Transducer Design and Modeling
42nd 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
Recent advancements in finite element software and computing power have changed the methods to analyze ultrasonic transducers.
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
ANSYS Workbench is set up in an object orientated project format "Drag and Drop"
Each analysis is self contained. Certain analysis elements can be connected (Materials, geometry).
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
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
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 the standard model library.
- Need to input into ANSYS using APDL programming language.
- Use APDL for material definition, and boundary conditions, special piezo results.

Input APDL Code for Modal and Harmonic
Transducer Design and Modeling
- Converter Stack Solid Model, Components -

• One common model for all analysis
• All stack components Modeled
Transducer Design and Modeling
- Converter Stack Solid Model, Modeling Techniques -

• Use solid modeling configuration utility to include or remove analysis details
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)
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

![Modal APDL Commands Piezo Material Input BC’s](Image)
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

Piezo Material Input

Assignment

Modal BC’s Open or Closed
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 permittivity matrix

\[
\begin{align*}
\{T\} &= \{c^E\} \{S\} - \{e\} \{E\} \\
\{D\} &= \{e\}^T \{S\} + \{\varepsilon^S\} \{E\}
\end{align*}
\]

or equivalently

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

where:
- \{T\} = stress vector (referred to as \{\sigma\} elsewhere in this manual)
- \{D\} = electric flux density vector
- \{S\} = strain vector (referred to as \{\varepsilon\} 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
- \([\varepsilon^S]\) = dielectric matrix (evaluated at constant mechanical strain)
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] Compliance Matrix (m^2/N) - Closed Circuit

<table>
<thead>
<tr>
<th></th>
<th>1.14E-11</th>
<th>-4.80E-12</th>
<th>-3.70E-12</th>
<th>0</th>
<th>0</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.14E-11</td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>-4.80E-12</td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>-3.70E-12</td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>3.19E-11</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3.24E-11</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3.19E-11</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### [c] Stiffness Matrix (N/m^2) - Closed Circuit

<table>
<thead>
<tr>
<th></th>
<th>1.484E+11</th>
<th>8.079E+10</th>
<th>8.219E+10</th>
<th>0</th>
<th>0</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.484E+11</td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>8.079E+10</td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>8.219E+10</td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>3.135E+10</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3.086E+10</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3.135E+10</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th></th>
<th>0</th>
<th>0</th>
<th>0</th>
<th>3.80E-10</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>-9.50E-11</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>-9.50E-11</td>
<td>2.50E-10</td>
<td>-9.50E-11</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3.80E-10</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th></th>
<th>0</th>
<th>-9.50E-11</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>2.50E-10</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>-9.50E-11</td>
<td>0</td>
<td>0</td>
<td>3.80E-10</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3.80E-10</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### ANSYS Input

### [c] Stiffness Matrix (N/m^2) - Closed Circuit

<table>
<thead>
<tr>
<th></th>
<th>1.484E+11</th>
<th>8.079E+10</th>
<th>8.219E+10</th>
<th>0</th>
<th>0</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.484E+11</td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>8.079E+10</td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>8.219E+10</td>
<td></td>
<td></td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>3.135E+10</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3.086E+10</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3.135E+10</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### [d] * [c] = Piezoelectric Matrix (C/m^2)

<table>
<thead>
<tr>
<th></th>
<th>-1.71</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>17.05</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>-1.71</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>11.91</td>
</tr>
<tr>
<td>11.91</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

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

Portion of ANSYS APDL Code
Permittivity, Piezo, Compliance Matrix
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
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

\[
\begin{align*}
\text{Cos}/\text{Cop} &= \text{transducer capacitance} \\
\text{Cs}/\text{Cp} &= \text{transducer stiffness} \\
\text{Ls}/\text{Lp} &= \text{transducer mass} \\
\text{Rs}/\text{Rp} &= \text{transducer loss (load)}
\end{align*}
\]
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

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

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

The 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.

\[
F_p = \frac{1}{2\pi} \sqrt{\frac{1}{L_s C_s} + (1 + \frac{C_s}{C_o})}
\]
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
Transducer Design and Modeling
- Modal Analysis, Boundary conditions – series and parallel analysis –
  - ANSYS APDL Code, Apply BC’s for parallel or series modal analysis

Named Component: CERAMIC (bodies)

Named Component: POSITIVE (Top surfaces)

Named Component: NEGATIVE (Bot Surfaces)
Transducer Design and Modeling
- Modal Analysis, Examining results -

Axial Resonance

<table>
<thead>
<tr>
<th>Mode</th>
<th>Frequency [Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>18014</td>
</tr>
<tr>
<td>2</td>
<td>18318</td>
</tr>
<tr>
<td>3</td>
<td>19482</td>
</tr>
<tr>
<td>4</td>
<td>19654</td>
</tr>
<tr>
<td>5</td>
<td>20107</td>
</tr>
<tr>
<td>6</td>
<td>20147</td>
</tr>
<tr>
<td>7</td>
<td>21048</td>
</tr>
<tr>
<td>8</td>
<td>21324</td>
</tr>
<tr>
<td>9</td>
<td>21918</td>
</tr>
</tbody>
</table>
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
Transducer Design and Modeling
- Harmonic Analysis, Use of APDL programming language -

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

- Piezo Material Input (Same as Modal)
- Assignment (Same as Modal)
- Harmonic BC’s
  *Use Driving Voltage*
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
Transducer Design and Modeling
- Harmonic Analysis, Boundary conditions -
  • ANSYS APDL Code, Apply BC’s for harmonic analysis

- Named Component: CERAMIC (bodies)
- Named Component: POSITIVE (Top surfaces)
- Named Component: NEGATIVE (Bot Surfaces)
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

Modify Analysis Settings to sweep frequency
Add APDL Code to Plot Impedance
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

Mode at 20117 Hz is “Visible”
Transducer Design and Modeling
- Harmonic Analysis, Examining Results Frequency Sweep -
  • Graphs of mechanical features

- 20117 Resonance
- Series Axial Resonance
- Parallel Resonance
- Booster Amplitude
Transducer Design and Modeling
- Static Analysis, Preload Setup -
  • Create Static Analysis in Workbench
Transducer Design and Modeling
- Static Analysis, Preload Setup -
  • Shut off non-required geometry, apply loads

20000 lbs Z force
Add Contact
Restrained Surface
Transducer Design and Modeling
- Static Analysis, Preload Results -
  • Plot Stress

Max Preload Thread 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 piezoelectric analysis

Thank you for your time !!