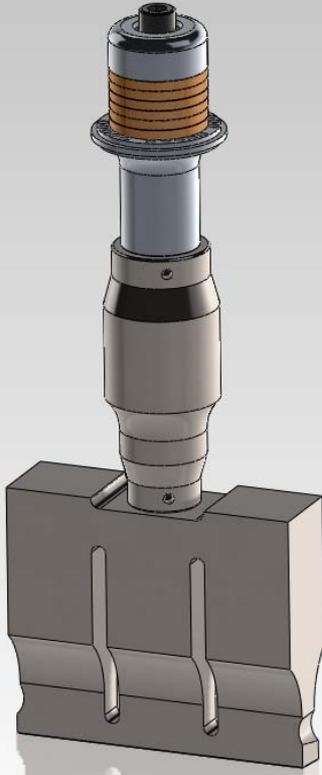
Press F1 for Help | 1 Message | No Selection | Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

# Transducer Design and Modeling

## - Converter Stack Solid Model, Components -

- One common model for all analysis
- All stack components Modeled

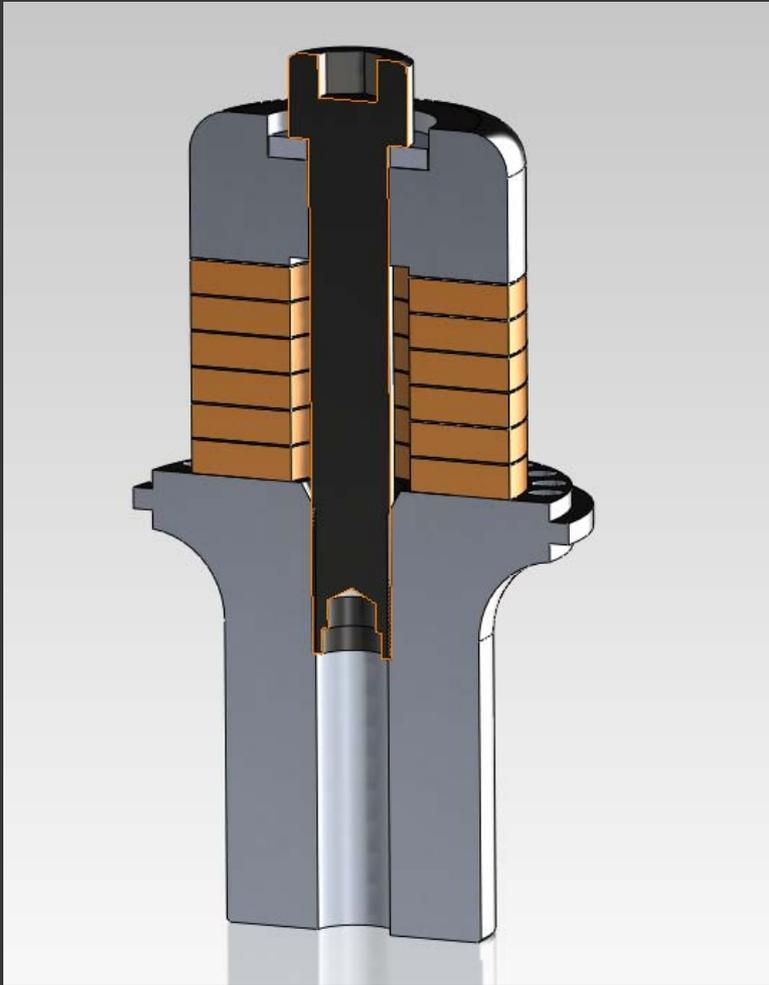
Generally  
Components Modeled  
Line to Line



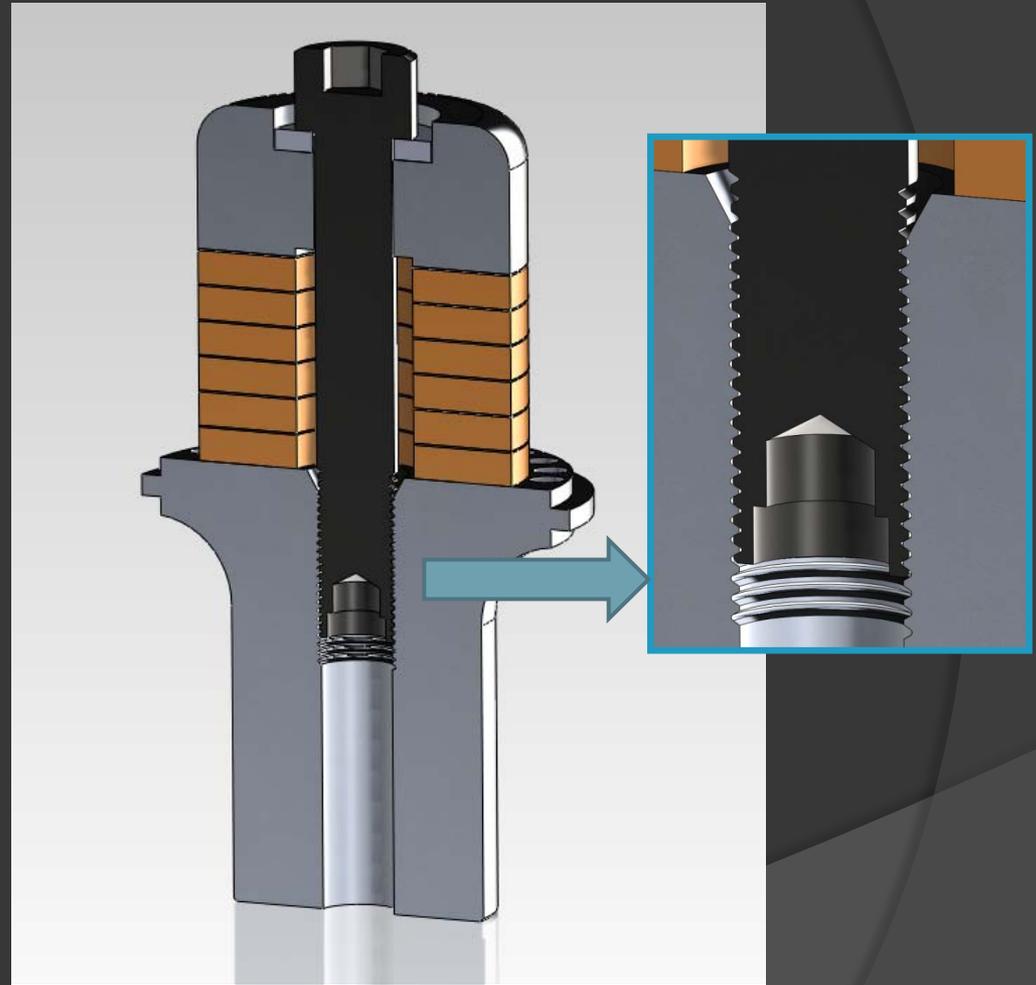
# Transducer Design and Modeling

## - Converter Stack Solid Model, Modeling Techniques -

- Use solid modeling configuration utility to include or remove analysis details



Non Thread Configuration



Thread Configuration

# Transducer Design and Modeling

## - Model Example - Modal Analysis -

- All model setup done in ANSYS Mechanical
- 2 of the three analysis are connected (modal and harmonic). Both will use the same meshed model (Preload is separate)

The screenshot displays the ANSYS Workbench interface for a multi-system model. The Project Schematic on the left shows a hierarchy of components:

- A** (Parent)
  - 1 Modal
  - 2 Engineering Data
  - 3 Geometry
  - 4 Model
  - 5 Setup
  - 6 Solution
  - 7 Results

The **Model** component (4) is expanded to show its sub-components:

- 1 Modal (A5)
- 2 Harmonic Response (B5)
- 3 Preload (C5)

The **Harmonic Response** component (2) is further expanded to show:

- 1 Solution (A6)
- 2 Harmonic Response (B5)
- 3 Solution (B6)

The **Preload** component (3) is expanded to show:

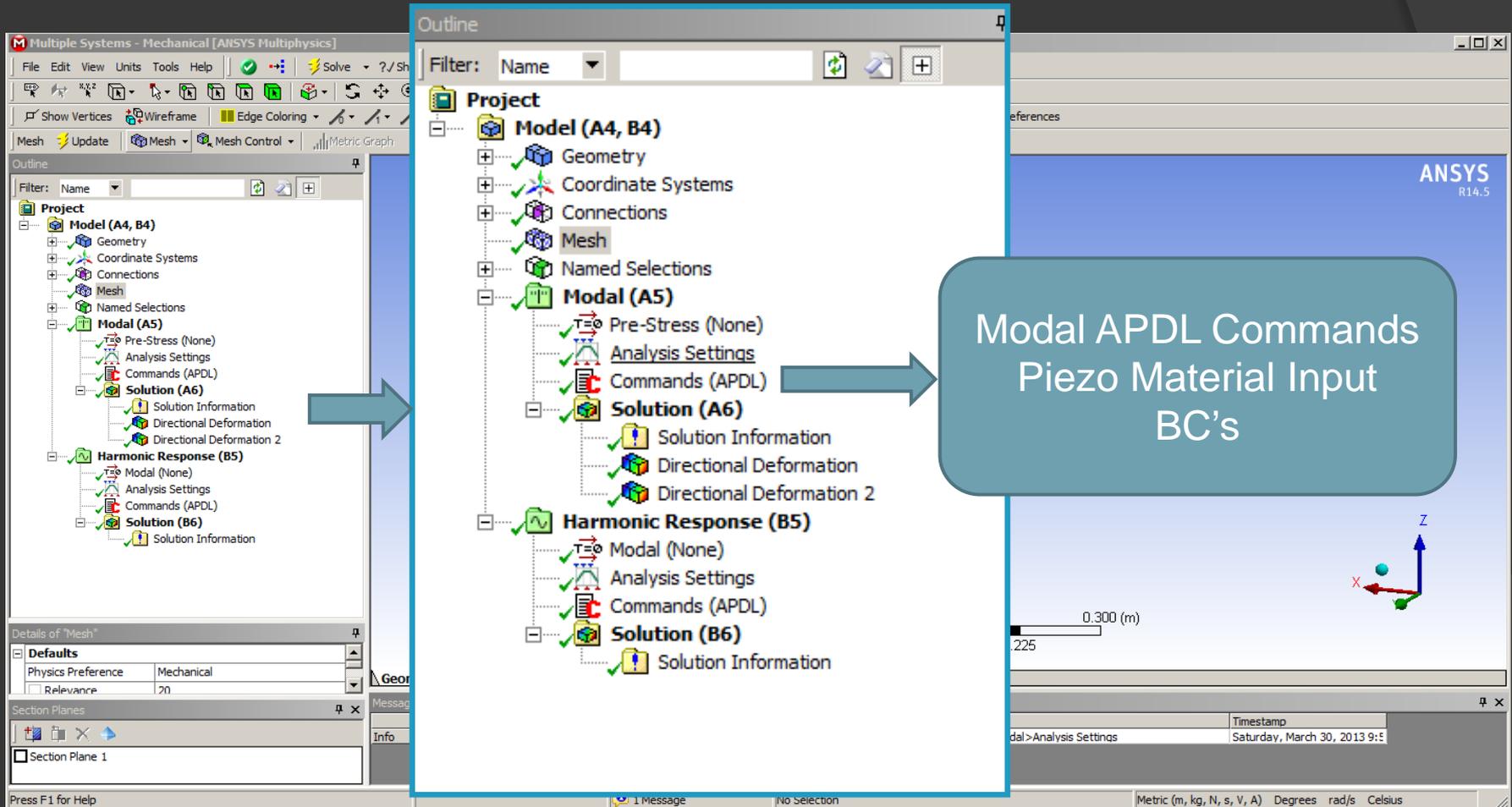
- 1 Engineering Data
- 2 Geometry
- 3 Model
- 4 Setup
- 5 Solution
- 6 Results

The main window displays a 3D model of the transducer with a mesh. The status bar at the bottom shows "Show Progress" and "Show 5 Messages".

# Transducer Design and Modeling

## - Modal Analysis, Use of APDL programming Language -

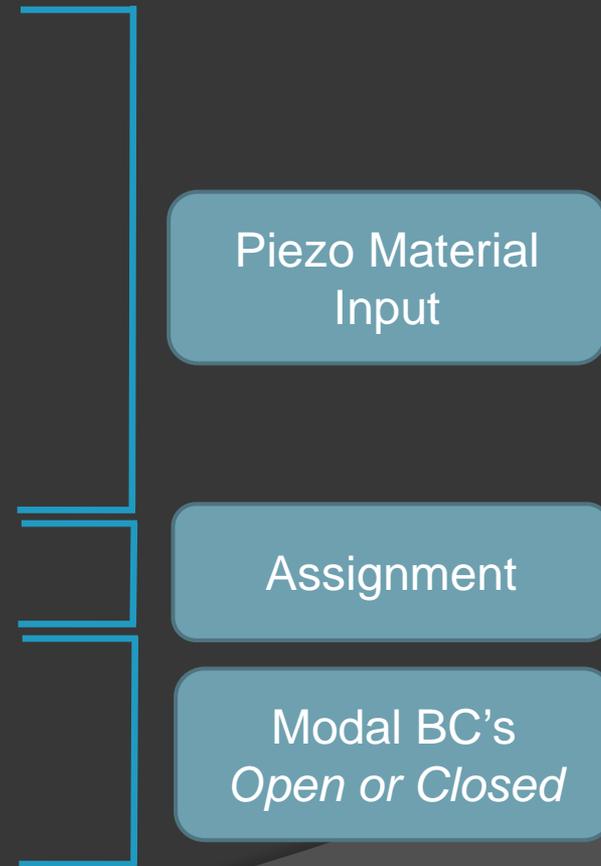
- All model setup done in ANSYS Mechanical
- Standard model setup for geometry, connections, mesh, modal setup
- Insert APDL command under Modal (A5) for Piezo input



# Transducer Design and Modeling

## - Modal Analysis, Set-up of piezo materials / assignments -

- ANSYS APDL Code for a modal analysis
- Material input, assignment, Modal BC's



# Transducer Design and Modeling

## - Modal Analysis, Set-up of piezo materials / assignments -

- Electrical / mechanical performance in high end FEA packages (ex ANSYS, Abaqus) is analyzed using coupled field elements.
- 3 input material matrices required – Piezo elasticity, piezo stress matrix, piezo permativity matrix

$$\{T\} = [c^E]\{S\} - [e]\{E\}$$

$$\{D\} = [e]^T\{S\} + [\epsilon^S]\{E\}$$

or equivalently

$$\begin{Bmatrix} \{T\} \\ \{D\} \end{Bmatrix} = \begin{bmatrix} [c^E] & [e] \\ [e]^T & -[\epsilon^S] \end{bmatrix} \begin{Bmatrix} \{S\} \\ -\{E\} \end{Bmatrix}$$

where:

$\{T\}$  = stress vector (referred to as  $\{\sigma\}$  elsewhere in this manual)

$\{D\}$  = electric flux density vector

$\{S\}$  = strain vector (referred to as  $\{\epsilon\}$  elsewhere in this manual)

$\{E\}$  = electric field intensity vector

$[c^E]$  = elasticity matrix (evaluated at constant electric field (referred to as  $[D]$  elsewhere in this manual))

$[e]$  = piezoelectric stress matrix

$[\epsilon^S]$  = dielectric matrix (evaluated at constant mechanical strain)

$$[c] = \begin{bmatrix} c_{11} & c_{12} & c_{13} & c_{14} & c_{15} & c_{16} \\ & c_{22} & c_{23} & c_{24} & c_{25} & c_{26} \\ & & c_{33} & c_{34} & c_{35} & c_{36} \\ \text{Symmetric} & & & c_{44} & c_{45} & c_{46} \\ & & & & c_{55} & c_{56} \\ & & & & & c_{66} \end{bmatrix}$$

Piezo Elasticity

$$[e] = \begin{bmatrix} e_{11} & e_{12} & e_{13} \\ e_{21} & e_{22} & e_{23} \\ e_{31} & e_{32} & e_{33} \\ e_{41} & e_{42} & e_{43} \\ e_{51} & e_{52} & e_{53} \\ e_{61} & e_{62} & e_{63} \end{bmatrix}$$

Piezo Stress

$$[\epsilon^S] = \begin{bmatrix} \epsilon_{11} & 0 & 0 \\ 0 & \epsilon_{22} & 0 \\ 0 & 0 & \epsilon_{33} \end{bmatrix}$$

Piezo Perm

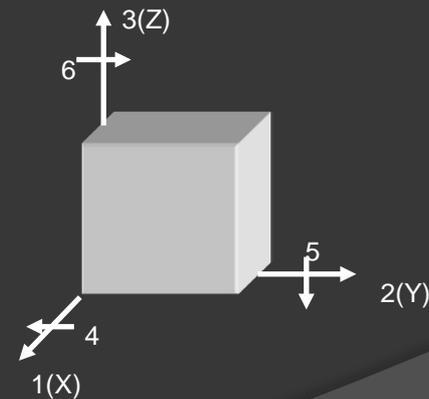
# Transducer Design and Modeling

## - Modal Analysis, Set-up of piezo materials / assignments -

- Piezoelectric Material properties gathered from manufacturers catalogs
  - Need  $d_{33}$ ,  $d_{31}$ ,  $d_{15}$ ,  $d_{32}$ ,  $d_{24}$  (d matrix)
  - Need  $s_{11E}$ ,  $s_{33E}$ ,  $s_{44E}$ ,  $s_{12E}$ ,  $s_{13E}$ ,  $s_{23E}$ ,  $s_{55E}$ ,  $s_{66E}$  (compliance)
  - Need  $e_{11S}$ ,  $e_{22S}$ ,  $e_{33S}$  (permittivity)

### Useful Relations for Catalog Values

- $d_{33}$  and  $d_{31}$  need to account for preload
- $d_{32} = d_{31}$
- $d_{24} = d_{15}$
- $s_{23E} = s_{31E}$
- $s_{55E} = s_{44E}$
- $s_{66E} = 2(s_{11E} - s_{13E})$



**Axis System for Matrix Calculations  
Catalog Values**

# Transducer Design and Modeling

## - Modal Analysis, Set-up of piezo materials / assignments -

- Material compliance input [s] and Piezoelectric Matrix Input [d] (ANSYS and ABAQUS)
- Some mathematical manipulation is required (Matrix operations, ANSYS shear terms adjusted)

**[s]<sup>E</sup> Compliance Matrix (m<sup>2</sup>/N) - Closed Circuit**

1.14E-11	-4.80E-12	-3.70E-12	0	0	0
-4.80E-12	1.37E-11	-4.80E-12	0	0	0
-3.70E-12	-4.80E-12	1.14E-11	0	0	0
0	0	0	3.19E-11	0	0
0	0	0	0	3.24E-11	0
0	0	0	0	0	3.19E-11

**[d] Piezoelectric Matrix (C/N)**

0	0	0	0	0	3.80E-10
-9.50E-11	2.50E-10	-9.50E-11	0	0	0
0	0	0	3.80E-10	0	0

**[c]<sup>E</sup> Stiffness Matrix (N/m<sup>2</sup>) - Closed Circuit**

1.484E+11	8.079E+10	8.219E+10	0	0	0
8.079E+10	1.296E+11	8.079E+10	0	0	0
8.219E+10	8.079E+10	1.484E+11	0	0	0
0	0	0	3.135E+10	0	0
0	0	0	0	3.086E+10	0
0	0	0	0	0	3.135E+10

**[d] Piezoelectric Matrix Transposed (C/N)**

0	-9.5E-11	0
0	2.5E-10	0
0	-9.5E-11	0
0	0	3.8E-10
0	0	0
3.8E-10	0	0

**[d] \* [c] = Piezoelectric Matrix (C/m<sup>2</sup>)**

0	-1.71	0
0	17.05	0
0	-1.71	0
0	0	11.91
0	0	0
11.91	0	0

### ANSYS Input

**[c] Stiffness Matrix (N/m<sup>2</sup>) - Closed Circuit**

1.484E+11	8.079E+10	8.219E+10	0	0	0
8.079E+10	1.296E+11	8.079E+10	0	0	0
8.219E+10	8.079E+10	1.484E+11	0	0	0
0	0	0	3.135E+10	0	0
0	0	0	0	3.135E+10	0
0	0	0	0	0	3.086E+10

Note: Shear terms adjusted for ANSYS input

### ANSYS Input

**[d] \* [c] = Piezoelectric Matrix (C/m<sup>2</sup>)**

0	-1.71	0
0	17.05	0
0	-1.71	0
11.91	0	0
0	0	11.91
0	0	0

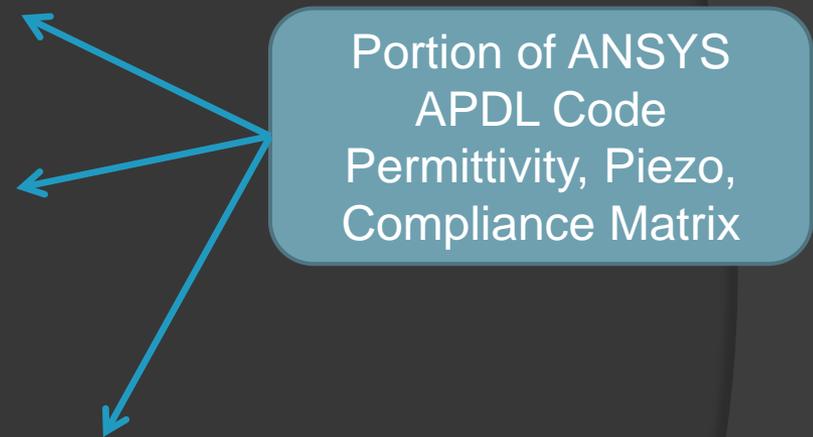
Note: Shear terms adjusted for ANSYS input

## Note on Shear Term Manipulation From the ANSYS Manual

For most published piezoelectric materials, the order used for the piezoelectric matrix is x, y, z, yz, xz, xy, based on IEEE standards (see ANSI/IEEE Standard 176–1987), while the ANSYS input order is x, y, z, xy, yz, xz as shown above. This means that you need to transform the matrix to the ANSYS input order by switching row data for the shear terms.

## Transducer Design and Modeling

- Modal Analysis - Modal Analysis, Set-up of piezo materials / assignments -
  - ANSYS APDL Code for piezo materials



# Transducer Design and Modeling

- Modal Analysis - Modal Analysis, Set-up of piezo materials / assignments -
  - Assign materials using ANSYS Mechanical Named Selections

Use Named Selections in ANSYS Mechanical

Portion of APDL Code Assigning Piezo Materials to a Named Selection

ANSYS R14.5

Multiple Systems - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help Solve ?/ Show Errors Worksheet

Show Vertices Wireframe Edge Color Show Mesh Random Colors Annotation Preferences

Mesh Update Mesh Mesh Cont

Outline

Filter: Name

Project

- Model (A4, B4)
  - Geometry
  - Coordinate Systems
  - Connections
  - Mesh
    - Named Selection
      - CERAMIC
      - POSITIVE
      - NEGATIVE
  - Modal (A5)
    - Pre-Stress (None)
    - Analysis Settings
    - Commands (APDL)
  - Solution (A6)
    - Solution Information
    - Directional Deformation
    - Directional Deformation 2
  - Harmonic Response (B5)
    - Modal (None)
    - Analysis Settings
    - Commands (APDL)
  - Solution (B6)
    - Solution Information

Details of "Mesh"

Defaults

Physics Preference Mechanical

Relevance 20

Section Planes

Section Plane 1

Geometry Print Preview Report Preview

Messages

Text

Info Material constant damping ratio defined in Engineering Data is not co

Timestamp

Saturday, March 30, 2013 9:5

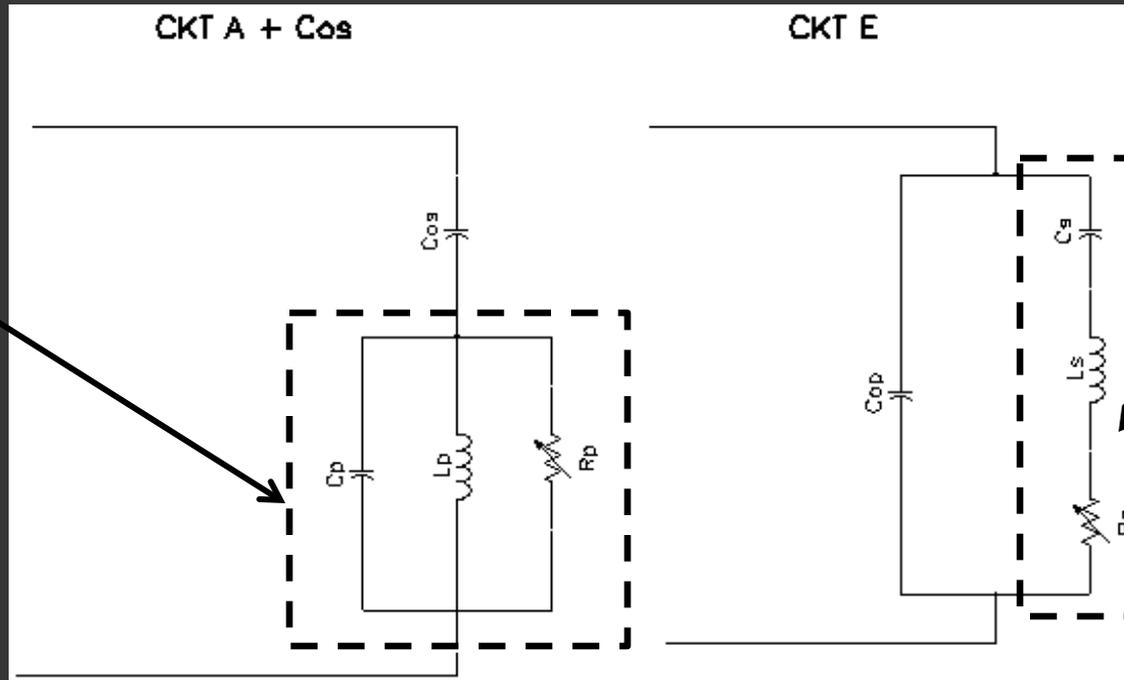
Press F1 for Help 1 Message No Selection Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

# Transducer Design and Modeling

## - Modal Analysis, Theory – Series and Parallel Analysis -

- Transducer Equivalent Circuit
- The transducer system can be modeled as a lumped parameter system
- Either circuit model below is valid for modeling a transducer using lumped parameters

transducer modeled as a parallel circuit



transducer modeled as a series circuit

$C_{os}/C_{op}$  = transducer capacitance

$C_s/C_p$  = transducer stiffness

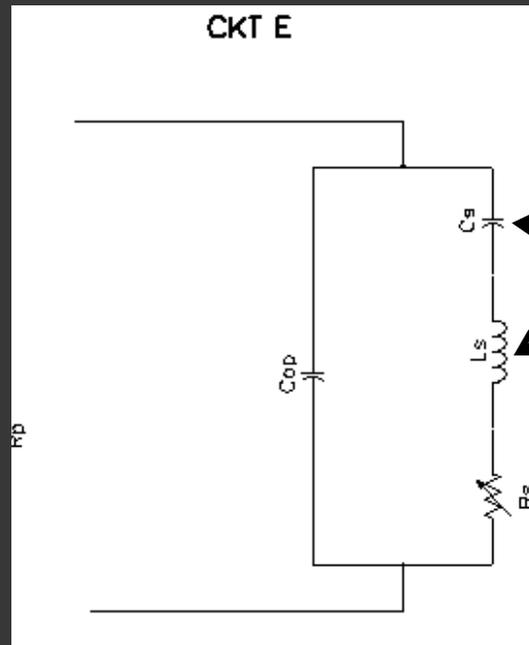
$L_s/L_p$  = transducer mass

$R_s/R_p$  = transducer loss (load)

# Transducer Design and Modeling

## - Modal Analysis, Theory – Series and Parallel Analysis -

- Series Resonance is the frequency where  $C_s$  resonates with  $L_s$
- $C_s$  and  $L_s$  effectively cancel, creating a short circuit (when  $R_s = 0$ ). This creates maximum current for minimum voltage = low impedance point
- This is called series resonance, short circuit resonance, or mechanical resonance
- In series resonant systems, converter amplitude is proportional to current (In the motional branch)
- In a load varying system, series resonant systems usually employ some sort of current control



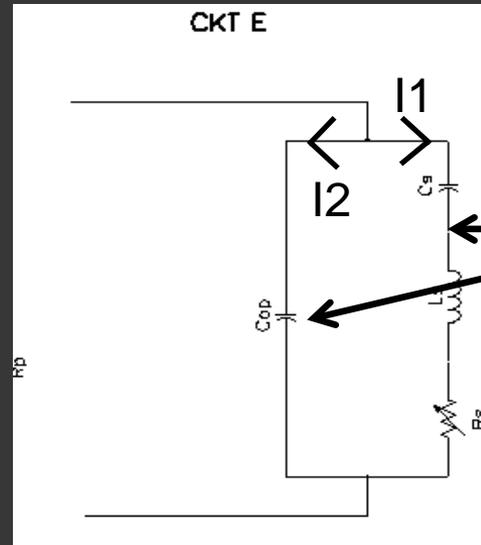
These reactive components cancel. Creates a short circuit when  $R_s=0$

$$F_s = 1/2\pi \sqrt{1/(L_s C_s)}$$

# Transducer Design and Modeling

## - Modal Analysis, Theory – Series and Parallel Analysis -

- Parallel Resonance is the frequency where the Cs/Ls combination resonates with Cop
- The I1 and I2 currents are equal and opposite in phase. They effectively cancel each other. No net current flows (when R=0). Circuit now looks like an open circuit.
- This is called parallel resonance, or open circuit resonance
- In parallel resonant systems, transducer amplitude is proportional to voltage (In the motional branch)
- In a load varying system, parallel resonant systems usually employ some sort of voltage control



Cs/Ls  
combination  
resonates with  
Cop

$$F_p = 1/2\pi \sqrt{1/L_s C_s + (1 + C_s/C_o)}$$

# Transducer Design and Modeling

- Modal Analysis, Boundary conditions – series and parallel analysis -
  - ANSYS APDL Code, Apply BC's for parallel or series modal analysis

Use Named Selections in ANSYS Mechanical for Ceramic Electrodes

Portion of APDL Code Assigning VOLT BC's for Series or Parallel Resonance *Open or Closed*

ANSYS R14.5

Project

- Model (A4, B4)
  - Geometry
  - Coordinate Systems
  - Connections
  - Mesh
    - Named Selection
      - CERAMIC
      - POSITIVE
      - NEGATIVE
  - Modal (A5)
    - Pre-Stress (None)
    - Analysis Settings
    - Commands (APDL)
  - Solution (A6)
    - Solution Information
    - Directional Deformation
    - Directional Deformation 2
  - Harmonic Response (B5)
    - Modal (None)
    - Analysis Settings
    - Commands (APDL)
  - Solution (B6)
    - Solution Information

Details of "Mesh"

Defaults

Physics Preference: Mechanical

Relevance: 20

Section Planes

Section Plane 1

Messages

Text	Association	Timestamp
Info	Material constant damping ratio defined in Engineering Data is not contributing to modal dampin	Project>Model>Modal>Analysis Settings
		Saturday, March 30, 2013 9:5

Press F1 for Help

1 Message

No Selection

Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

# Transducer Design and Modeling

- Modal Analysis, Boundary conditions – series and parallel analysis -
  - ANSYS APDL Code, Apply BC's for parallel or series modal analysis

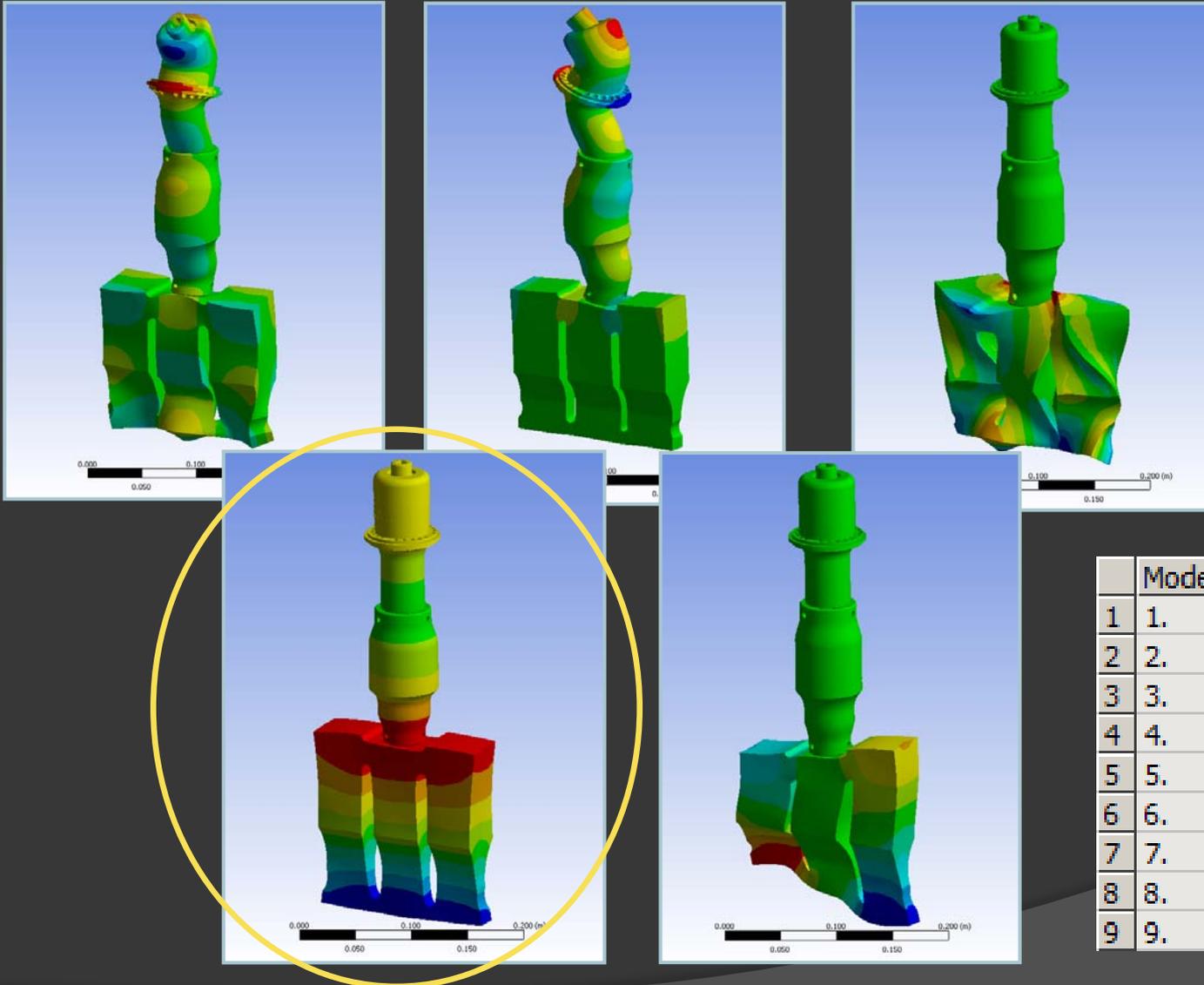
The screenshot displays the ANSYS Multiphysics interface. The main window shows a 3D model of a cylindrical transducer with a central hole, rendered in a light blue color. The model is composed of several stacked layers. Three blue callout boxes with white text and arrows point to specific parts of the model:

- Named Component: CERAMIC (bodies)**: Points to the central cylindrical body of the transducer.
- Named Component: POSITIVE (Top surfaces)**: Points to the top surface of the transducer.
- Named Component: NEGATIVE (Bot Surfaces)**: Points to the bottom surface of the transducer.

The software interface includes a menu bar (File, Edit, View, Units, Tools), a toolbar, and a tree view on the left. The tree view shows the project structure, including a 'Model (A4, B4)' and a 'Geometry' folder containing various components like 'Stud-2', 'U502-1@U502\_CERAMIC-5', 'U502-1@U502\_CERAMIC-2', 'U502-1@U502\_ELECTRODE-1', 'U502-1@U502\_CERAMIC-1', 'U502-1@U502\_FD-1', 'U502-1@U502\_CERAMIC-6', 'U502-1@U502\_ELECTRODE-6', 'U502-1@U502\_BOLT-1', 'U502-1@U502\_ELECTRODE-4', 'U502-1@U502\_CERAMIC-4', 'U502-1@U502\_ELECTRODE-5', 'U502-1@U502\_WASHER-1', 'U502-1@U502\_BD-1', 'U502-1@U502\_ELECTRODE-3', 'U502-1@U502\_CERAMIC-3', 'U502-1@U502\_ELECTRODE-2', and 'booster\_011-600-031-1'. The 'Details of Multiple Selection' panel is open, showing 'Graphics Properties' (Definition: Suppressed, Stiffness Behavior, Coordinate System, Reference Temperature) and 'Material' (Assignment: Ceramtec P8, Nonlinear Effects, Thermal Strain Effects). The 'Section Planes' panel shows 'Section Plane 1'. The 'Messages' panel at the bottom shows a warning message: 'Material constant damping ratio defined in Engineering Data is not contributing to modal damping'. The status bar at the bottom indicates 'Multiple Selection (6 Objects Selected)', '1 Message', and 'No Selection'.

# Transducer Design and Modeling

## - Modal Analysis, Examining results -

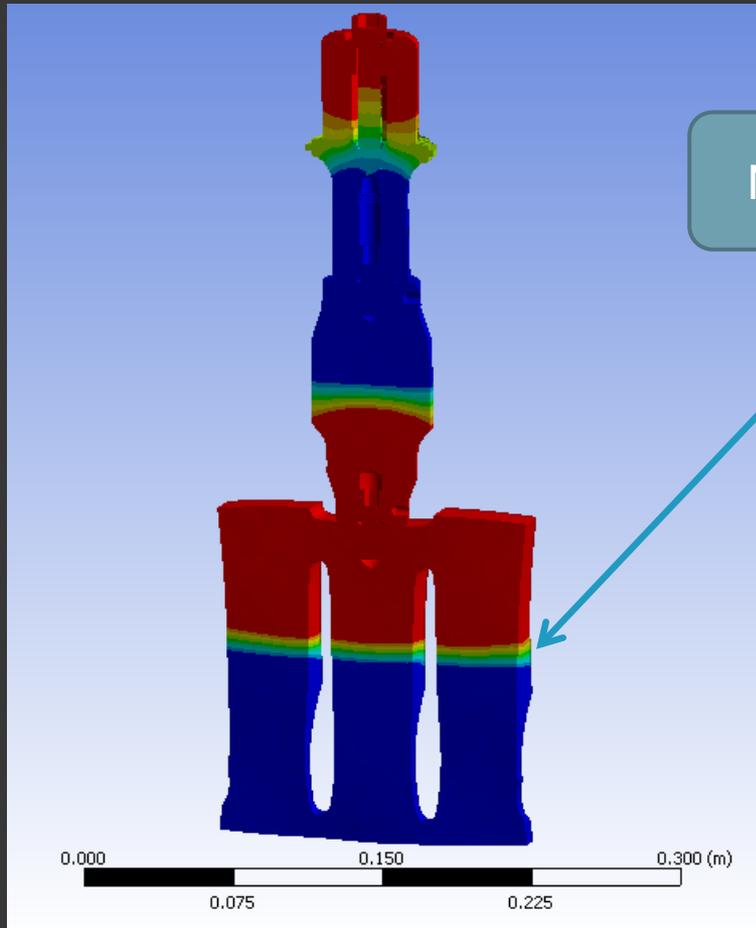


	Mode	<input checked="" type="checkbox"/> Frequency [Hz]
1	1.	18014
2	2.	18318
3	3.	19482
4	4.	19654
5	5.	20107
6	6.	20147
7	7.	21048
8	8.	21324
9	9.	21918

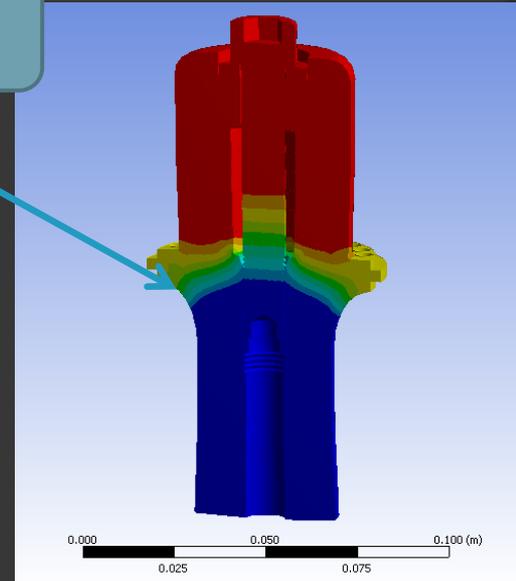
Axial Resonance

# Transducer Design and Modeling

## - Modal Analysis, Examining results -



Node Locations



# Transducer Design and Modeling

## - Harmonic Analysis, Use of APDL programming Language -

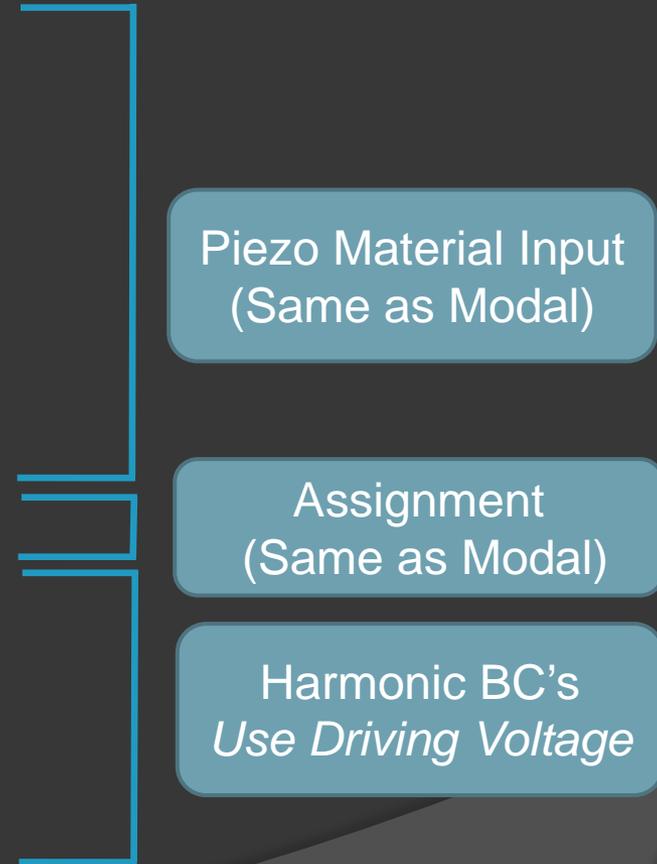
- All model setup done in ANSYS Mechanical
- Standard model setup for geometry, connections, mesh, modal setup
- Insert APDL command under Harmonic Response (B5) for Piezo input

The image shows a screenshot of the ANSYS Mechanical software interface. The main window displays the Outline tree on the left, which lists the model components: Project, Model (A4, B4), Geometry, Coordinate Systems, Connections, Mesh, Named Selections, Modal (A5), Pre-Stress (None), Analysis Settings, Commands (APDL), Solution (A6), Solution Information, Directional Deformation, Directional Deformation 2, Harmonic Response (B5), Modal (None), Analysis Settings, Commands (APDL), Solution (B6), and Solution Information. A blue callout box on the right contains the text: "Harmonic APDL Commands Piezo Material Input BC's". A blue arrow points from the callout box to the "Commands (APDL)" entry under the "Harmonic Response (B5)" component in the Outline tree. Another blue arrow points from the "Commands (APDL)" entry in the Outline tree to the callout box. The background shows a 3D model of a mechanical part with a mesh.

# Transducer Design and Modeling

## - Harmonic Analysis, Use of APDL programming language -

- ANSYS APDL Code for a harmonic analysis
- Material input, assignment, harmonic BC's



# Transducer Design and Modeling

## - Harmonic Analysis, Boundary Conditions -

- ANSYS APDL Code, Apply BC's for harmonic analysis

Use Named Selections in ANSYS Mechanical for Ceramic / Electrodes

Portion of APDL Code Assigning VOLT BC's for Harmonic Analysis Use Driving Voltage

ANSYS R14.5

Project

- Model (A4, B4)
  - Geometry
  - Coordinate Systems
  - Connections
  - Mesh
    - Named Selection
      - CERAMIC
      - POSITIVE
      - NEGATIVE

- Modal (A5)
- Pre-Stress (None)
- Analysis Settings
- Commands (APDL)
- Solution (A6)
- Solution Information
- Directional Deformation
- Directional Deformation 2
- Harmonic Response (B5)
- Modal (None)
- Analysis Settings
- Commands (APDL)
- Solution (B6)
- Solution Information

Details of "Mesh"

Defaults

Physics Preference: Mechanical

Relevance: 20

Section Planes

Section Plane 1

Messages

Text	Association	Timestamp
Info	Material constant damping ratio defined in Engineering Data is not contributing to modal dampin	Project>Model>Modal>Analysis Settings
		Saturday, March 30, 2013 9:5

Press F1 for Help

1 Message

No Selection

Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

# Transducer Design and Modeling

## - Harmonic Analysis, Boundary conditions -

- ANSYS APDL Code, Apply BC's for harmonic analysis

The screenshot displays the ANSYS APDL interface for a mechanical model. The central 3D view shows a green cylindrical component with a central hole, representing a ceramic transducer. Three callout boxes with blue arrows point to specific parts of the model:

- Named Component: CERAMIC (bodies)**: Points to the main body of the transducer.
- Named Component: POSITIVE (Top surfaces)**: Points to the top surface of the transducer.
- Named Component: NEGATIVE (Bot Surfaces)**: Points to the bottom surface of the transducer.

The software interface includes a tree view on the left showing the model hierarchy, a details panel at the bottom left for material properties (Ceramatec P8), and a messages panel at the bottom right.

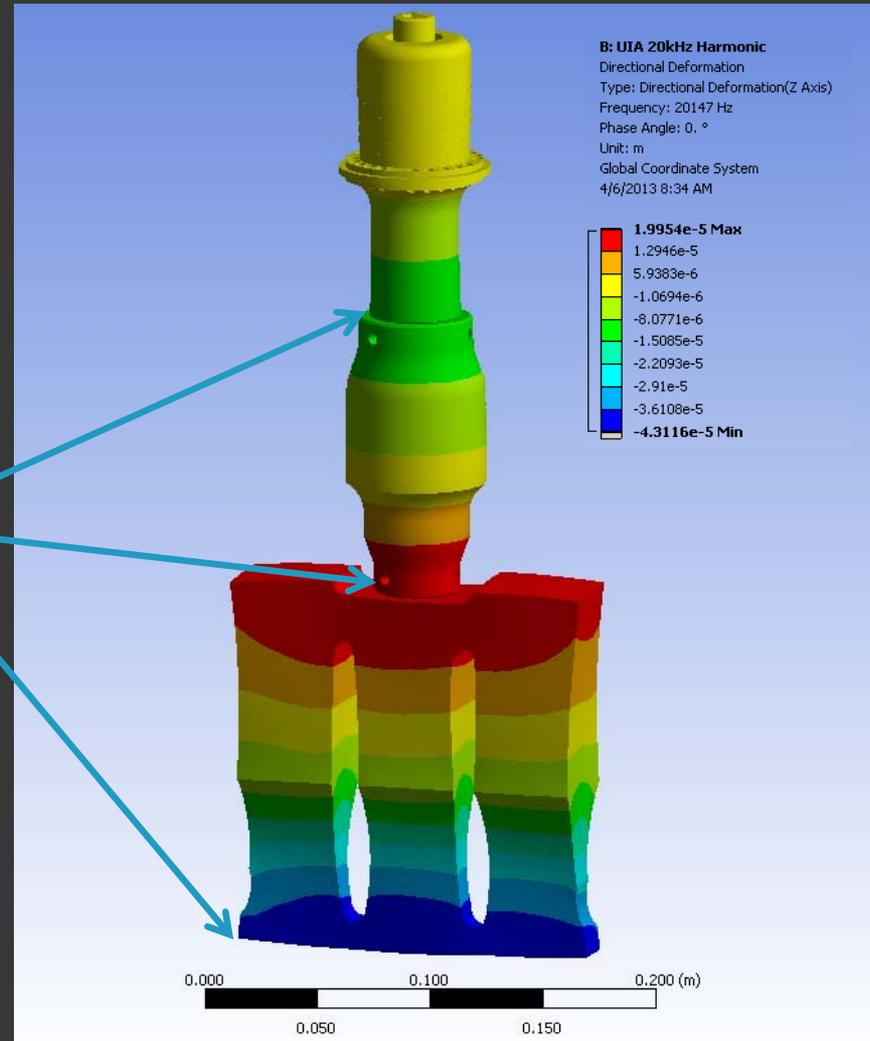
Section	Property	Value
Definition	Suppressed	No
	Stiffness Behavior	Flexible
	Coordinate System	Default Coordinate System
	Reference Temperature	By Environment
Material	Assignment	Ceramatec P8
	Nonlinear Effects	Yes
	Thermal Strain Effects	Yes

# Transducer Design and Modeling

## - Harmonic Analysis, Examining Single Point Results -

- In this example model is run at the parallel operating point

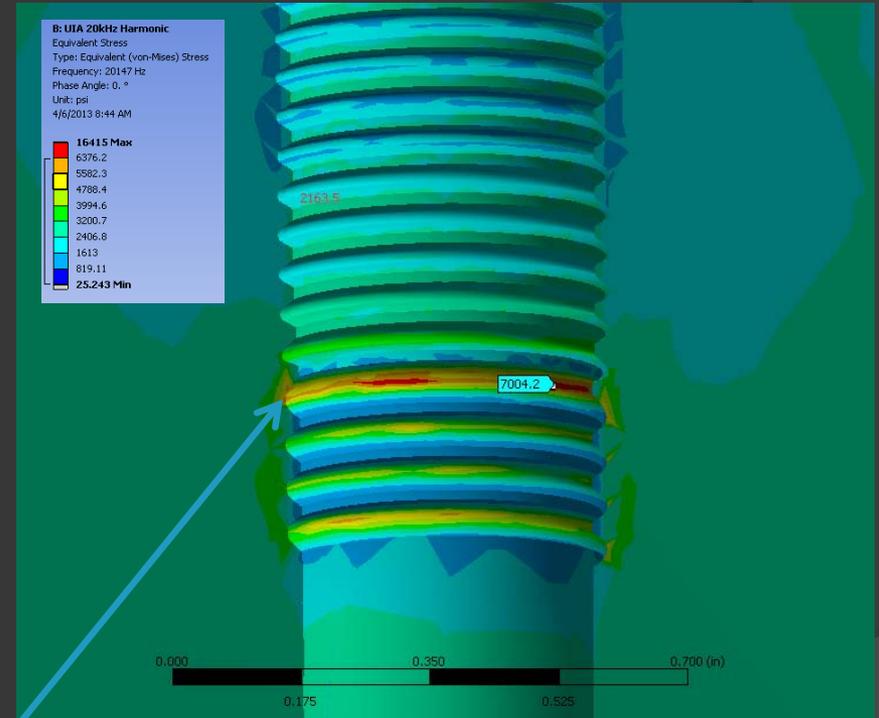
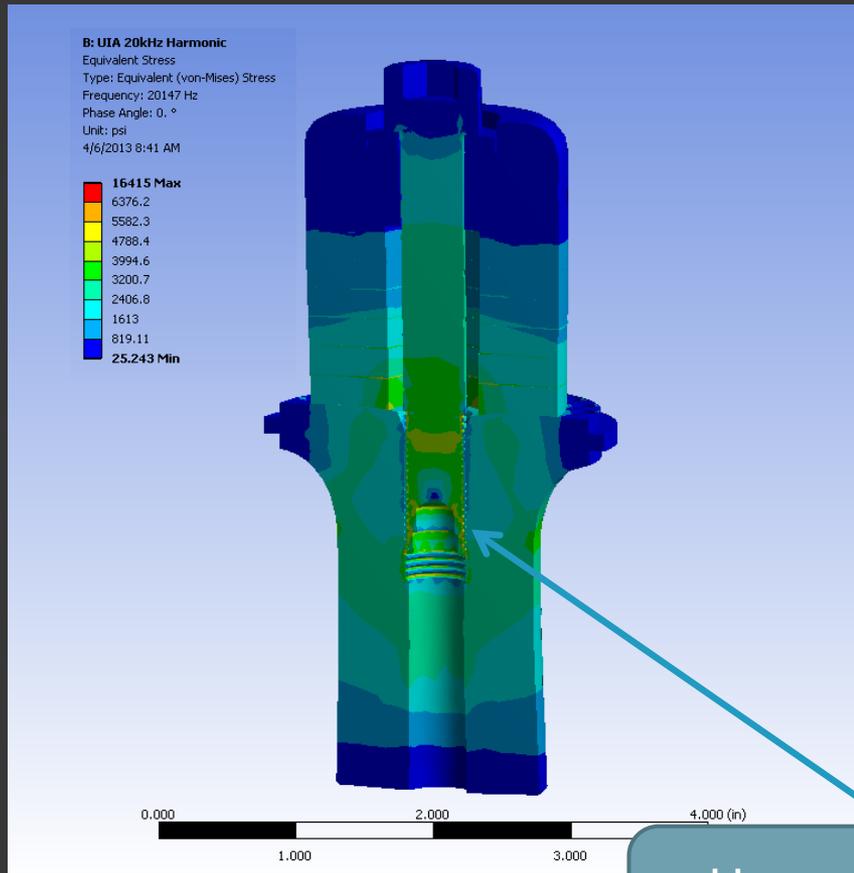
Harmonic Results  
Verify model  
Amplitudes



# Transducer Design and Modeling

## - Harmonic Analysis, Examining Single Point Results -

- In this example model is run at the parallel operating point



Harmonic Results  
Transducer Stress

# Transducer Design and Modeling

## - Harmonic Analysis, Frequency Sweep Analysis -

- By modifying the Analysis Settings a frequency sweep can be performed
- APDL code can be added to examine Impedance

The screenshot displays the ANSYS Multiphysics software interface. The main window shows a 3D model of a transducer with a mesh. The software version is ANSYS R14.5. The interface includes a menu bar (File, Edit, View, Units, Tools, Help), a toolbar, and a command window. The Outline panel on the left shows the project structure:

- Project
  - Model (A4, B4)
    - Geometry
    - Coordinate Systems
    - Connections
    - Mesh
    - Named Selections
  - Modal (A5)
  - Harmonic Response (B5)
    - Modal (None)
    - Analysis Settings
    - Commands (APDL)
  - Solution (B6)
    - Solution Information
    - Directional Deformation
    - Equivalent Stress
    - Commands (APDL)
    - Post Output
    - Post Output 2

The Details of "Mesh" panel shows the following settings:

Defaults	
Physics Preference	Mechanical
<input type="checkbox"/> Relevance	20

The Section Planes panel shows Section Plane 1. The Messages panel is empty. The status bar at the bottom indicates "No Messages", "No Selection", and units: "Metric (m, kg, N, s, V, A) Degrees rad/s Celsius".

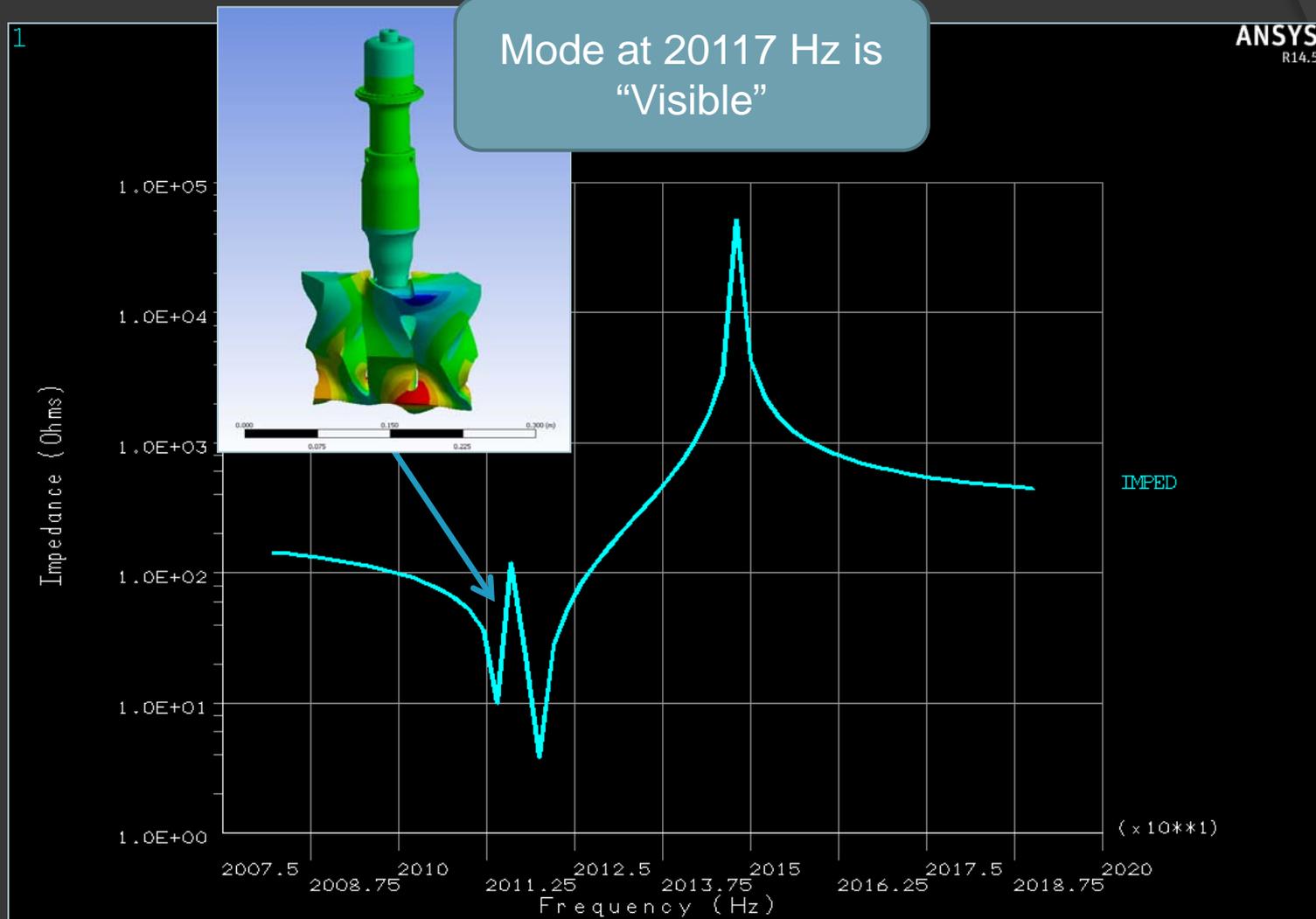
Two callout boxes are present:

- A blue callout box with the text "Modify Analysis Settings to sweep frequency" points to the "Analysis Settings" item in the Outline panel.
- A blue callout box with the text "Add APDL Code to Plot Impedance" points to the "Commands (APDL)" item in the Outline panel.

# Transducer Design and Modeling

## - Harmonic Analysis, Examining Results Frequency Sweep -

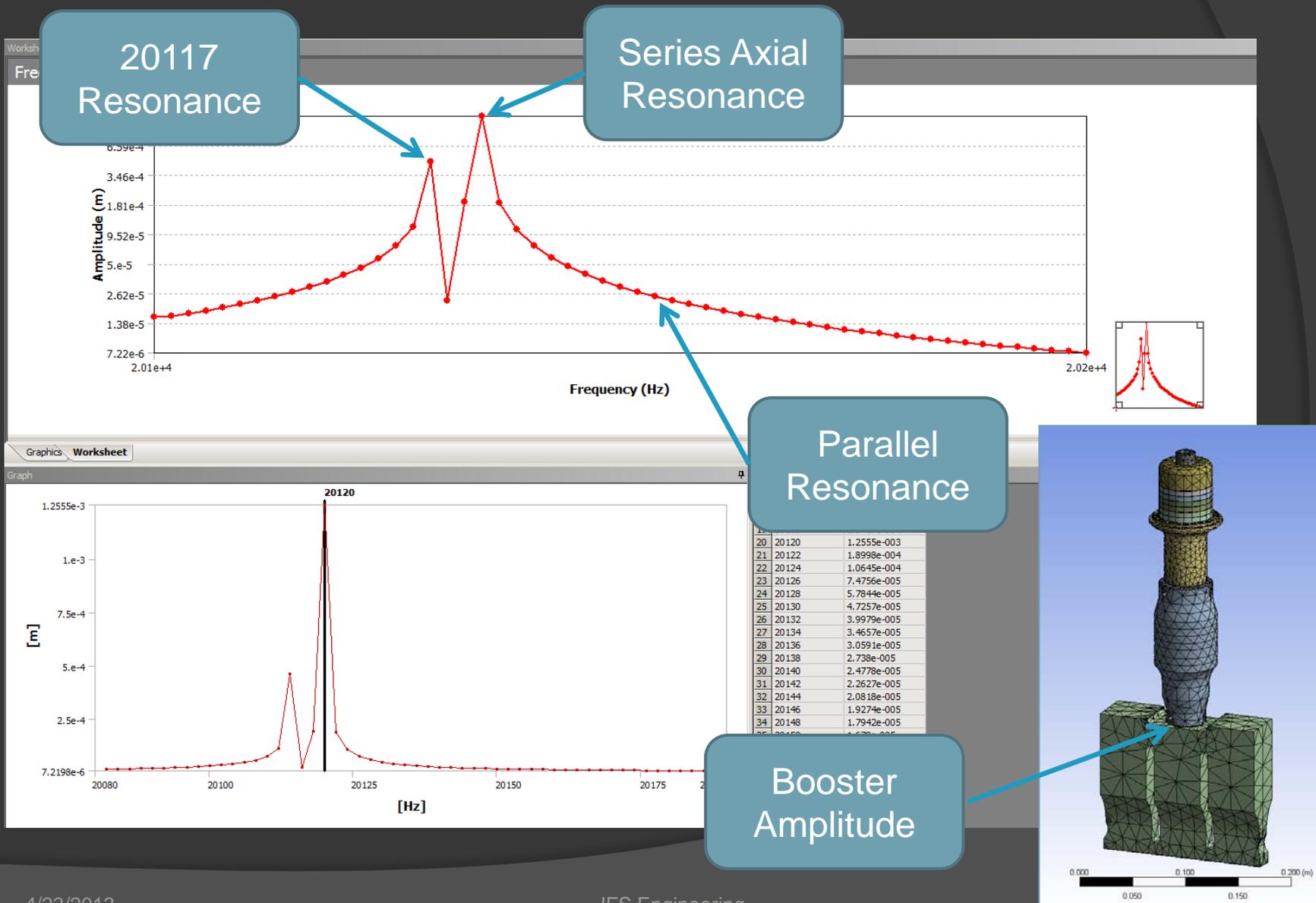
- Graphs of Zmag and Zphase viewed through ANSYS
- Note that mode at 20117 Hz shows up



# Transducer Design and Modeling

## - Harmonic Analysis, Examining Results Frequency Sweep -

- Graphs of mechanical features



# Transducer Design and Modeling

## - Static Analysis, Preload Setup -

- Create Static Analysis in Workbench

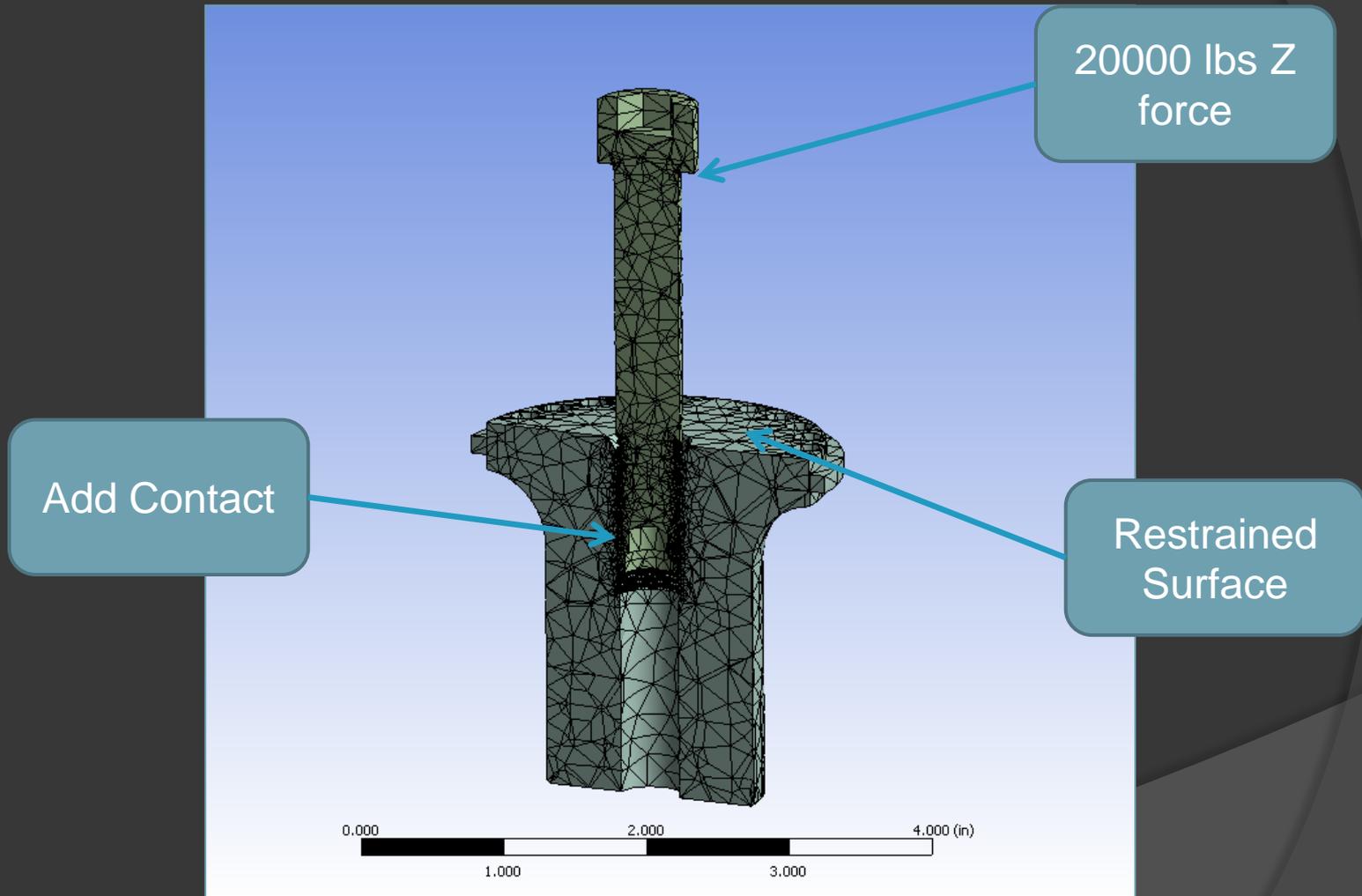
The screenshot displays the ANSYS Workbench interface. The Project Schematic shows three analysis systems: A (Modal), B (Harmonic Response), and C (Static Structural). System C is highlighted with a blue circle and a callout box labeled "Static Preload Analysis". The Properties of Project Schematic panel on the right shows a table with columns A and B, and rows 1, 2, and 3.

	A	B
1	Property	Value
2	Notes	
3	Notes	

# Transducer Design and Modeling

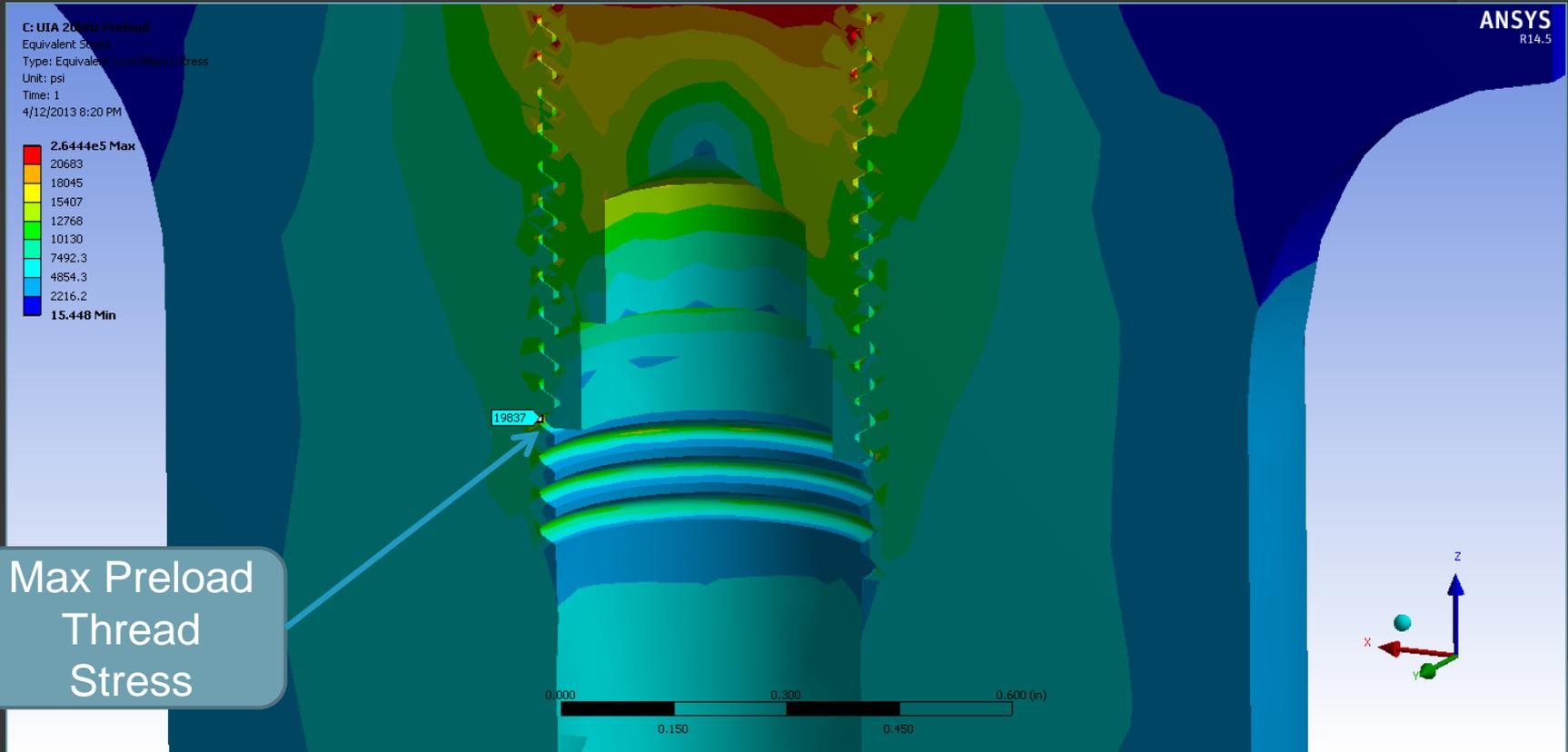
## - Static Analysis, Preload Setup -

- Shut off non-required geometry, apply loads



# Transducer Design and Modeling

- Static Analysis, Preload Results -
  - Plot Stress



# Transducer Design and Modeling

## - Conclusion -

- ANSYS can be setup to be an excellent tool for analysis of ultrasonic transducers
- Direct link to solid model is a great advantage in the management of transducer designs and manufacturing data
- ANSYS workbench in its current form requires coding to achieve true piezo-electric analysis

*Thank you for your time !!*

