### Concepts:
- MOS transistors are used as linear devices for signal amplification and conditioning
- MOS transistors are used as non-linear devices for power amplification
- Design tradeoffs among gain, bandwidth, output swing, stability, and noise are provided
- Feedback allows another degree of freedom to achieve design goals
- Design requires drawing from model information, making compromises and analyzing results relative to desired specifications
- Noise in transistor-based circuits

### Applications:
- Single stage linear amplifiers
- Multi-stage linear amplifiers
- Class A, Class B, and Class AB amplifiers

### Tools:
- SPICE
- Electronic circuit editor
- Cadence schematic and simulation tools

---

### Linear Amplifier Operation and Design
- Design bias circuits in single and multi-stage amplifiers using active loads for achieving operational specifications
- Analyze and optimize design for achieving fundamental specifications such as gain, bandwidth, and output swing
- Calculate and articulate tradeoffs in amplifier configurations relative to performance
- Show first-order effects and sources of parasitic elements as related to performance of linear amplifiers

### Device Behavior in Circuits
- Determine region of operation, bias points
- Determine equivalent circuits

### Noise and Perturbations on Signal Integrity
- Depict common gate, drain, and source configurations
- Analyze circuits for transfer functions of voltage, current and transconductance

### Waveform Generation and Shaping
- Analyze common topologies for sinusoid, pulse and triangular waveform generation
- Design waveform generators to basic, first order specifications

### Engineering Procedures and Tools
- Display lab notebook that meets industrial needs for documentation and intellectual property instantiation
- Employ SPICE as a routine tool to further understand calculations and measurements
- Extract parameters from measurements to modify model parameters for better matching of simulation to experiment
- Use LabView for data acquisition and analysis and extract parameters using math functions

---

<table>
<thead>
<tr>
<th>IN</th>
<th>OUT</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Analysis and Design Using Models</strong>&lt;br&gt;• Express diode, MOSFET and BJT regions of operation by function and bias</td>
<td><strong>Linear Amplifier Operation and Design</strong>&lt;br&gt;• Design bias circuits in single and multi-stage amplifiers using active loads for achieving operational specifications</td>
</tr>
<tr>
<td><strong>Device Behavior in Circuit Configurations</strong>&lt;br&gt;• Determine region of operation, bias points&lt;br&gt;• Determine equivalent circuits</td>
<td><strong>Device Behavior in Circuits</strong>&lt;br&gt;• Determine region of operation, bias points&lt;br&gt;• Determine equivalent circuits for any region</td>
</tr>
<tr>
<td><strong>Linear Signal Amplification, Transfer Functions</strong>&lt;br&gt;• Depict common gate, drain, &amp; source configs&lt;br&gt;• Analyze configurations for transfer functions of voltage, current and transconductance</td>
<td><strong>Noise and Perturbations on Signal Integrity</strong>&lt;br&gt;• Depict common gate, drain, and source configurations&lt;br&gt;• Analyze circuits for transfer functions of voltage, current and transconductance</td>
</tr>
<tr>
<td><strong>Thevenin and Norton Equivalent Circuits</strong>&lt;br&gt;• Transform sources and impedances to equivalent forms to analyze circuit behavior</td>
<td><strong>Waveform Generation and Shaping</strong>&lt;br&gt;• Analyze common topologies for sinusoid, pulse and triangular waveform generation&lt;br&gt;• Design waveform generators to basic, first order specifications</td>
</tr>
<tr>
<td><strong>SPICE Simulation</strong>&lt;br&gt;• Simulate circuits&lt;br&gt;• Use simulation to confirm hand calculations for single-transistor amplifiers</td>
<td><strong>Engineering Procedures and Tools</strong>&lt;br&gt;• Display lab notebook that meets industrial needs for documentation and intellectual property instantiation&lt;br&gt;• Employ SPICE as a routine tool to further understand calculations and measurements&lt;br&gt;• Extract parameters from measurements to modify model parameters for better matching of simulation to experiment&lt;br&gt;• Use LabView for data acquisition and analysis and extract parameters using math functions</td>
</tr>
<tr>
<td><strong>Laboratory Procedures</strong>&lt;br&gt;• Connect devices and evaluate bias circuits and time-varying behavior&lt;br&gt;• Analyze measurements and display results in Bode plots for transfer functions&lt;br&gt;• Extract device properties (e.g. threshold voltage) from measured data&lt;br&gt;• Use LabView to derive I-V characteristics of devices and customize V1’s</td>
<td><strong>Pre-requisites</strong>&lt;br&gt;• ECE 331 with a C or higher</td>
</tr>
</tbody>
</table>

**Pre-requisites**
- ECE 331 with a C or higher