CoventorWare Tutorial

Presented by Brian Pepin

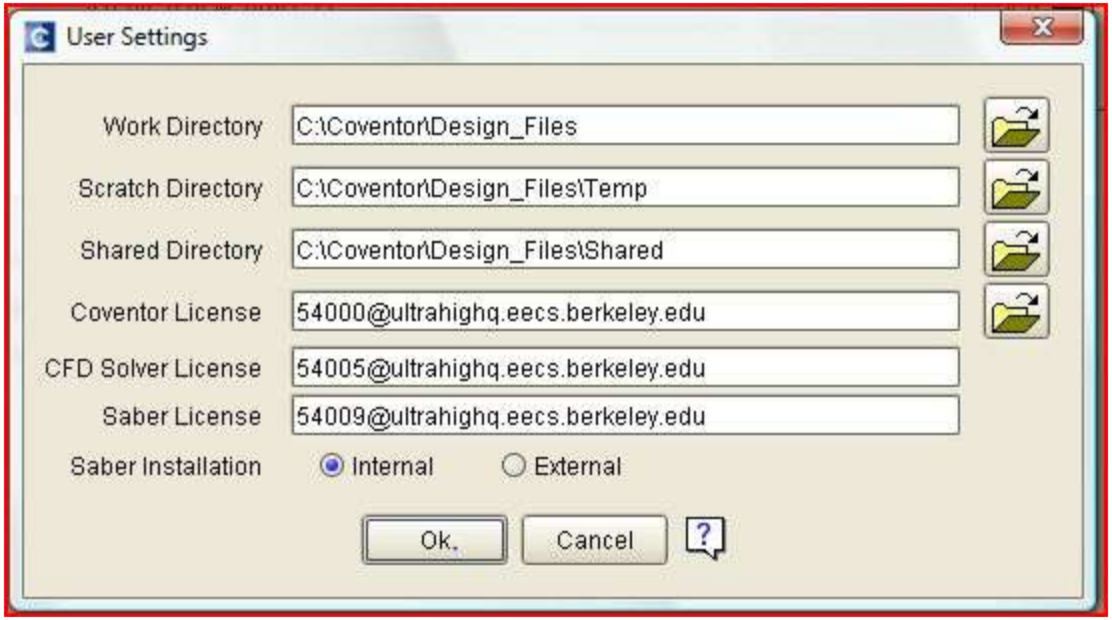
11/07/2011

Downloading Software

- **CoventorWare™ 2010 Full Release available at** http://www.coventor.com/mems/download.html
  - Click on link to download CoventorWare 2010 for Windows XP Pro and Windows 7 (873 MB)
  - Username: ucberk3
  - Password: sensor
  - Release for Red Hat Linux also available
  - Do NOT download SABER component library
- After downloading, click on .exe file to begin installation (just follow along with the menus)
  - May take a very long time (>30 minutes) – be PATIENT!
  - When the program asks for license file at the end of the installation, just press “cancel”
  - Restart computer when finished (even though not prompted)

Licensing

- After opening CoventorWare for the first time, you will be prompted to enter license info
  - Change all licenses to @ultrahighq.eecs.berkeley.edu as shown below

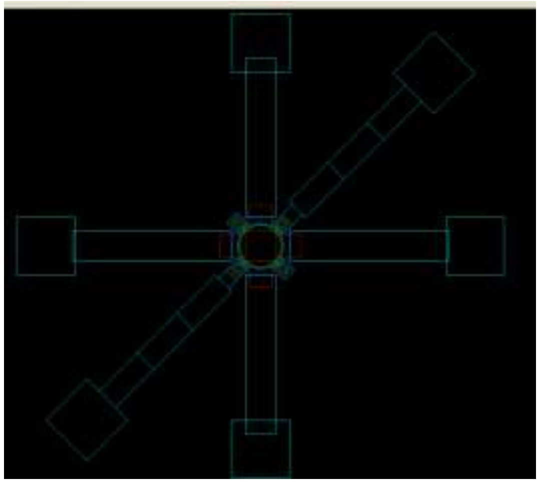


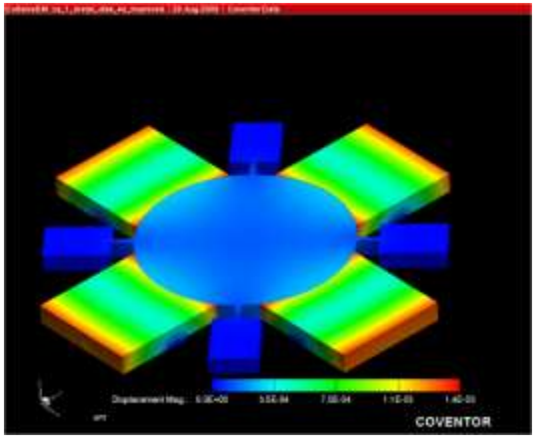
Tutorial

- To learn most useful aspects of CoventorWare, only need to complete first two tutorial sections (2-4 hours)
  – memsdesign_vol1.pdf; Section 1 (Introduction) and Section 2: Beam Design
  – Other sections of interest – Section 3 (extraction of lumped elements from FEM models); Section 9 (Resonator Analysis); Section 9 (Thermal Analysis)

Overview

- Motivation – Why Coventor?
- Coventor Simulation Flow
- Designer
  - Material Properties Editor
  - Process Editor
  - Layout Editor
- Analyzer
  - Meshing
  - MemElectro
  - MemMech
  - CoSolve
  - Parametric Simulations
  - Visualization
- Advanced Solvers and Reduced Order Modeling
- Starting to use Coventor





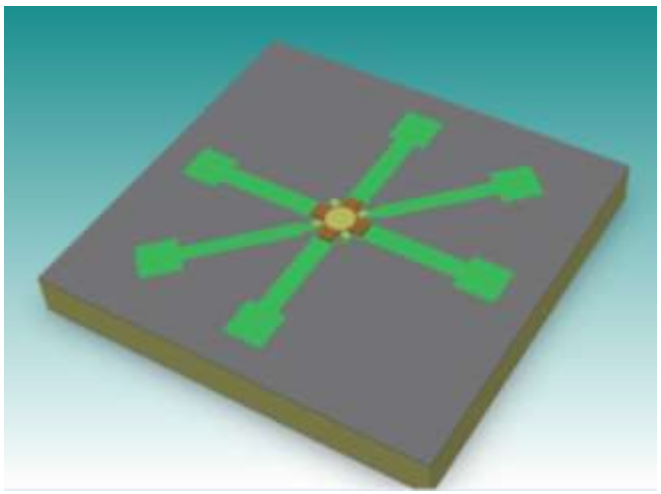
Motivation – Why Coventor?

- Intuitive for MEMS engineer
  - 3D model construction from GDSII files
  - Streamlined operation for common MEMS simulations, including:
    - Coupled electromechanical (ex. Pull-in, electrostatic spring softening)
    - Coupled thermomechanical (ex. Thermoelastic damping, thermal stability)
    - Piezoelectric effects (ex. Stress stiffening)
- In addition to modal and harmonic analyses, capable of transient simulation with nonlinear boundary conditions (i.e. contact)
- Straightforward addition of damping to any simulation (stokes or squeeze film air damping, anchor loss, themoelastic, etc.)
- Powerful visualization tools
- Coupling with ODE models

COVENTOR
WHAT'S NEXT. AND NEXT. AND NEXT.

Coventor Simulation Flow

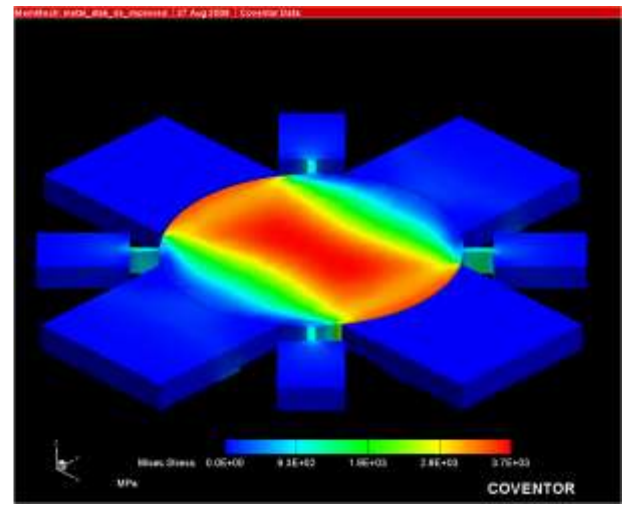
Designer



Inputs: - Materials Data
- Process Data
- Layout (GDSII or Manual)

Output: - 3D Model

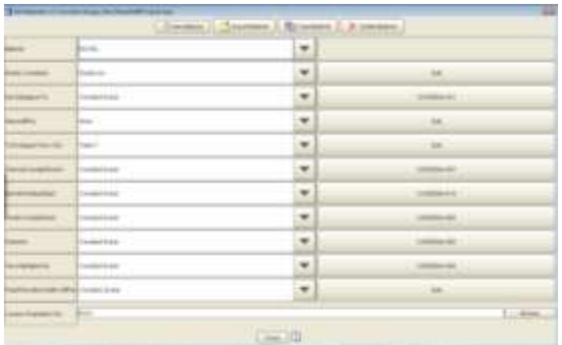
Analyzer

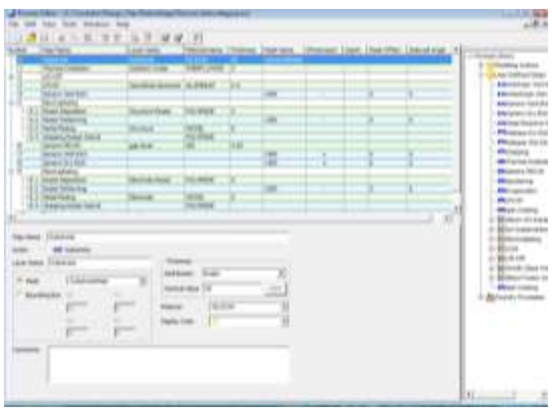


Inputs: - 3D Model from Designer
- Meshing Info
- Analysis Definition

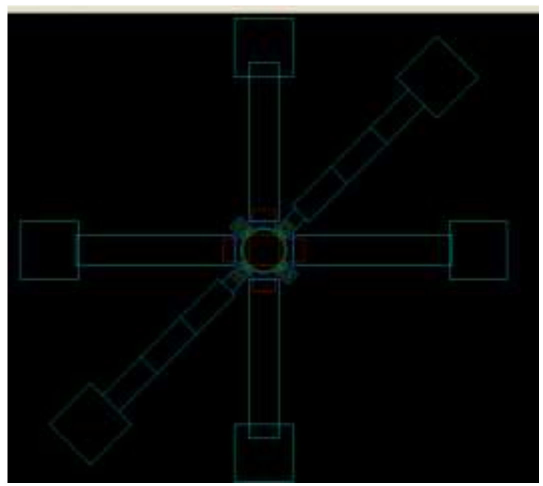
Output: - Simulation Results

Designer - Overview

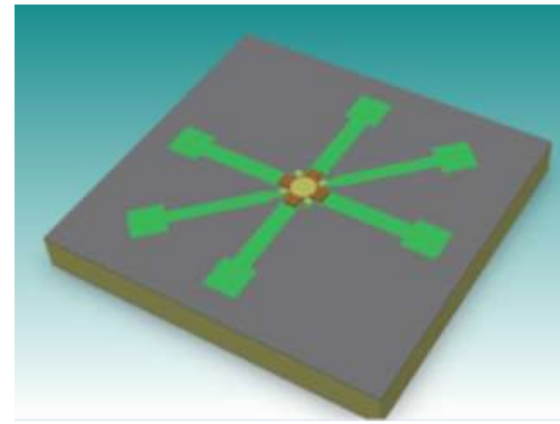


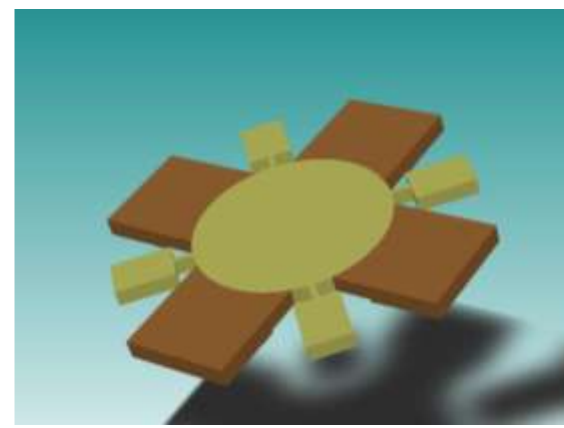


Materials & Process



GDSII Layout Files





3D Model

Designer - Materials

Database of >40 commonly used MEMS materials, or can define custom materials

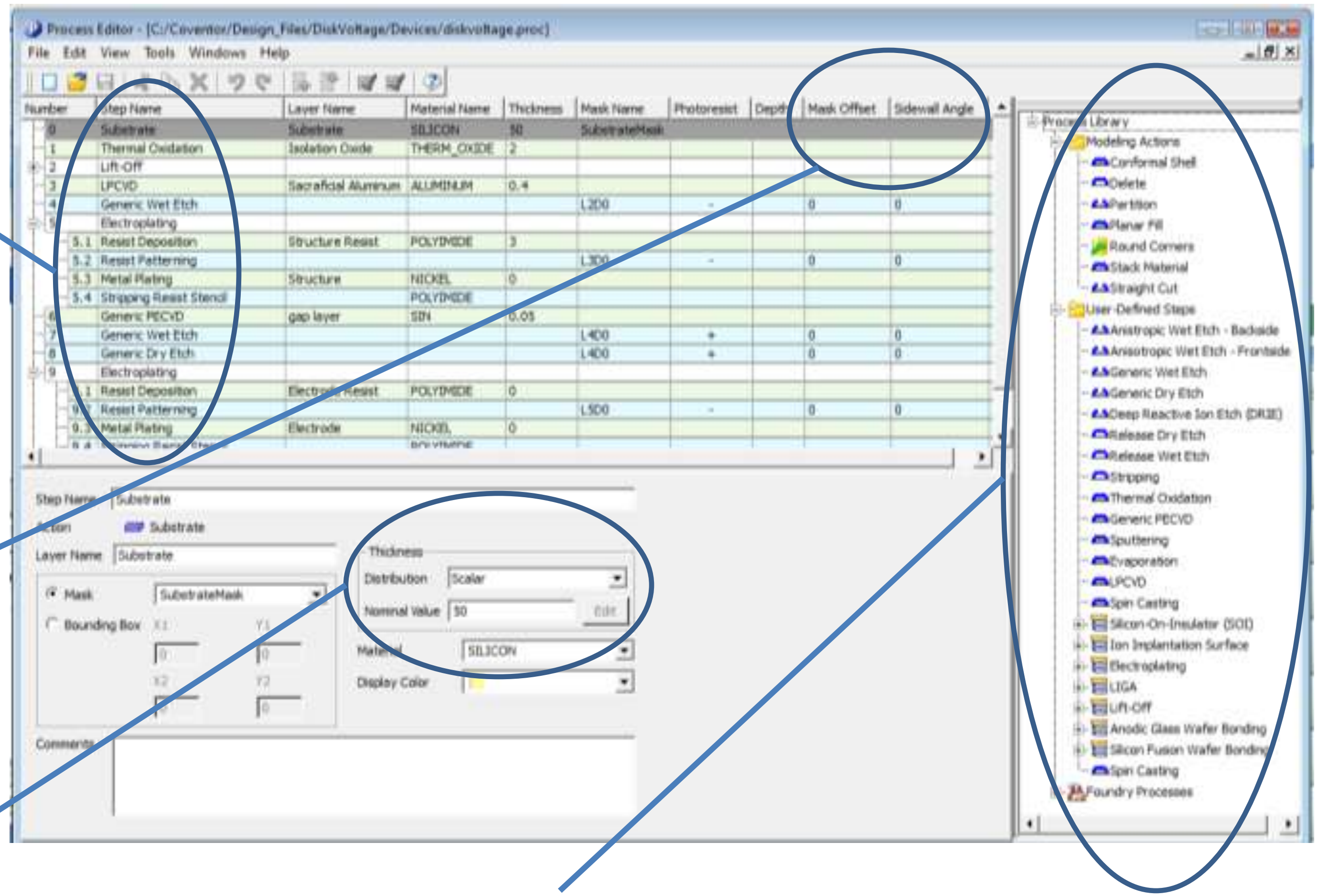
- Young's Modulus and Poisson Ratio
- Residual stress distribution and magnitude
- Electrical conductivity (constant, exponential, look-up table, diffusion limited)
- Piezoresistive properties

Designer – Process Editor

Model is defined using actual process steps, coupled with masking layers

Can define non-idealities such as sidewall angles

Supports different types of material depositions (i.e. conformal shell, planar fill and stack material)



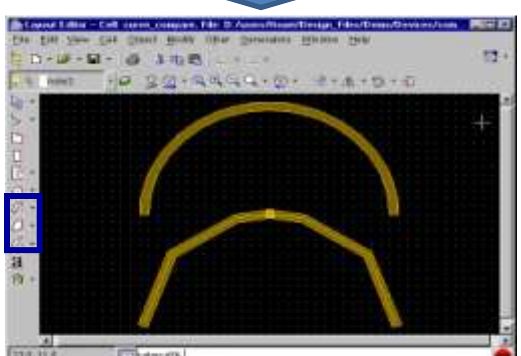
Includes idealized modeling actions as well as common MEMS process steps and sequences

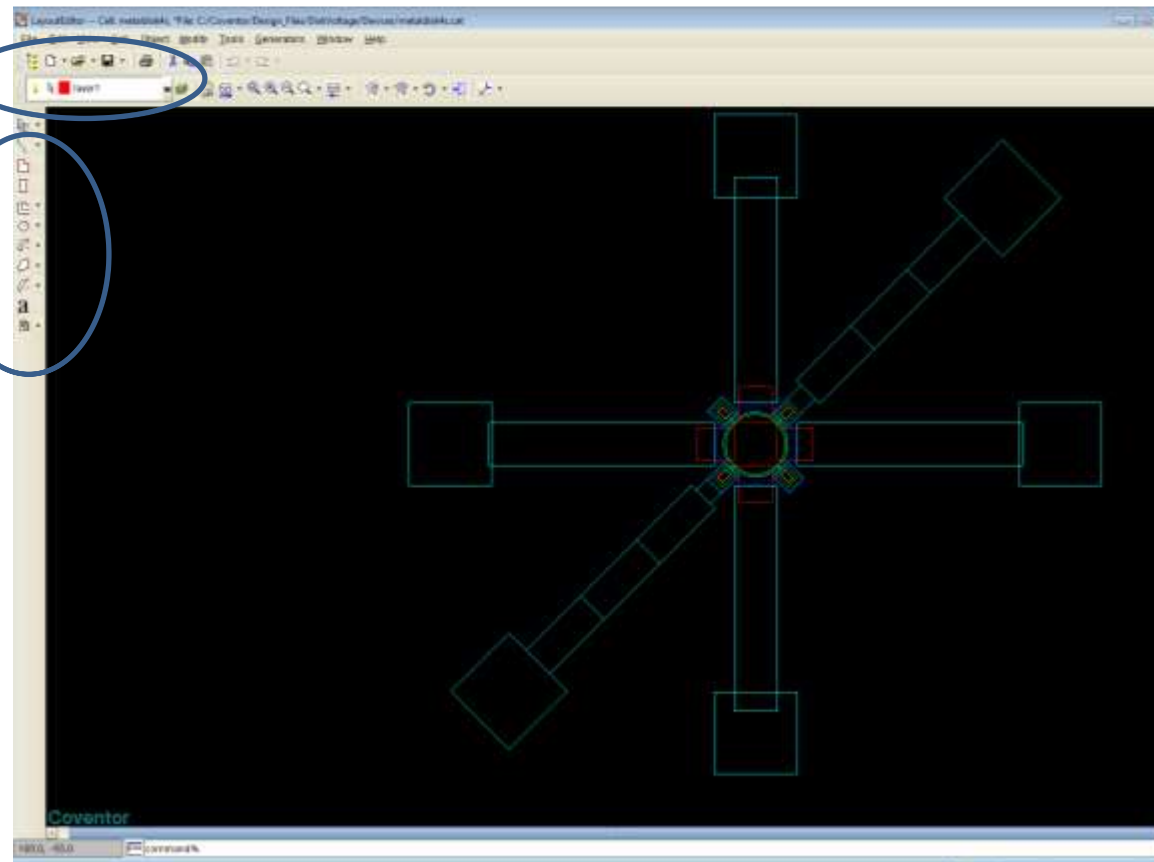
Designer – Layout Editor

Can import GDSII layout files or draw layout in Coventor using drawing toolset

Layer names transfer directly from Cadence-generated GDSII files

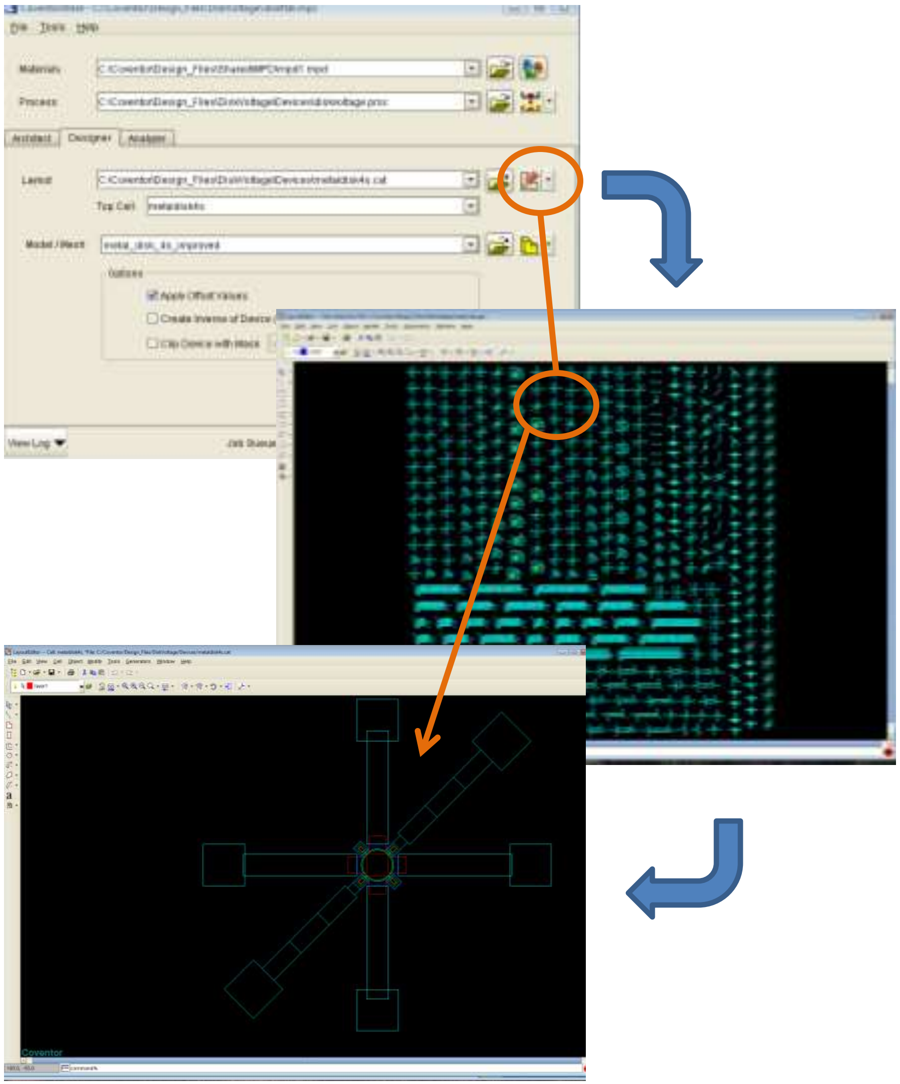
Drawing toolset supports drawing of true curves – better for meshing





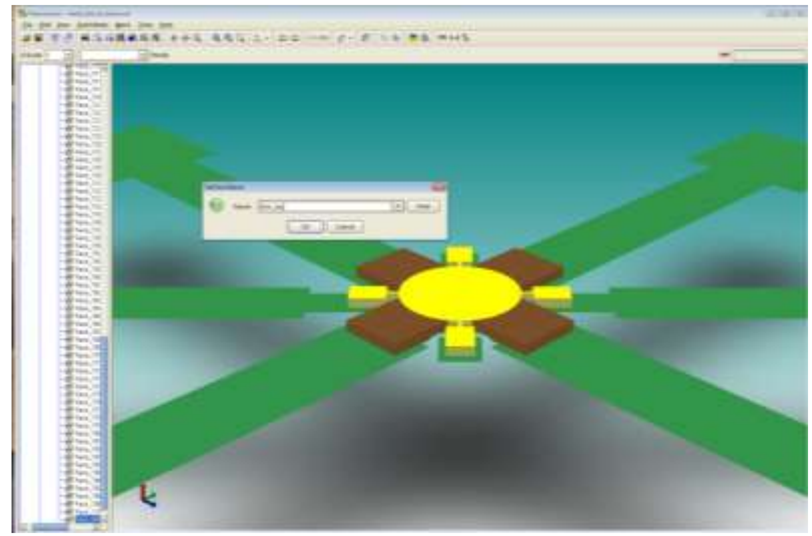
Importing GDSII Files

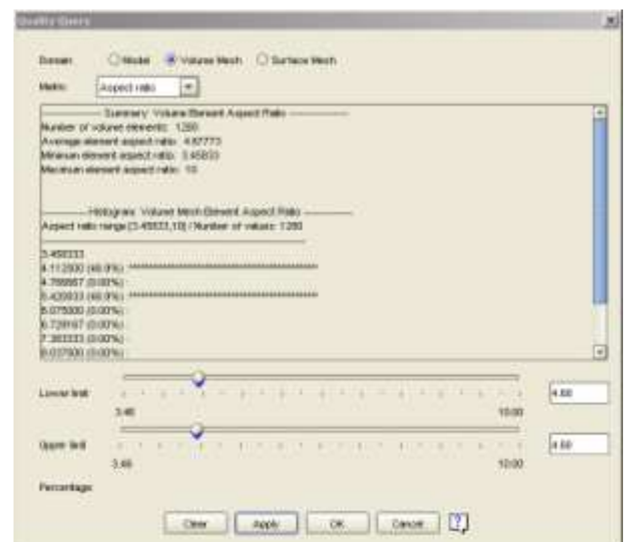
- 1. Open a new layout in Layout Editor
- 2. From the file menu click “open”, making sure that you have GDSII selected as the file type. Open a GDSII file for single die (any larger will crash the computer)
- 3. When prompted to select a “template file”, press cancel
- 2. Select the structure of interest using the “Select Area” feature from the Edit menu
- 3. Select “Copy” from the Edit menu, and open a new layout in the Layout Editor. This will automatically be a .cat file
- 4. Select “Paste” from the Edit Menu. Your “Top Cell” name should be the same as your .cat file name



Designer - Preprocessor

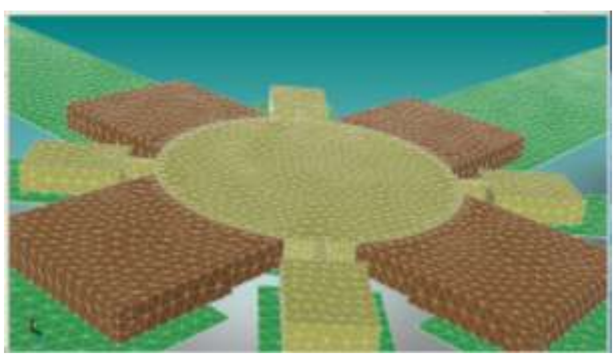
- Can select and name faces, volumes, and conductors
  - Several faces can be selected at once to define a continuous surface
  - Give as many features as possible unique, descriptive names – only named features can be BC's!
- Quality Query of Solid Model
  - Check for sliver faces and small edges
  - "Heal" function can repair some problems
  - Try using "planar fill" instead of "conformal shell" to eliminate faces with very high aspect ratios (>100)
  - Best practice is to build up your model **one process step at a time**, checking in the Preprocessor for abnormalities after each new step is added
- **Once the process is finalized, any structure on the same die can be easily modeled!**



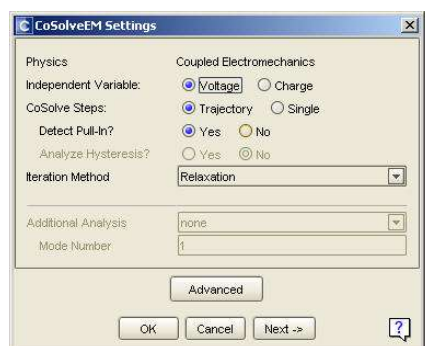


Analyzer - Overview

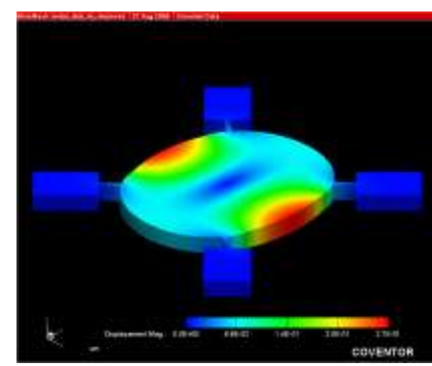
Electro-Mechanical Bundle	**MemElectro –** Electrostatic and electroquasistatic solvers **MemMech** - mechanical, piezoelectric and thrermomechanical solvers, including structural, modal, harmonic, and transient analyses (electromechanical and thermal) **CoSolveEM** – Coupled electro-mechanical solver All solvers include options for parametric studies
MemPZR	Computes resistance field, equilibrium potential, and current density fields of resistors under mechanical stress.
MemHenry	Solves for frequency dependent resistance and inductance



Meshing



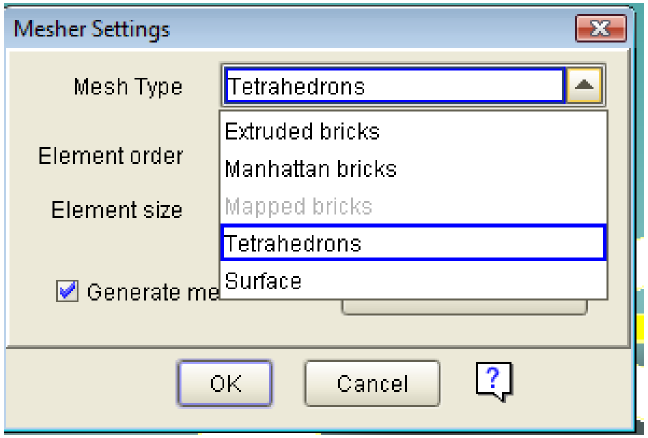
Settings and BCs



Visualization

Analyzer - meshing

- Continuum Elements – Linear or Parabolic
  - 3D hexahedrons (Manhattan or Extruded Bricks)
  - 3D tetrahedrons
- Surface elements : 2D triangles and quadrilaterals
- Mesh type depends on analysis type
  - Parabolic most accurate, but **cannot be used for contact**
  - Linear can be computationally efficient, but has limitations
- Use hexehedral (brick shaped) whenever possible for best convergence
- Tetrahedral accurate for small displacements, and can mesh complicated geometries (rounded corners, sidewalls, etc.)
  - Should use parabolic
  - Must have very fine mesh



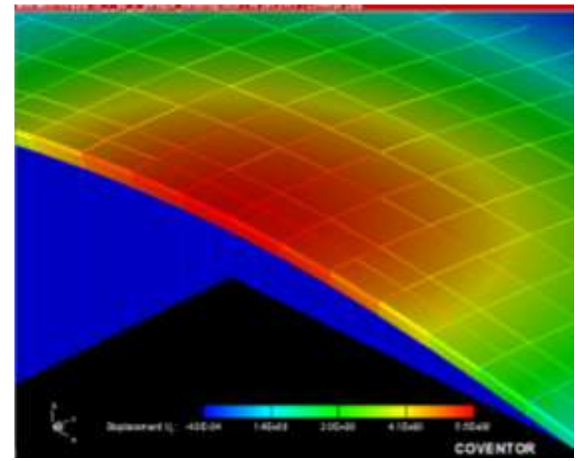
Mesh optimization

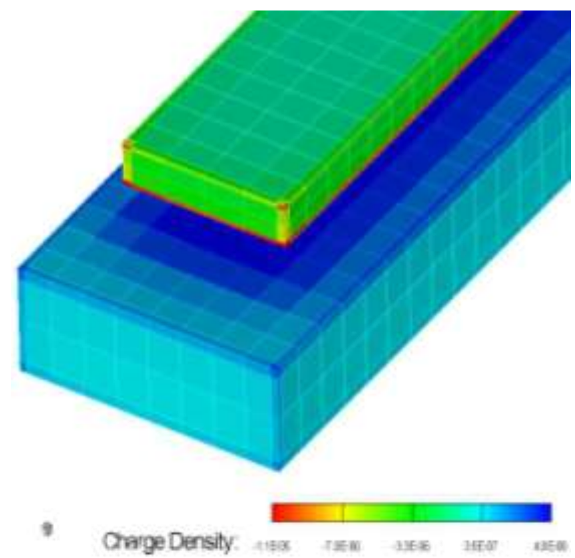
- Linear : Structural / Thermal
- Parabolic : All solvers
  - 10-node tetrahedral: Accurate for thermal and structural small displacement in arbitrary geometries
  - 27-node brick: Used for all other parabolic element representations
  - 20-node brick: Used when PZE materials are included in model
- Surface : Electrostatics and Damping

Solver	Not Suitable	Suitable	Best Practice
MemElectro	None	Surface, Tetrahedral	Hexahedral
MemMech	Surface	Tetrahedral	Hexahedral
CoSolve	Surface (for moving parts)	Surface for non-moving parts	Hexahedral
MemPZR	Surface, tetrahedral for electrical parts	Linear hexahedral for electrical parts	Parabolic Hexahedral
MemHenry	Surface, Tetrahedral	Hexahedral	Hexahedral
DampingMM, Reynolds	Surface, Tetrahedral	Hexahedral	Hexahedral
DampingMM, Stokes	None	Any	Any

Mesh Optimization

- Meshing for mechanical simulations
  - Must be refined enough to capture stress gradients
  - Aspect ratio for nonlinear problems should be <30
- Meshing for thermal simulations
  - Must be refined enough to capture thermal gradients
  - High aspect ratios are feasible
- Meshing for electrostatic simulations
  - Uses BEM so surface meshes are feasible for certain geometries
  - Turbo option will automatically refine mesh at edges as necessary



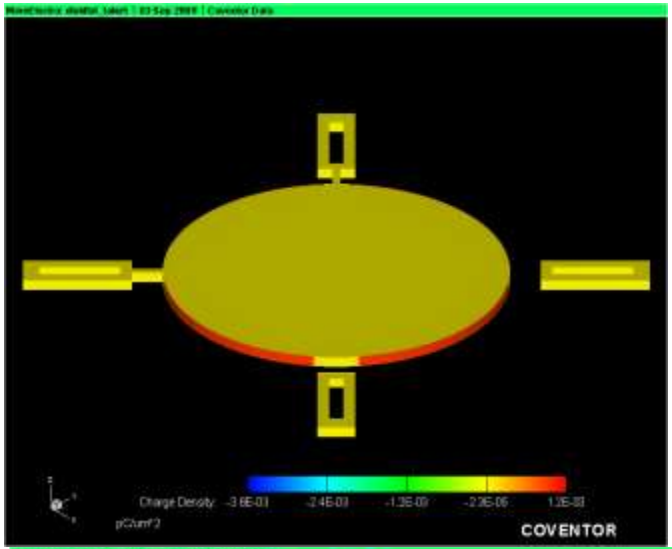


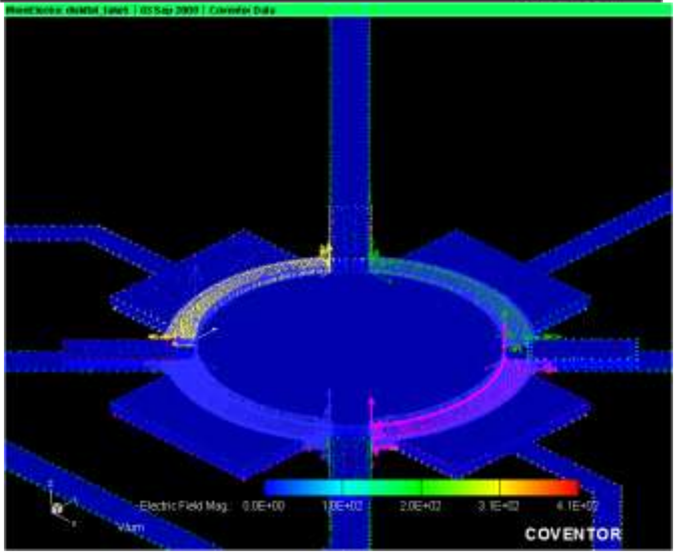
Mesh Optimization

- When mesh is complete, check Quality Query for the following metrics:
  - Aspect Ratio (maximum and average)
  - Corner Angles
  - Edge Length
- Check .msg file in Coventor/Design_Files/Temp directory to examine warnings or errors in mesh generation
- To optimize performance, will likely have to redefine mesh through iterative process for each solver. For MemElectro:
  - After simulation, check Near Field Size: “Direct multiplications=“; should be in the 0.5 to 3% range. CPU time and memory use will be poor outside this range.
  - Optimum mesh ~5kb per panel. Lower quality meshes take more memory. Optimum mesh with more panels solve faster than a poor mesh with fewer panels
  - For more meshing information, check the Appendix at the end of the presentation

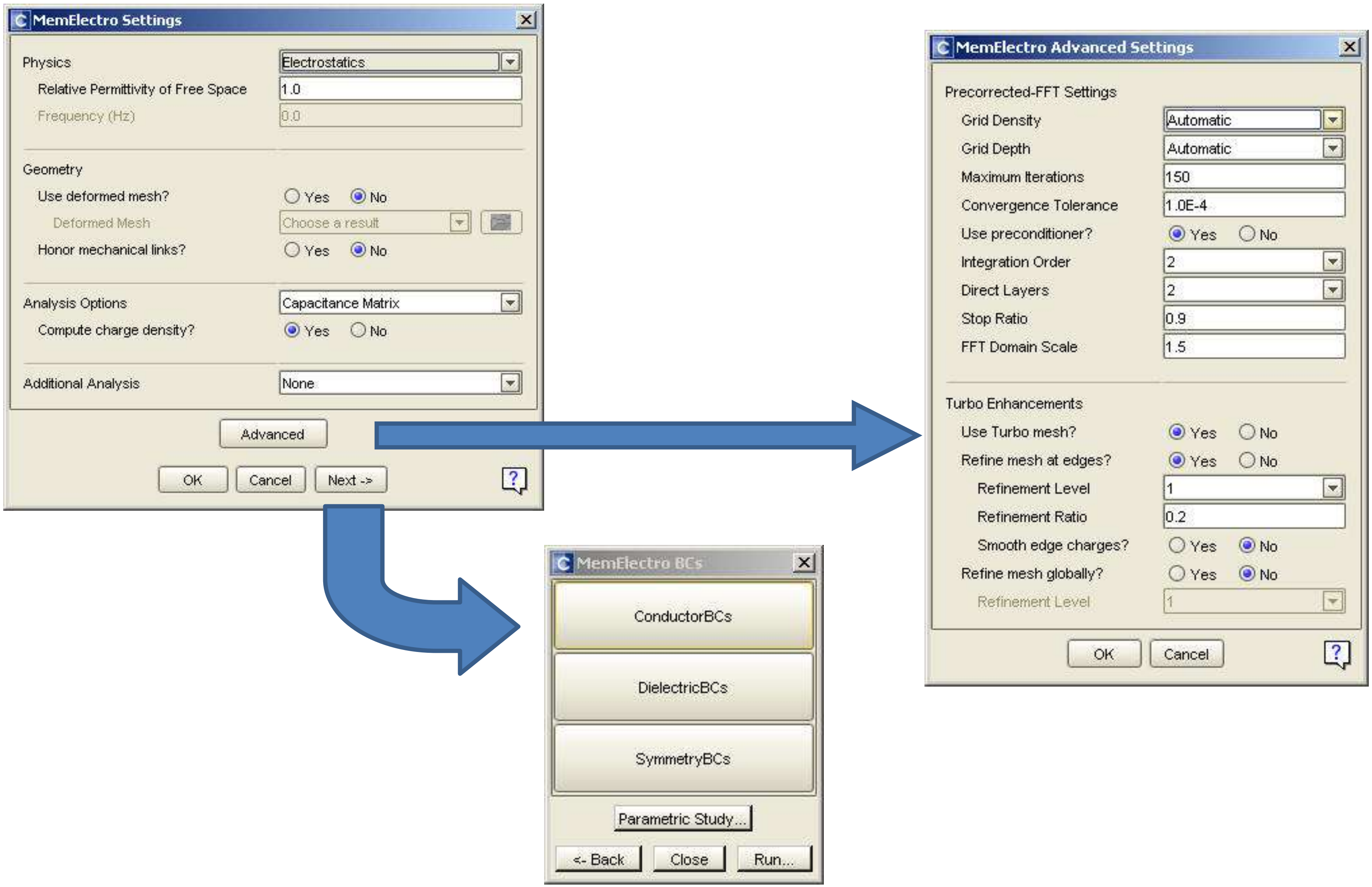
Analyzer - MemElectro

- MemElectro solves electrostatics and quasi-statics
  - Capacitance Matrix
  - Conductance (lossy media)
  - Forces on conductors and dielectrics
- Precorrected-FFT Accelerated: near linear cost with panel number
- BEM solver for capacitance matrix – requires only a mesh of the part surface
- Heuristic mesh refinement at edges



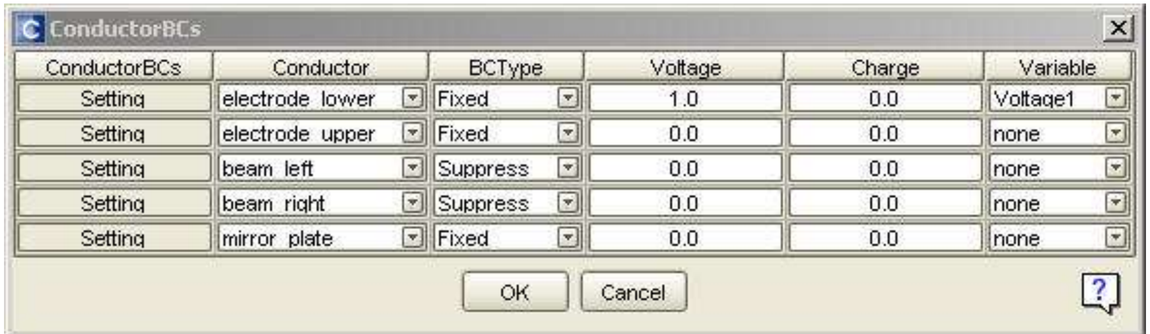


MemElectro User Interface



MemElectro Boundary Conditions

Conductor BC's



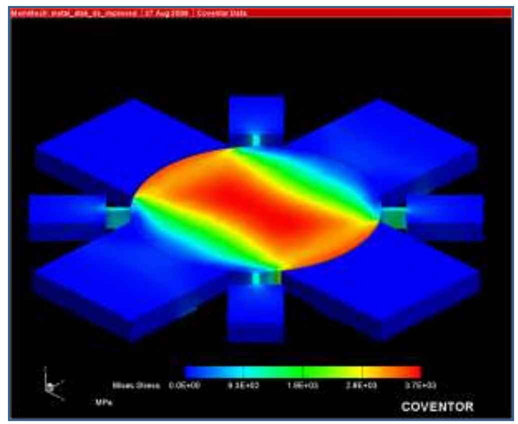
- **Fixed**: Use when conductor potential or charge is known
- **Suppress**: Use to remove a conductor from capacitive simulations
- **Float**: Use when potential is unknown

Dielectric BC's

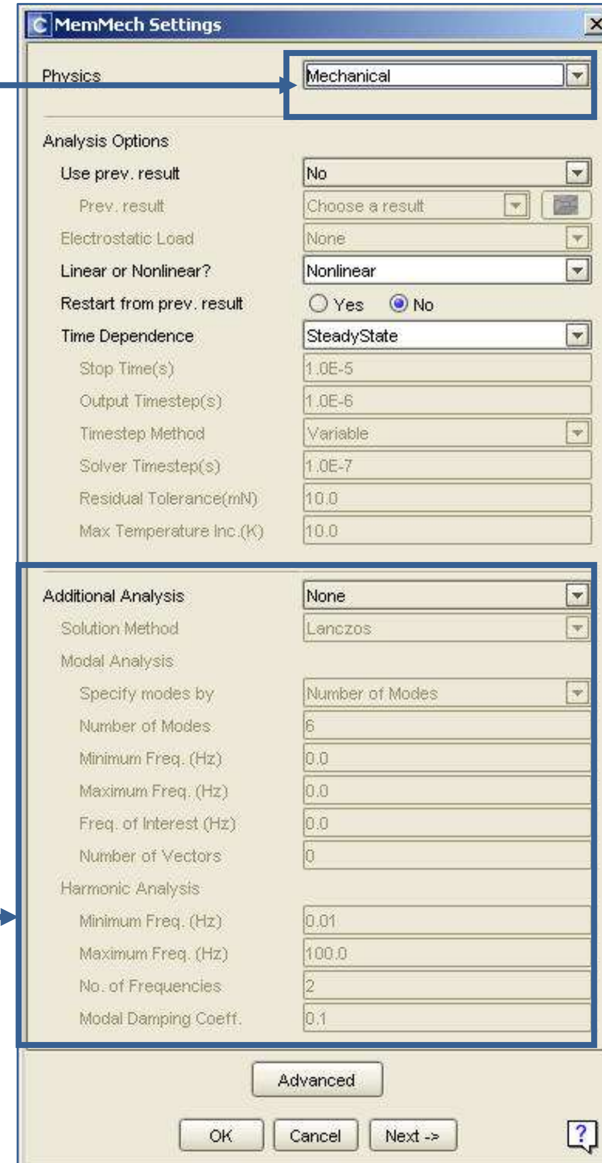
-Can suppress if not of interest in particular simulation
-Adding dielectrics causes more panels to be created for simulation, dramatically increasing simulation time

Analyzer - MemMech

- Can perform a variety of multi-physics analyses in steady state or transient
  - Mechanical/Thermomechanical/Electrothermomechanical
    - Mechanical deformation
    - Stress and Rxn forces
  - Thermal
  - Piezoelectric

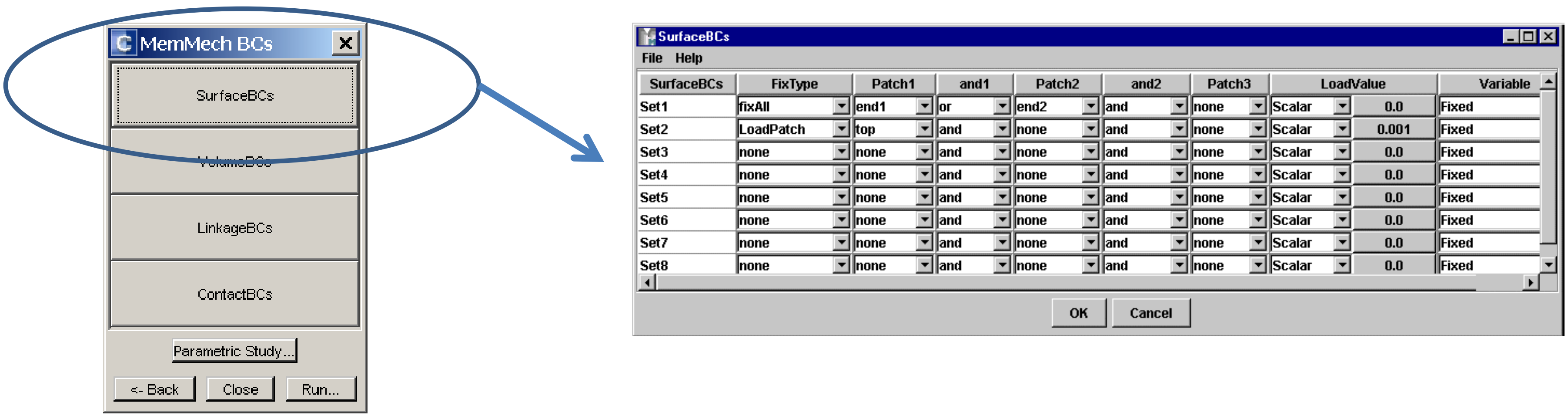


- Modal analysis settings – no stress/strain calculations



MemMech BC’s

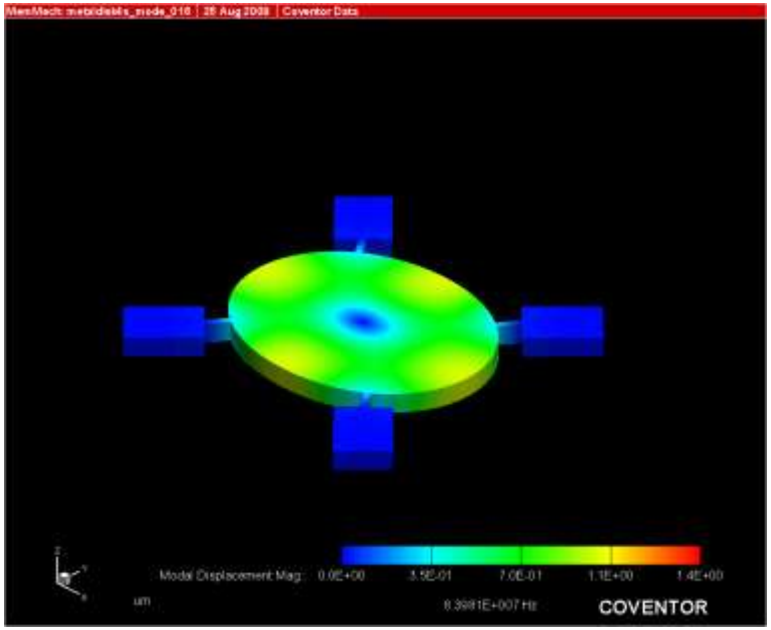
- **SurfaceBCs:** surface boundary conditions, e.g fixing and loading.
- **Volume BCs:** Temperature, stress gradient or acceleration.
- **LinkageBCs:** Couple or link different meshes
- **ContactBCs:** Used to set contact conditions between one or more parts



- **FixType:** Load condition on surface (fixed, load patch, temperature, etc)
- **Patch1, 2, 3:** Name of surface (from solid model)
- **And1, 2:** Boolean and/or operations connecting load surface. “Or: is additive!

Modal Analyses

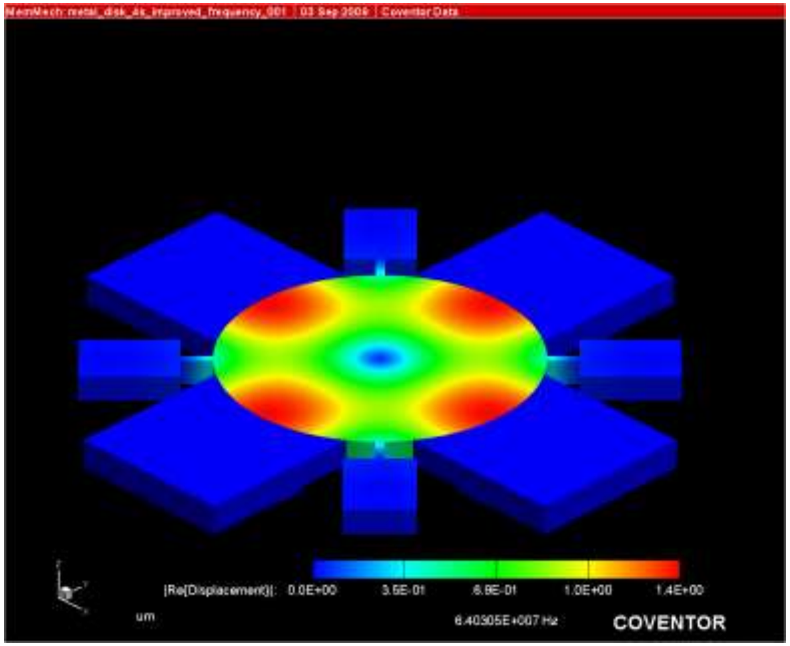
- Can perform modal analyses similar to Comsol/Ansys, but potentiallyy more accurate due to model construction directly from GDSII files
  - No stress/strain calculated
  - Have to make sure and apply the correct boundary conditions!
- Can consider effects of:
  - Electrostatic spring softening
  - Residual Stress
- Make sure to suppress unnecessary structures for modal simulation (and other mechanical simulations)



Harmonic Analyses

- Can perform direct simulation of excitation by a harmonic waveform (pressure, temperature, voltage, etc)
  - Good to perform a modal analysis first to determine correct frequency range
  - Can select arbitrary number of frequencies to sweep through

- Considers some damping effects
  - Modal damping (function of critical damping of system particular to each eigenmode)
  - Thermoelastic damping



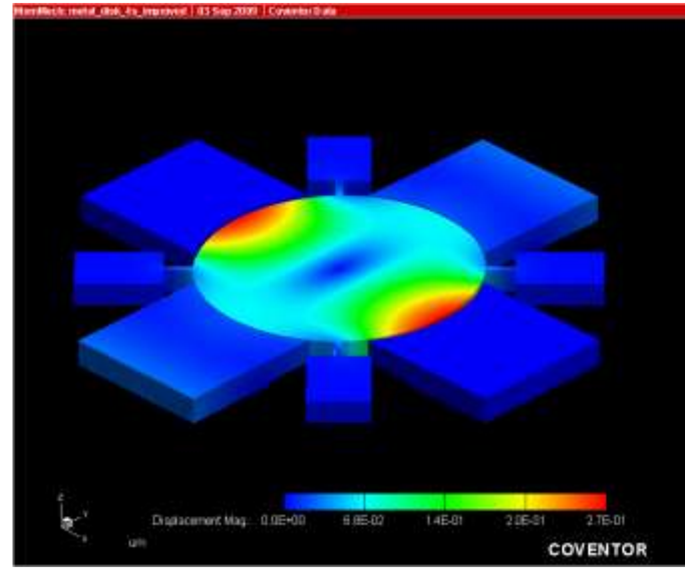
Contact boundary conditions

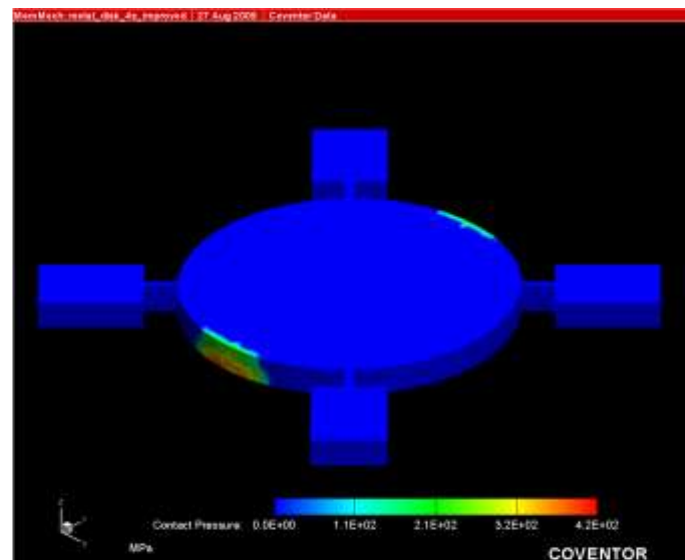
- Contact BCs defined between two continuous surfaces
  - "Surface" Master should be more rigid
  - "toPatch" slave can bend, stress, etc
- Can only use linear meshes for master and slave surfaces
- Master surface can "expand" to improve stability
- Can simulate complex interactions
  - Friction
  - Soft contact (both deforming)

ContactBCs

ContactBCs	Surface	Offset		Expand	toPatch	Friction	Hcrit	SoftCS	SoftPO
Set1	output_elec_one	Vector	Edit	0.1	output_disk_one	0.0	0.0	0.0	0.0
Set2	output_elec_two	Vector	Edit	0.1	output_disk_two	0.0	0.0	0.0	0.0
Set3	input_elec_one	Vector	Edit	0.1	input_disk_one	0.0	0.0	0.0	0.0
Set4	input_elec_two	Vector	Edit	0.1	input_disk_two	0.0	0.0	0.0	0.0

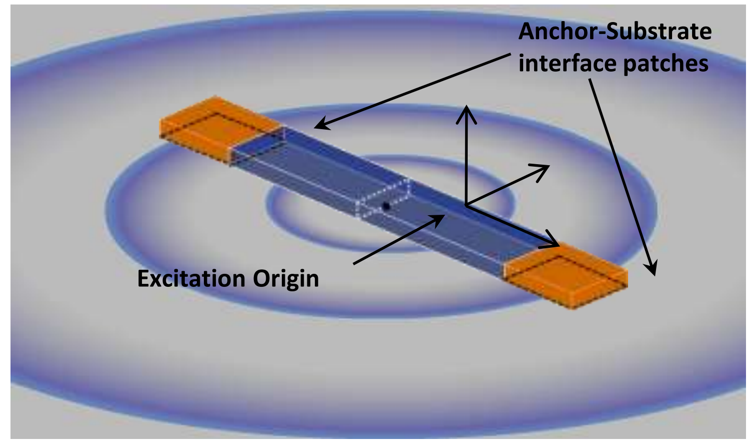
OK Cancel

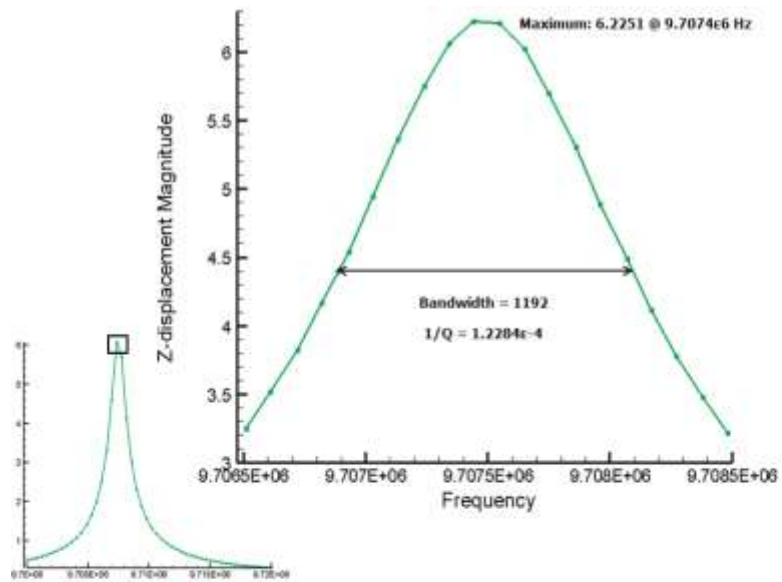




Damping

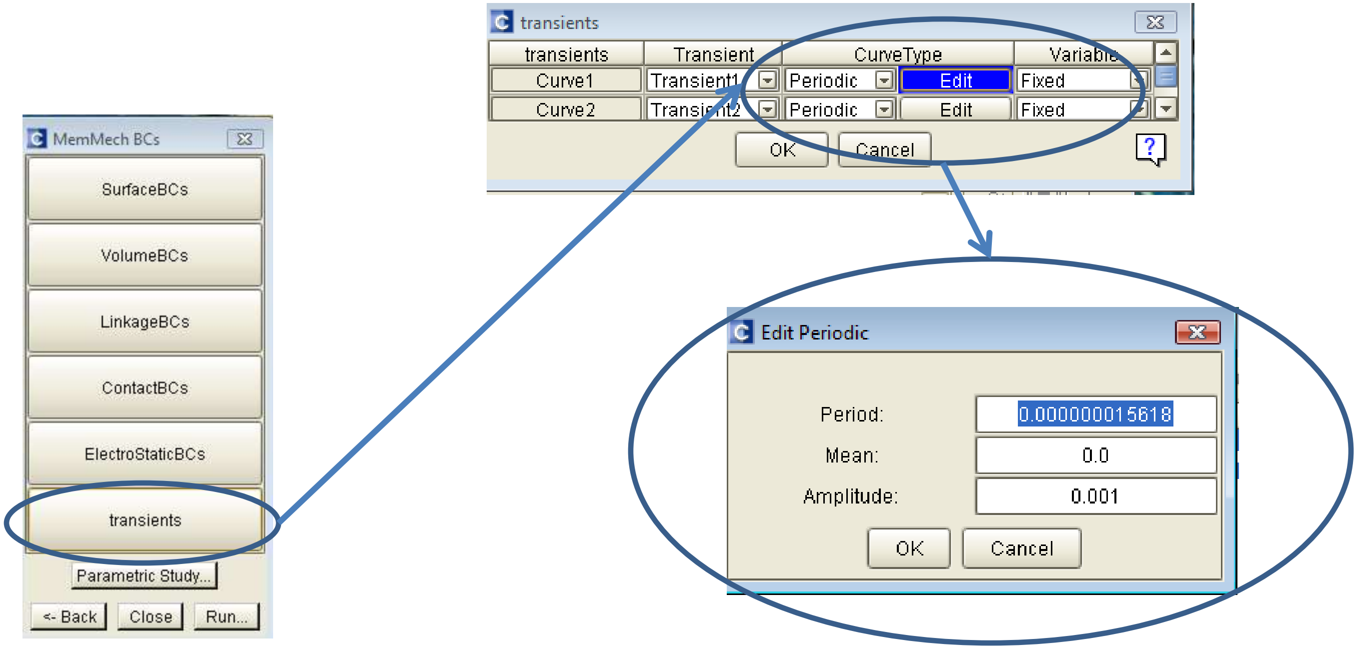
- Coventorware can account for several common types of damping, including anchor loss, TED, material losses, and stokes damping
- Ex. Anchor Loss
  - Uses fictitious infinite substrate to transmit energy (QuietBoundary)
  - Outputs Q factors for different frequencies
  - Working directly with Coventor application engineers on implementing for Nguyen group resonators



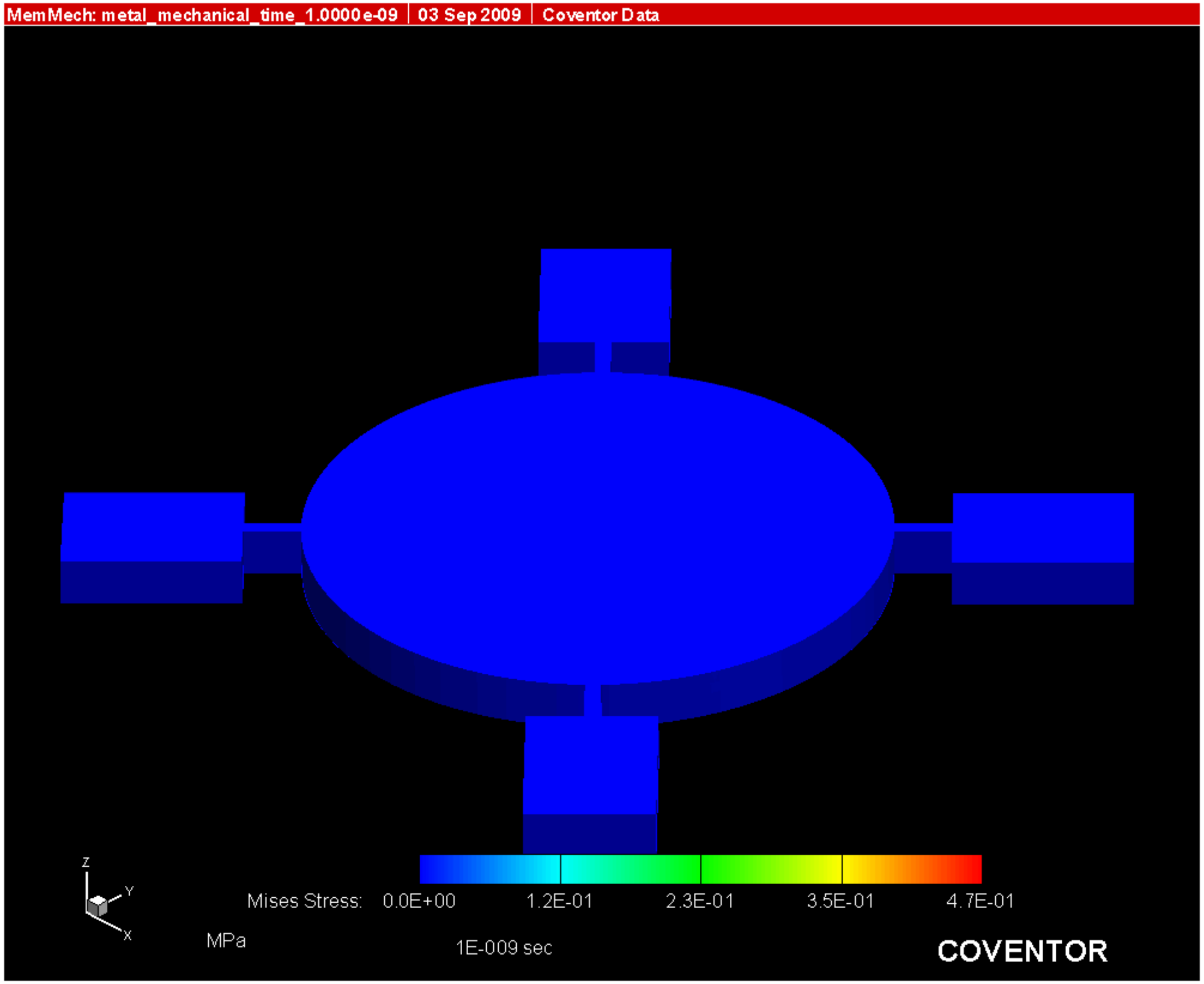


Transient Analysis

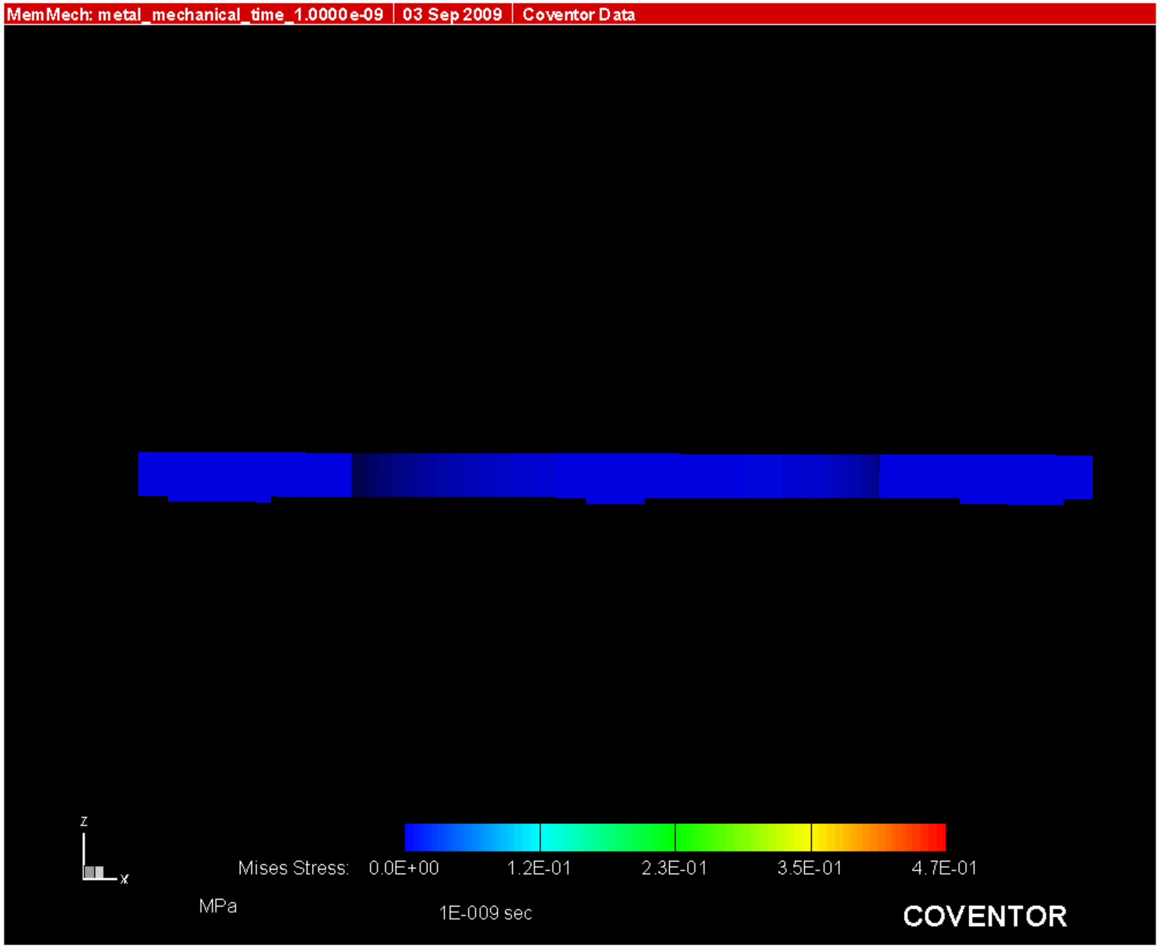
- Allows two load conditions (thermal, electrical, or mechanical) to be multiplied by transient waveforms
- Makes it possible to simulate stress/strain, displacement, forces, etc at resonance, as opposed to statically
- Allows for nonlinearities such as contact boundaries and anchor losses
- EXTREMELEY computationally intensive – don’t use to debug



Transient Results

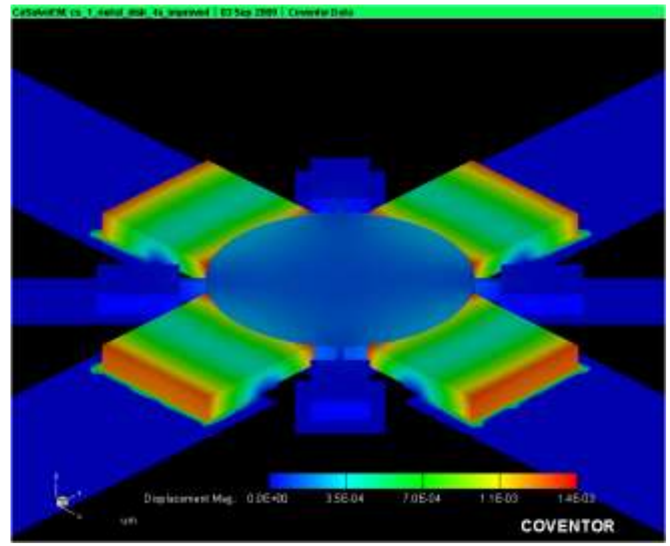


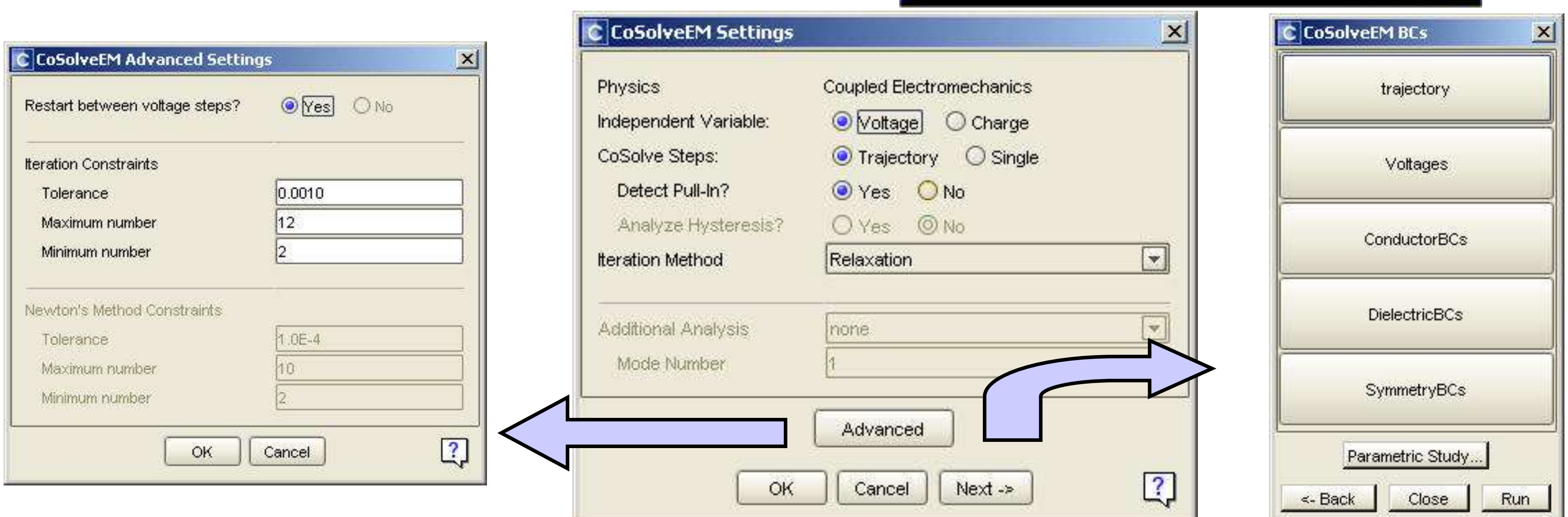
Transient Results



Analyzer - CoSolve

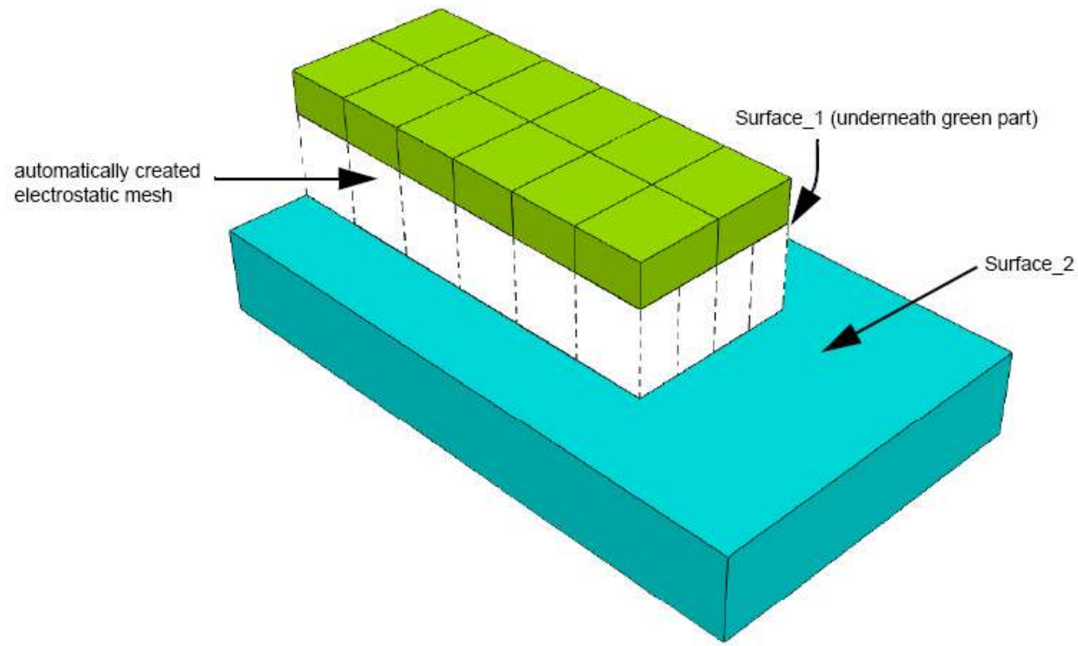
- CoSolve is a coupled Electrostatic-Mechanical solver
  - Combination the MemElectro and MemMech
  - Can solve iteratively or simultaneously
- First define mechanical BCs in MemMech, then use CoSolve menu to define electrical BCs





Coupled Electrostatic (MemMech)

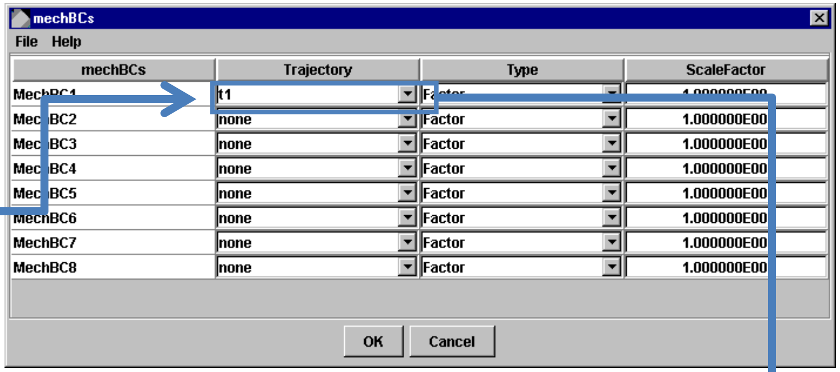
- If a transient CoSolve-type simulation is desired, a coupled electro-mechanical simulation can be conducted inside MemMech
  - Applies potential difference between two surfaces
  - Uses Abaqus solver (FEM) as opposed to BEM
  - Needs to mesh the gap
  - Partition the model to avoid fringing fields
  - In general, cannot use if fringing fields are important

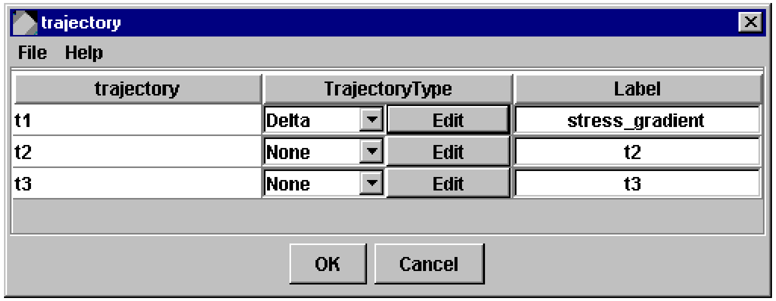


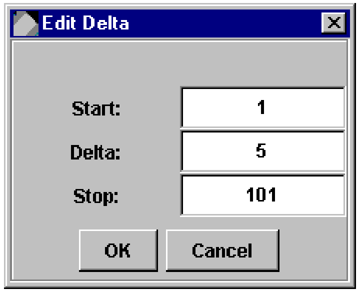
Parameterized Studies

- View trends or sensitivity to a given parameter
- Sets variations automatically and iterates using appropriate solver.

Use the Mech BCs window to apply the trajectory

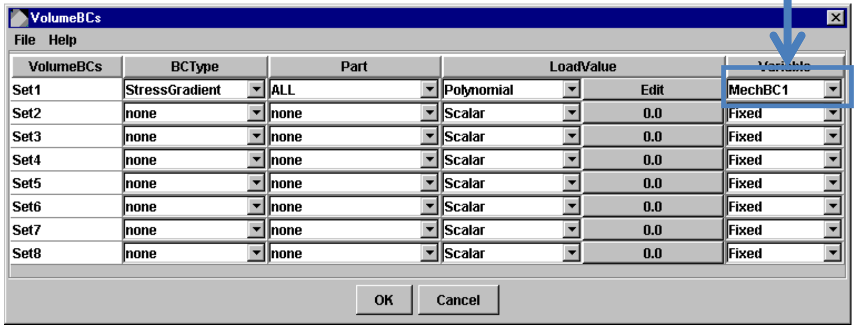






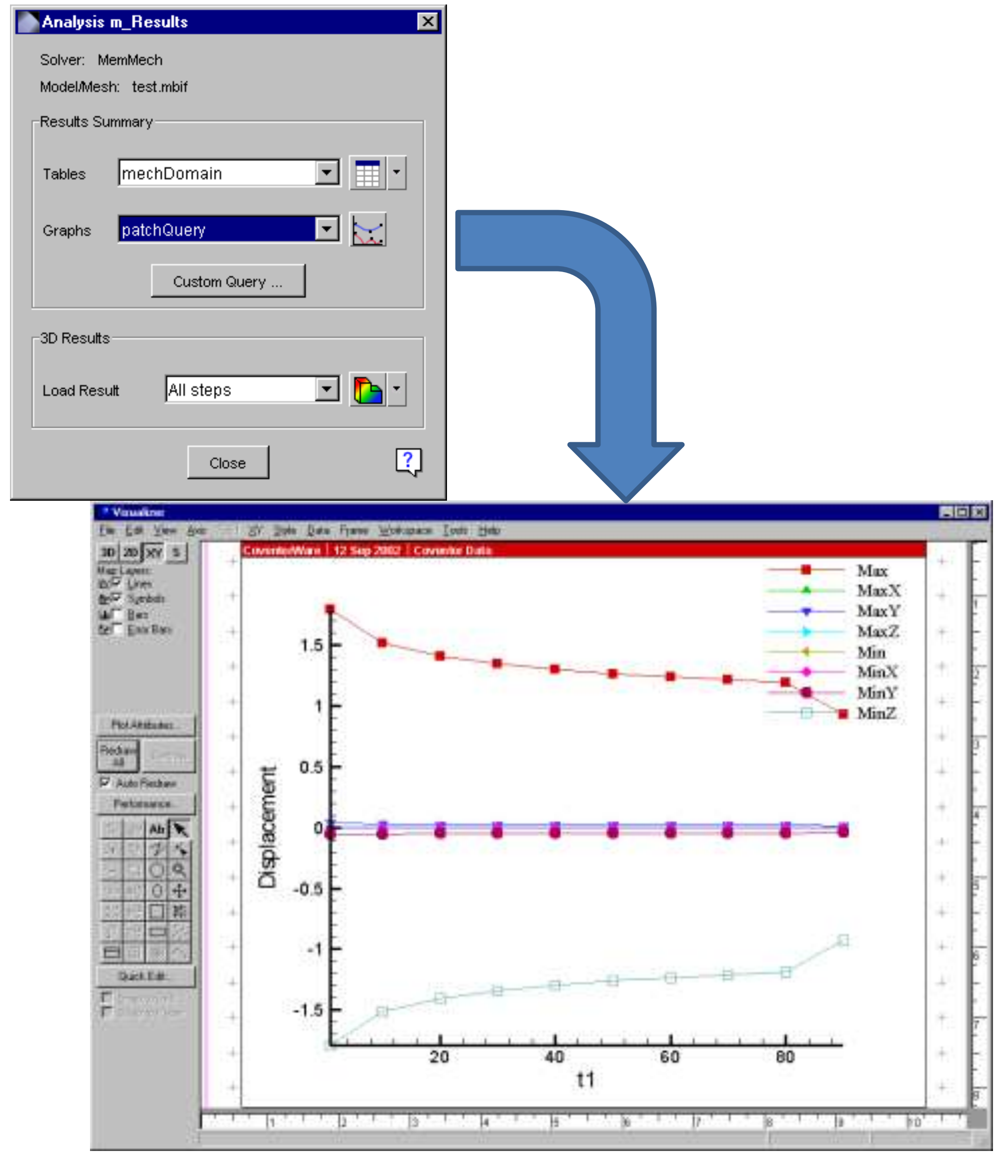
Use the trajectory window to define an incremental range of steps, eg. From 1 to 101 MPa

Apply the Mech BC the appropriate boundary condition



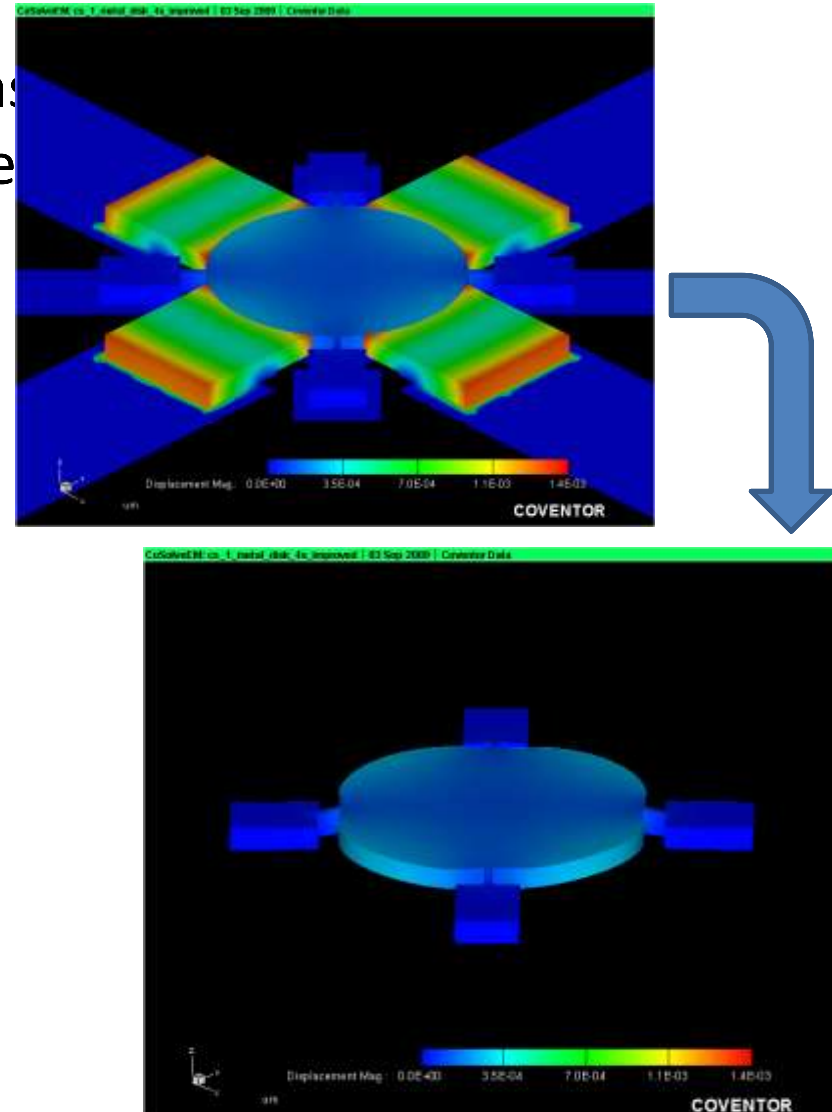
Visualization -2D

- Query gui is customized for each module.
- Simulation data tables can be exported.
- Control the visible lines using “Plot Attributes”, and hide/show the relevant zones.
- Control the axis range and type using the “Axis” menu.



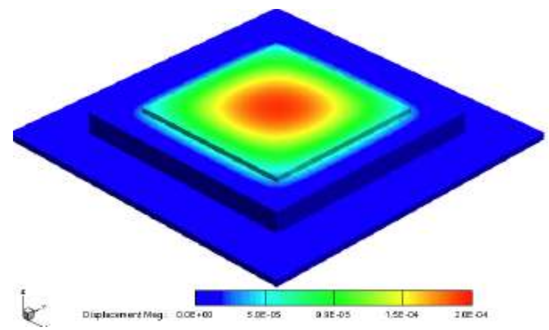
Visualization – 3D

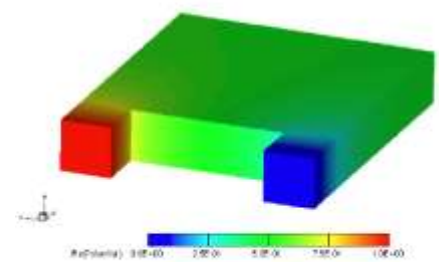
- Interactive slicing – cross section
- Transparent plots (show multiple plots at once)
- Data manipulation
  - Hide structures
  - Deform using displacements
- Plot annotation
- .avi movie output, plus user frame rate control and higher resolution capability.
- Values at all nodes in parabolic elements are now used – high resolutions

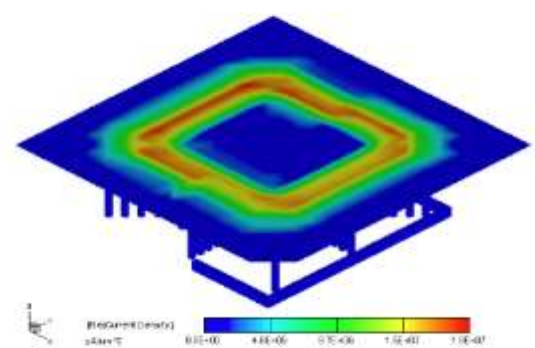


Advanced Solvers

- **PZE Solver** - MemMech can be configured for piezoelectric analysis.
  - Computes stress that develops when deformation is prevented or restrained by surrounding materials
  - Assumes linear coupling relationship between electrical displacement or field strength and mechanical factors.
- **MemPZR** - uses stress and material's PZR coefficients to compute the piezoresistive sensor's potential field and the resulting change in current.
  - Facilitates design of multiple piezoresistive sensors of arbitrary shape and size
  - To match process parameters, such as diffusion depth, the user may independently control the process-dependent geometry of each resistor.
- **MemHenry** - computes frequency-dependent resistance and inductance matrices for a set of conductors
  - Accurate, 3D computation can be applied to on-chip passive inductor analysis and parasitic extraction for packaging analysis.
  - Builds SPICE models, which can be configured to capture proximity effects.



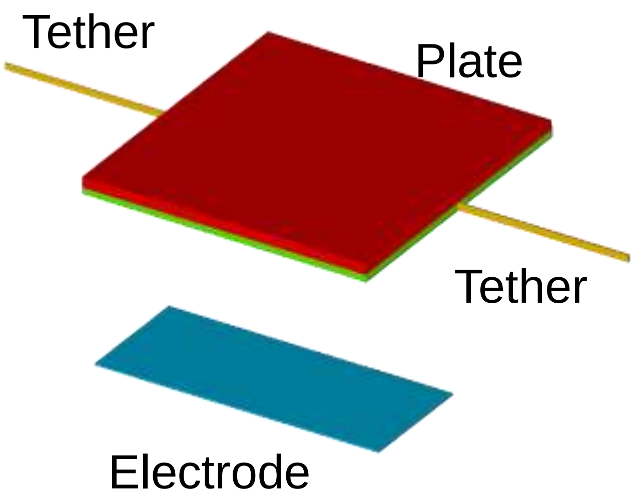


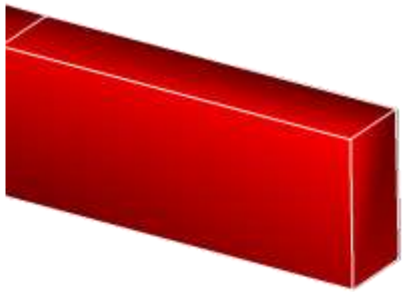


Appendix

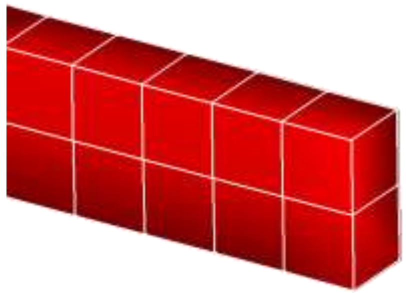
Meshing Study

- Single Axis Mirror Example
  - Separate devices into constituent parts
  - Optimize meshes for tether and plate separately
- As you change the density of the mesh, observe how results like displacement, stress, and rxn forces change

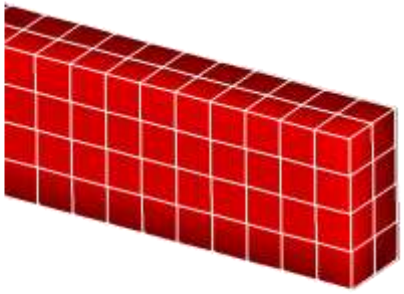




x=10 μm; y=2 μm; z=4 μm



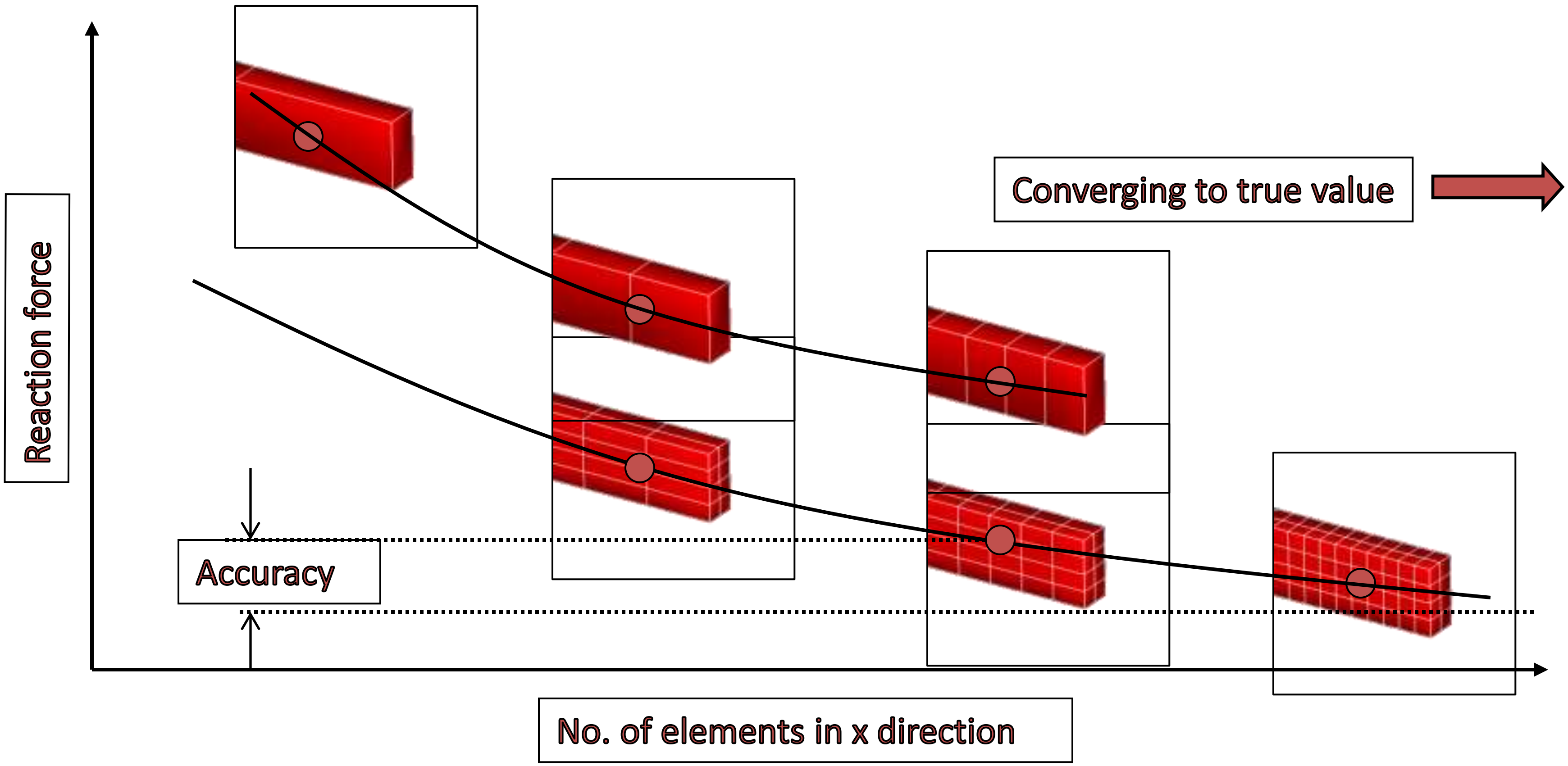
x=2 μm; y=2 μm; z=2 μm



x=1 μm; y=1 μm; z=1 μm

Meshing Study

- A quantity cannot be measured more accurately than the mesh that is used to resolve it!
- A mesh by itself does not guarantee accurate results – it must be verified as returning reliable data



MemMech PZE

- Structural nonlinearity in PZE procedures
  - Stress stiffening effect in structural, modal and harmonic analysis
- Direct integration steady state dynamic procedure used
- In comparison to modal based direct integration requires more resources but generates more accurate result

MemPZR

- Structural and PZR meshes can “live” in separate models
- For *Mechanical Piezoresistive* analysis structural part is set in MemMech. MemPZR GUI shows structural model
- In *Piezoresistive* analysis structural results must be imported