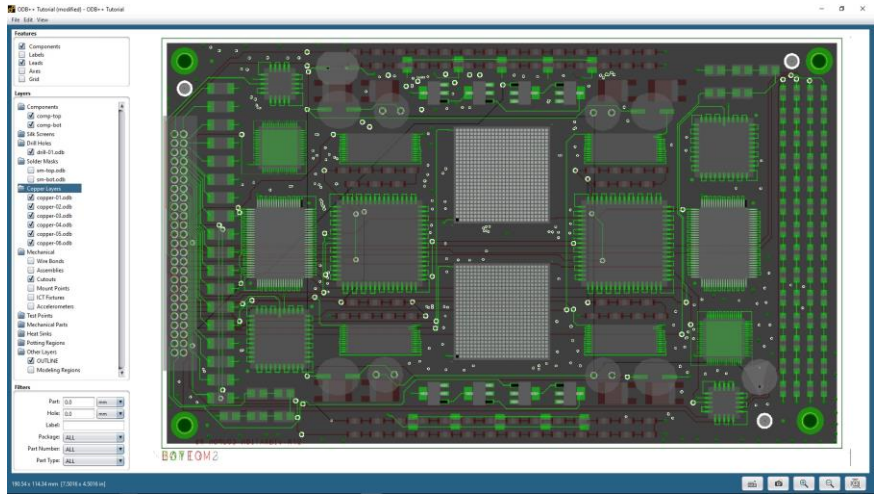
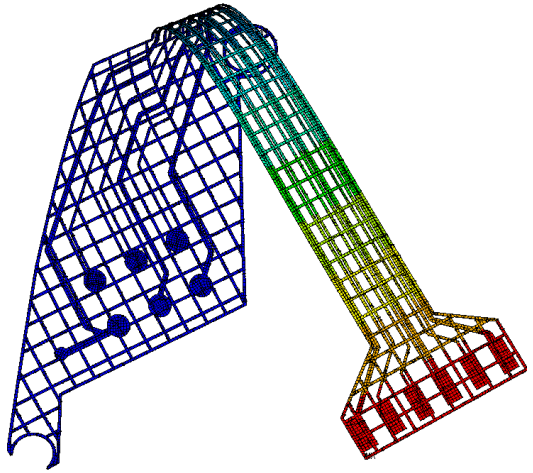
Release 2022 R1 Highlights

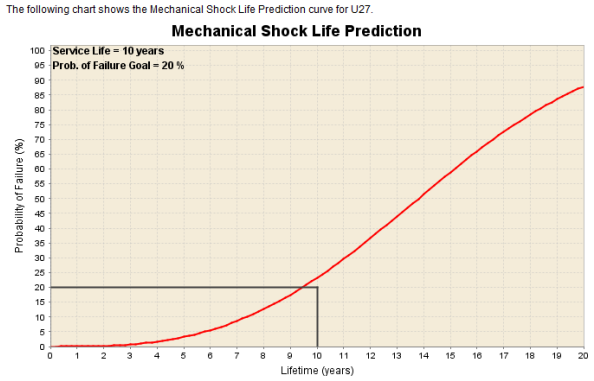
Ansys Sherlock & Electronics Reliability

Ansys Electronics Reliability Updates – 2022 R1

- Ansys Sherlock
  - Semi-Automated Reinforcement Element Workflow Solution
  - Sherlock-Icepak – EDB File Export
  - Import of GDSII/EDB Files
  - Sherlock Automation APIs
  - Sherlock-optiSLang Connection Details
  - General Functionality Enhancements
  - Documentation Updates
- Ansys Mechanical
  - Trace Mapping – Support for Solid-Shell Elements
  - Reinforcement Element Enhancements
- Ansys LS-DYNA
  - Multiscale Analyses
  - Solder Reflow Simulations
- Ansys Icepak
- Ansys AEDT Mechanical
- Additional Resources

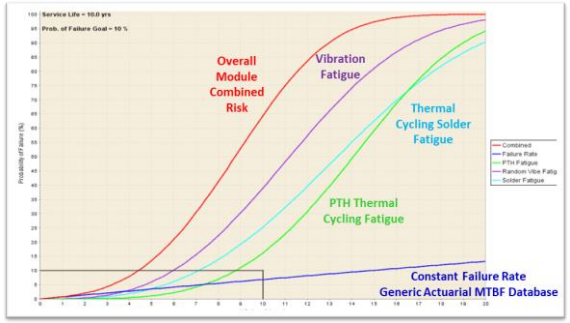




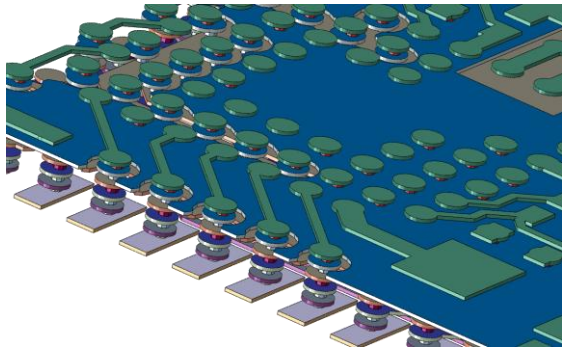


Ansys Sherlock

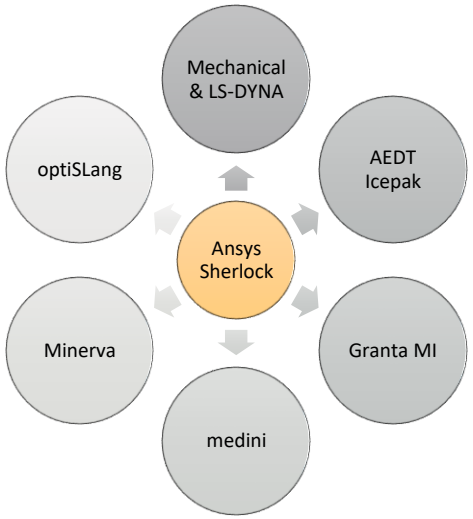
Ansys Sherlock – Typical Uses



Sherlock for Reliability



Sherlock as a Pre-Processor

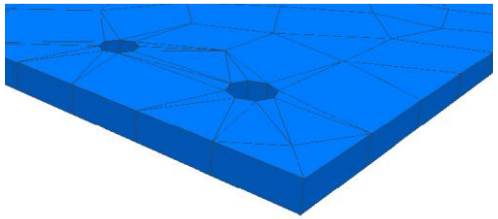


Sherlock Integrations

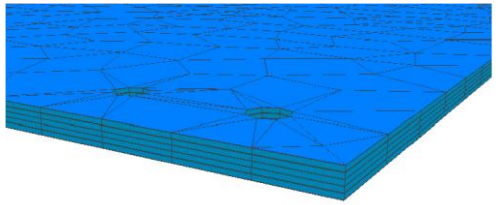
PCB Modeling Approaches

- Ansys provides leading solutions for the modeling of PCBs and Electronic Components. Multiple levels of fidelity supported.
- Start by importing ECAD files directly into Ansys tools, such as Ansys Sherlock and Ansys Mechanical.

Homogenized/Lumped

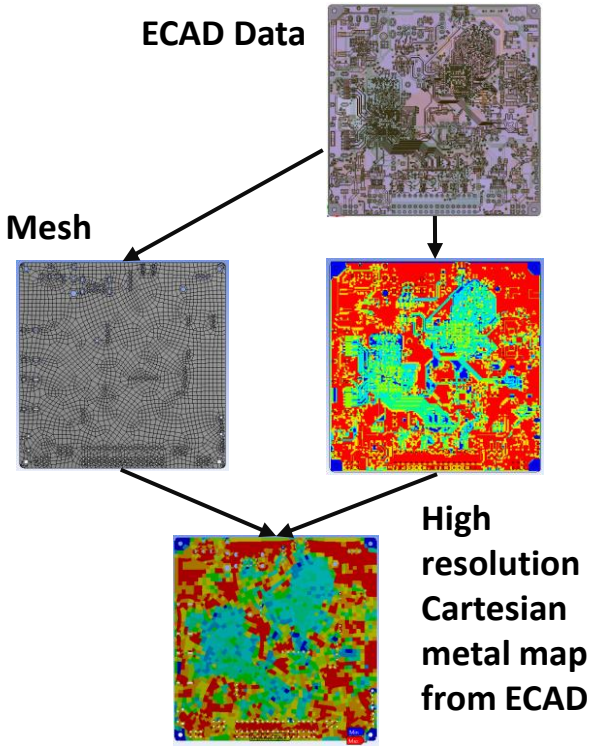


Lumped over entire board



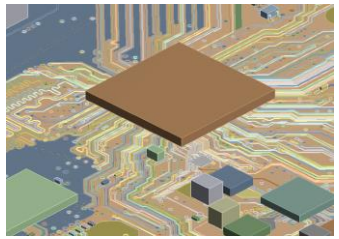
Lumped by Layer

Trace Mapping



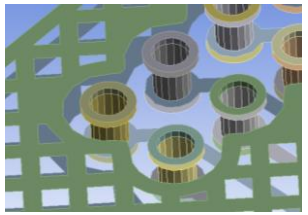
Element-wise material mapping

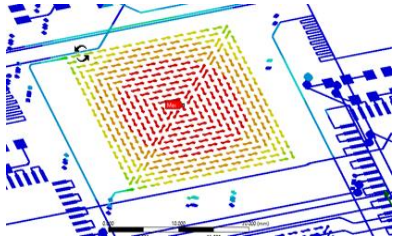
Reinforcements



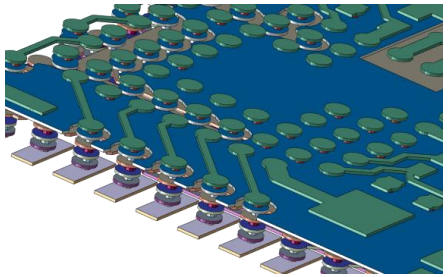
Embed surfaces (traces) and beams (vias) within a base mesh

Improved accuracy without full 3D Trace Modeling

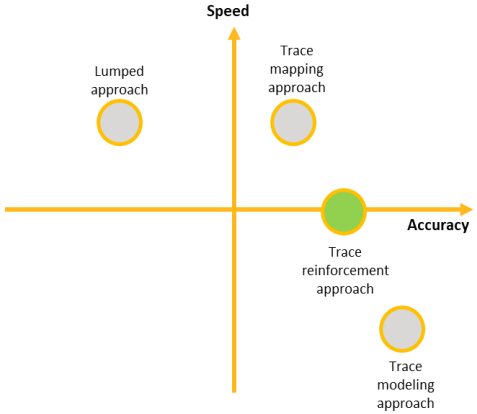




Trace Modeling



Full 3D Detail



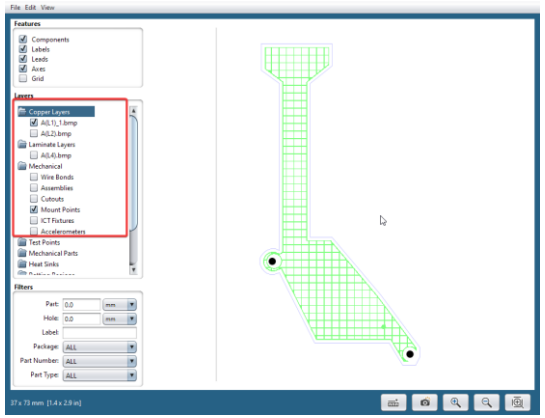
New in 2022 R1: Semi-Automated Reinforcement Element Workflow

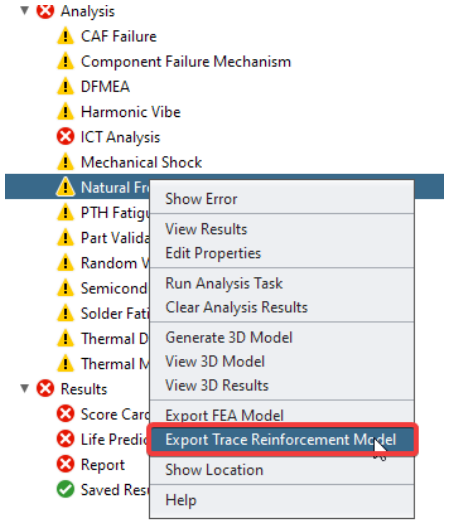
Time-Saving automation tasks performed upon export from Ansys Sherlock:

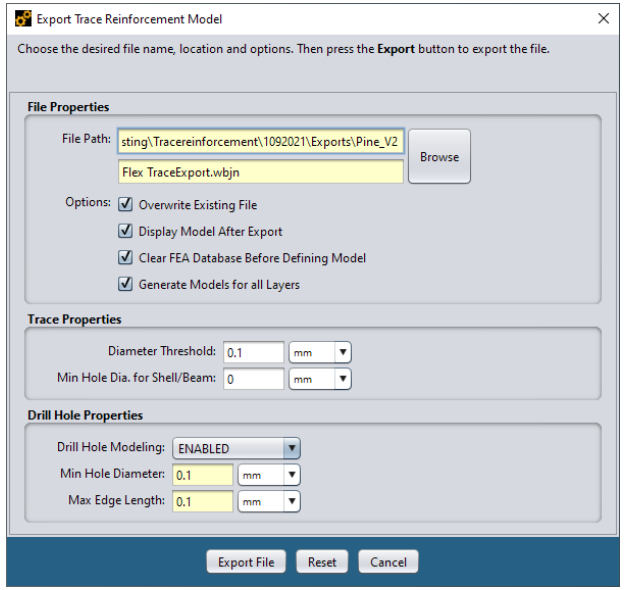
Automation of Ansys Workbench/Mechanical setup with:

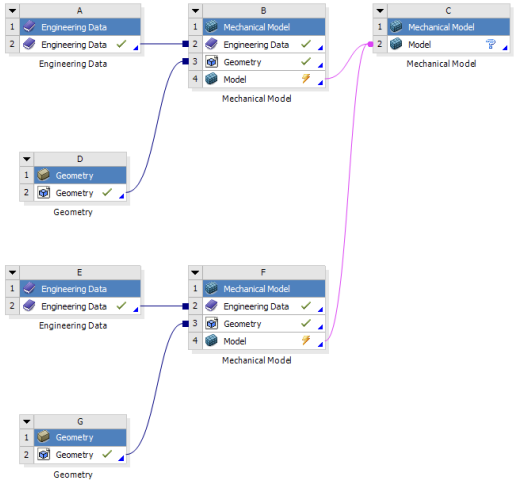
- Assigns thickness for Reinforcement Surface Bodies
- Assigns material properties
- Assign cross-section for the reinforcement Beams.
- Contacts between all the reinforcement bodies are automatically been removed.
- Generate layer based Named Selections

Semi-Automated Export and Model Build





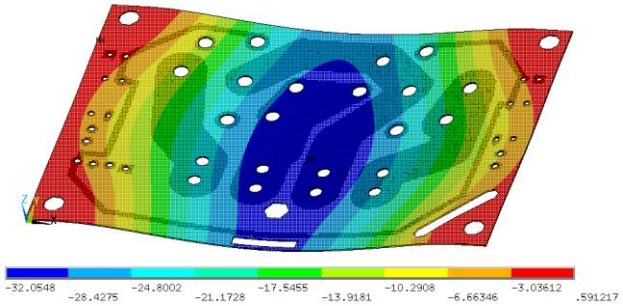




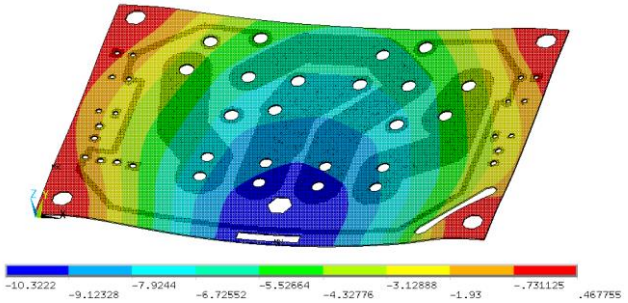
New in 2022 R1: Bending Stiffness for Smeared Reinforcement

- Greatly improves the solution accuracy with 3D smeared reinforcing (REINF) models
- Eliminates the need to use multiple REINF layers to capture the bending stiffness
- Enhances the REINF modeling usability in the new PCP/Chip simulation workflow

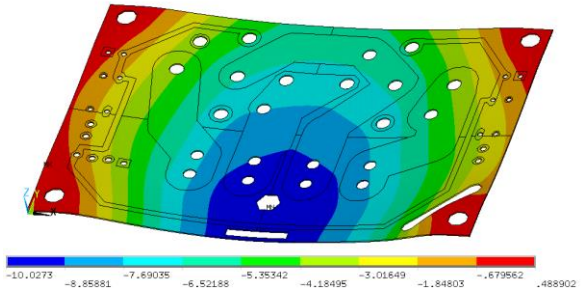
Accurate simulation results: REINF with bending vs. full 3D model



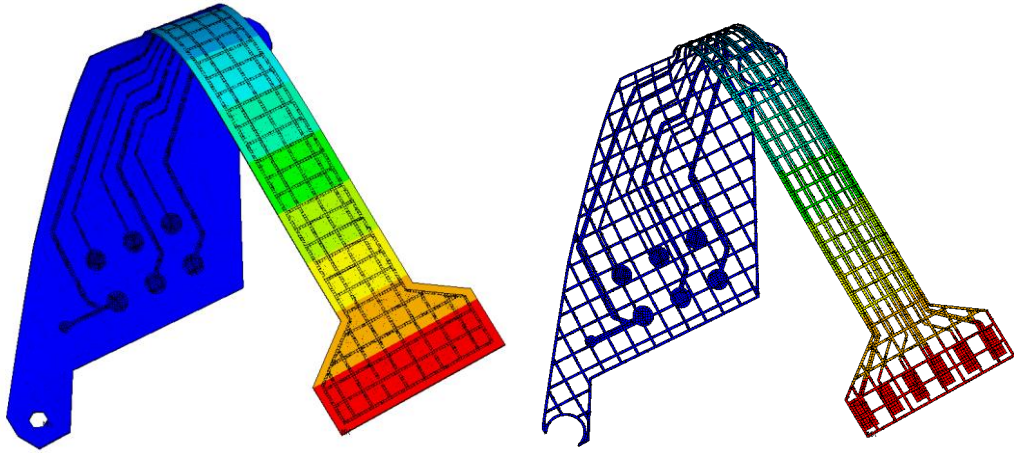
Reinforcing w/o bending stiffness



Reinforcing with bending stiffness



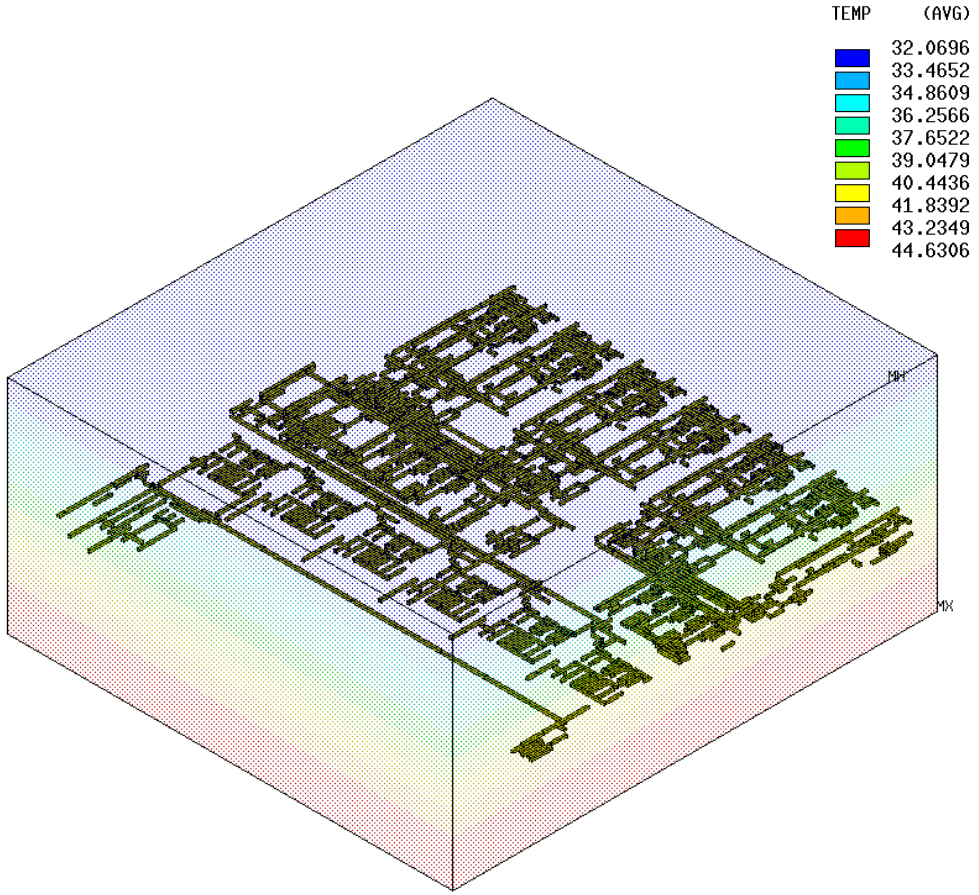
Full 3D model (trace modeling)



Robust and accurate simulation of flexible PCBs under large deformation

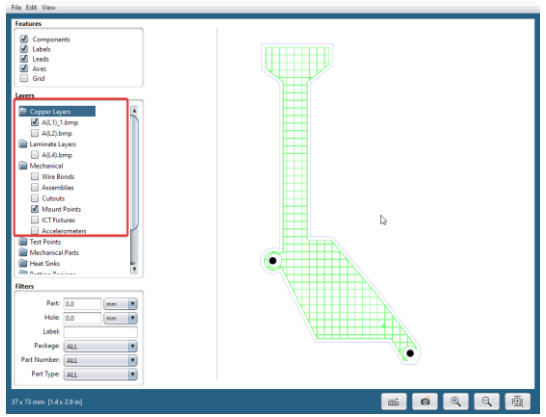
New in 2022 R1: Reinforcing Performance Enhancements

- Motivated by the requirements to account for large models (full PCB and chip models)
- Improved performance in pre-processing
  - Allows large number of reinforcing members in one base element
  - Reduces time needed for load mapping
- Improved solution efficiency
- Improved performance in post-processing
  - Significantly reduced time for querying min/max member results
  - Improved inter-member result smoothing



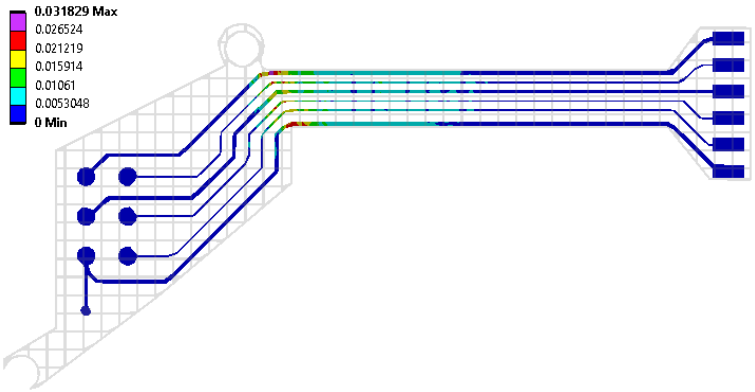
Application Highlight

Reinforcement Element Workflow Highlight – Flexible PCBs

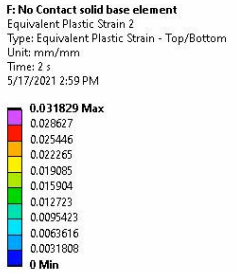


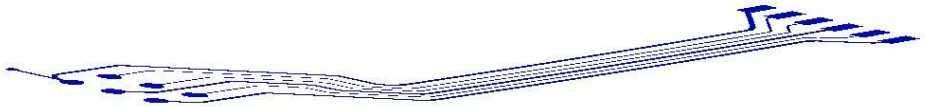
Sherlock

+



Mechanical





Sherlock-Icepak Connection

EDB File Export

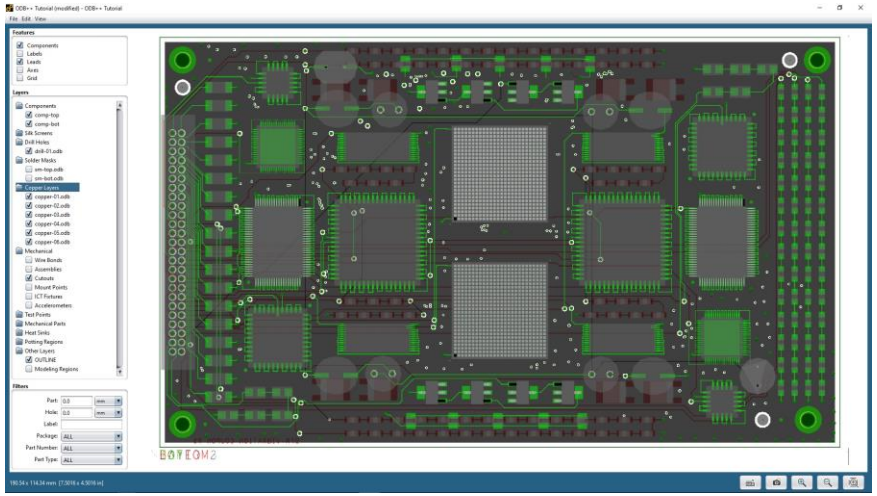
Ansys

New in 2022 R1: Sherlock-Icepak Connection (EDB File Export)

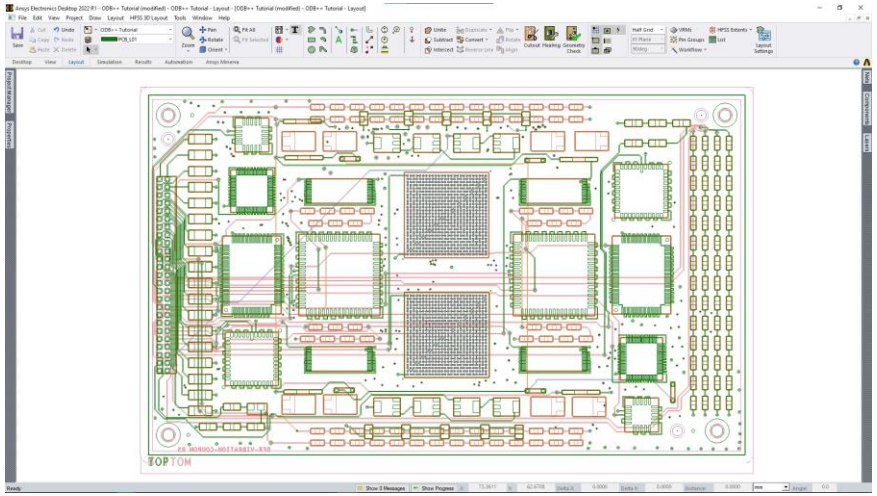
- Users can now export a CCA using the EDB format that is used by Ansys AEDT.
  - Ansys ECAD Database (*.def) file (Commonly referred to as an EDB file).
  - EDB files can then be imported directly in to AEDT.
  - If installed, AEDT can be opened automatically (including the resulting file import).
- Consequently, users can leverage the power of Ansys Sherlock to rapidly generate high-fidelity PCB models for use with Ansys AEDT Icepak.

<u>Supported Features in 2022 R1</u>:

- Stackup Layers
- Board Outline
- Holes
  - Vias
  - Plated Through-Holes
  - Non-Plated Through-Holes
- Traces
- Cutouts
- Components
- Pads (Stored as Pins)
- Material Properties



Ansys Sherlock

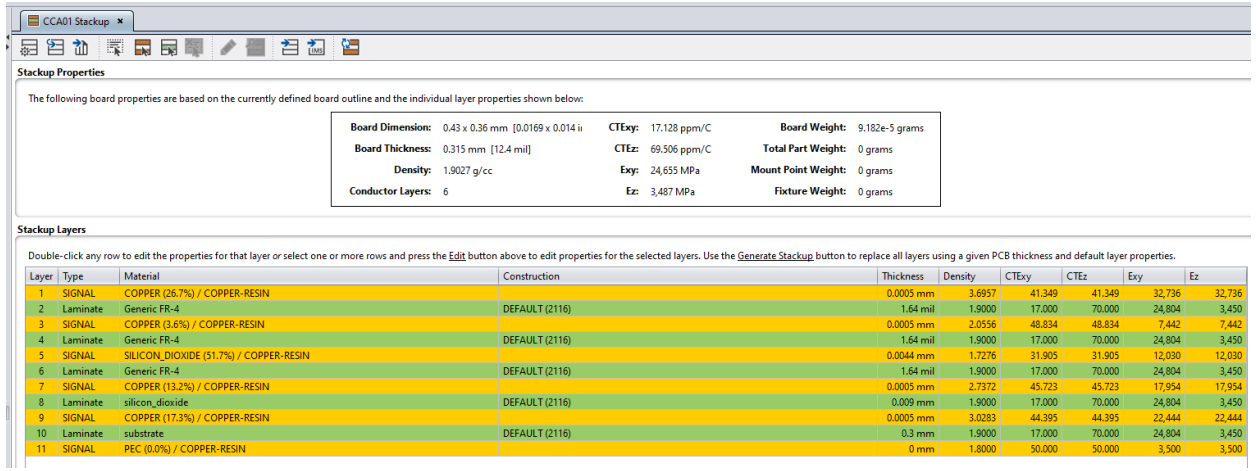


Ansys AEDT/Icepak

GDSII/EDB Import

New in 2022 R1: GDSII/EDB Import

- Sherlock now provides the ability to import stackups, stackup layers, and board outlines from an Ansys ECAD Database (*.def) file. From the main menu, select **Project > Import Sherlock EDB File**.
- In Sherlock, you can now import the stackup and stackup layers from a GDSII file. From the main menu, select **Project > Import GDSII File**.
- The ability to import GDSII files facilitates the preparation of Mechanical Models of Chip- and Die-Level, and other electronics structures for Thermal-Mechanical and other studies.



CCA01 Stackup

Stackup Properties

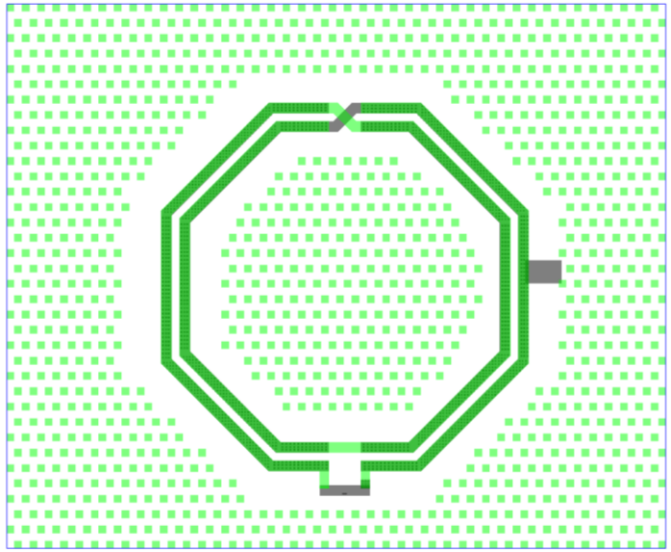
The following board properties are based on the currently defined board outline and the individual layer properties shown below:

Board Dimension:	0.43 x 0.36 mm [0.0169 x 0.014 i	**CTExy:**	17.128 ppm/C	**Board Weight:**	9.182e-5 grams
Board Thickness:	0.315 mm [12.4 mil]	**CTEz:**	69.506 ppm/C	**Total Part Weight:**	0 grams
Density:	1.9027 g/cc	**Exy:**	24,655 MPa	**Mount Point Weight:**	0 grams
Conductor Layers:	6	**Ez:**	3,487 MPa	**Fixture Weight:**	0 grams

Stackup Layers

Double-click any row to edit the properties for that layer or select one or more rows and press the Edit button above to edit properties for the selected layers. Use the Generate Stackup button to replace all layers using a given PCB thickness and default layer properties.

Layer	Type	Material	Construction	Thickness	Density	CTExy	CTEz	Exy	Ez
1	SIGNAL	COPPER (26.7%) / COPPER-RESIN		0.0005 mm	3.6957	41.349	41.349	32,736	32,736
2	Laminate	Generic FR-4	DEFAULT (2116)	1.64 mil	1.9000	17.000	70.000	24,804	3,450
3	SIGNAL	COPPER (3.6%) / COPPER-RESIN		0.0005 mm	2.0556	48.834	48.834	7,442	7,442
4	Laminate	Generic FR-4	DEFAULT (2116)	1.64 mil	1.9000	17.000	70.000	24,804	3,450
5	SIGNAL	SILICON_DIOXIDE (51.7%) / COPPER-RESIN		0.0044 mm	1.7276	31.905	31.905	12,030	12,030
6	Laminate	Generic FR-4	DEFAULT (2116)	1.64 mil	1.9000	17.000	70.000	24,804	3,450
7	SIGNAL	COPPER (13.2%) / COPPER-RESIN		0.0005 mm	2.7372	45.723	45.723	17,954	17,954
8	Laminate	silicon_dioxide	DEFAULT (2116)	0.009 mm	1.9000	17.000	70.000	24,804	3,450
9	SIGNAL	COPPER (17.3%) / COPPER-RESIN		0.0005 mm	3.0283	44.395	44.395	22,444	22,444
10	Laminate	substrate	DEFAULT (2116)	0.3 mm	1.9000	17.000	70.000	24,804	3,450
11	SIGNAL	PEC (0.0%) / COPPER-RESIN		0 mm	1.8000	50.000	50.000	3,500	3,500



Sherlock Automation APIs

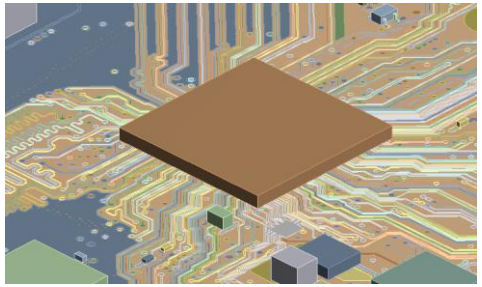
Ansys

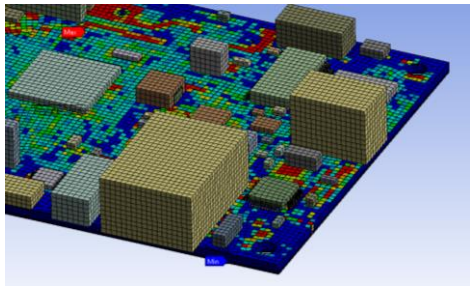
Scripting and Automation – Example Workflows

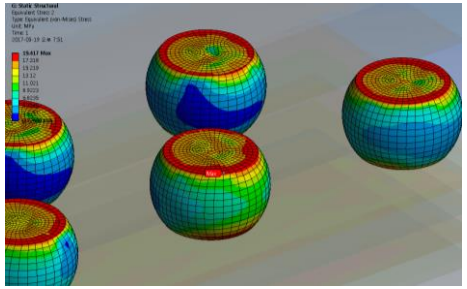
- There are many possibilities for automation in the Electronics Reliability space.
  - From general Sherlock Automation to streamline everyday tasks, to partially- or fully-automated workflows featuring leading Ansys solutions (such as the Reinforcement Element workflow).

Python Example

```
import SherlockLifeCycleSer
import SherlockLifeCycleSer
import grpc

channel = grpc.insecure_cha
```







Sherlock General Automation

Pre-Processing Automation and Results Extraction

End-to-End Workflow Automation

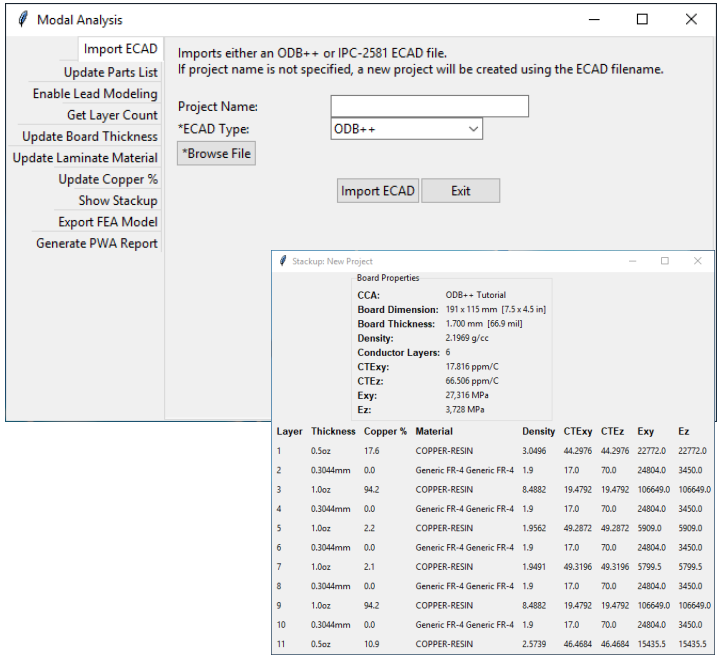
Advanced Applications

Ansys Sherlock Scripting - APIs

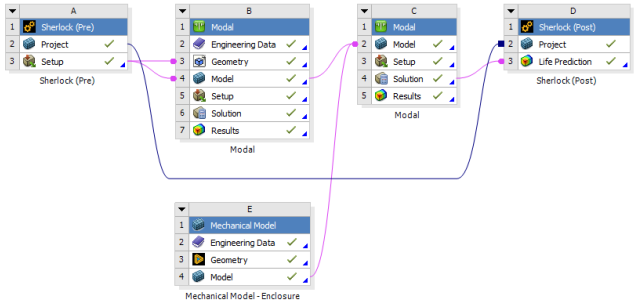
- Application Programming Interfaces (APIs) are available for Ansys Sherlock.
  - APIs – Flexible framework allows for development in several languages, including Python.
  - Additional APIs that address additional features are being planned for future releases.

- Enables users to:
  - Run Simulations in Batch
  - Automate processes and standardize methods
  - Explore the impact of design and event variations on life prediction and other metrics.

- Automation Feature Examples
  - Open existing projects
  - Import ECAD files
  - Life Cycle and Event Creation
  - Update Parts from Parts Library
  - Query and modify Stackup Properties
  - Run Analyses, such as Solder Fatigue, Natural Frequency, Random Vibration, etc.



Custom Toolkit Leveraging Sherlock APIs

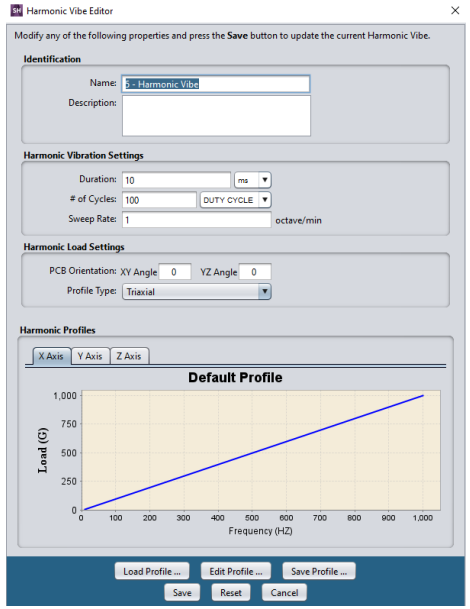


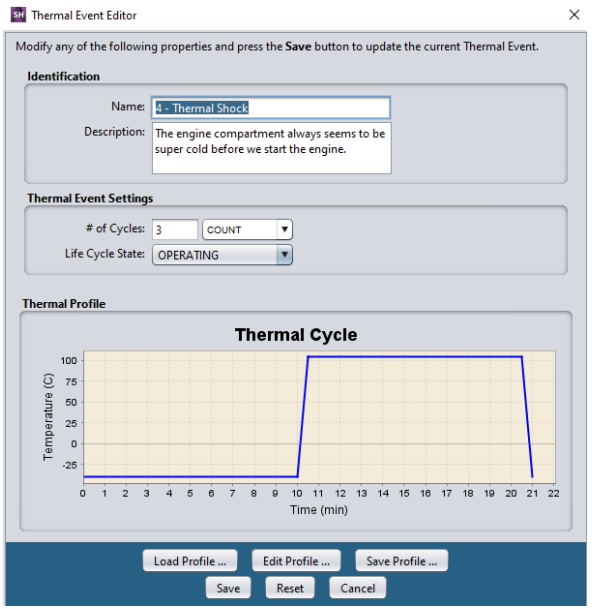
Sherlock-WB Connection
Leveraging Sherlock APIs

Ansys Sherlock API Examples

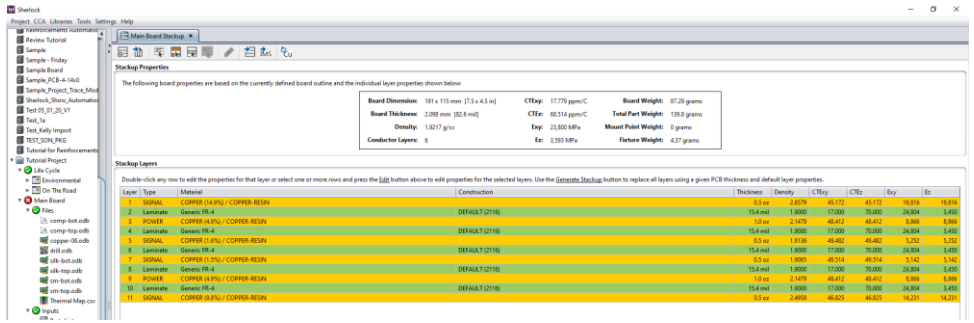
Python Example

```
import SherlockLifeCycleService_pb2
import SherlockLifeCycleService_pb2_grpc
import grpc

channel = grpc.insecure_channel('localhost:9090')

stub = SherlockLifeCycleService_pb2_grpc.SherlockLifeCycleServiceStub(channel)

message = SherlockLifeCycleService_pb2.AddHarmonicEventRequest()

message.project = "Tutorial Project"
message.phaseName = "On The Road"
message.eventName = "Test Harmonic API"
message.description = "Test Description"
message.duration = 2
message.durationUnits = "ms"
message.numOfCycles = 100
message.cycleType = "DUTY CYCLE"
message.sweepRate = 1
message.orientation = "1, 3"
message.profileType = "Uniaxial"
message.loadDirection = "0, 0, -1"

response = stub.addHarmonicEvent(message)

print(str(response))
```

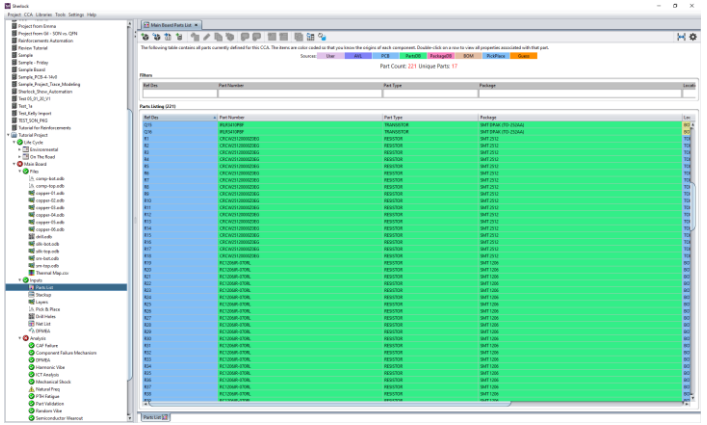




Automate Creation and Re-Use of Life Cycles and Events



Stack-Up Information



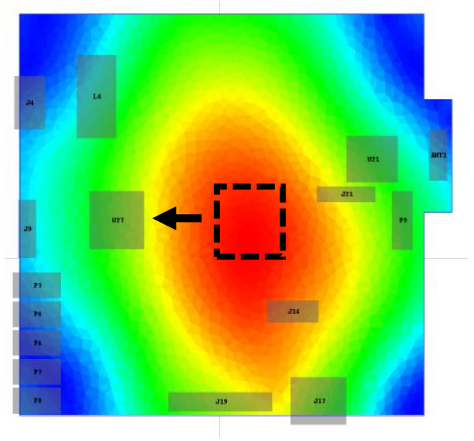
Update Parts from Libraries

New in 2022 R1: Part Location APIs

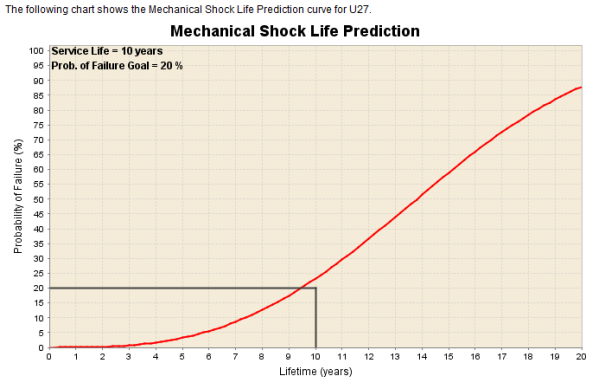
Part Location APIs can help users rapidly evaluate the trade-offs associated with moving components on a board subjected to a variety of loading conditions.

New Part Location APIs

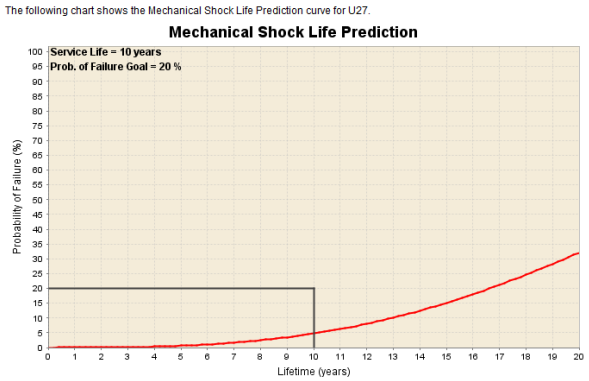
- setPartLocation() API to set a part's location properties.
- getPartLocationUnits() API to get a list of valid part location units.
- getBoardSides() API to get a list of valid board side values.
- getPartLocation() API to get a part's location properties.



Move Component?



Reliability – Move Left



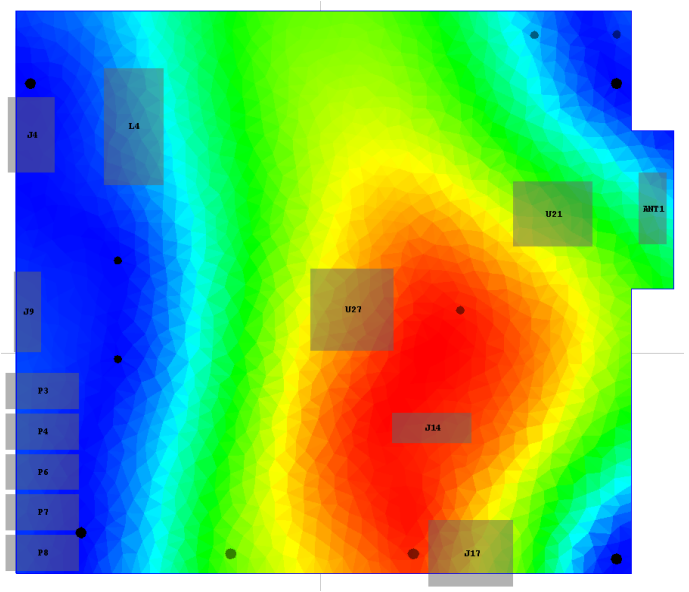
Reliability – Move Right

New in 2022 R1: Mount Point APIs

Mount Point APIs can help users rapidly get available Mount Point information programmatically.

New Mount Point APIs

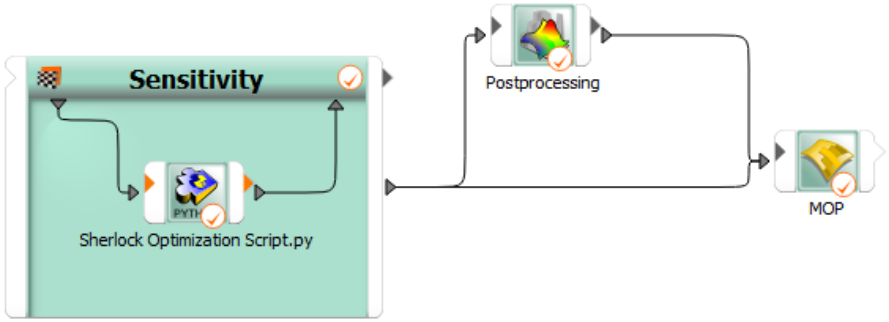
- getMountPointTypes() API to get a valid list of mount point types.
- getMountPointShapeTypes() API to get a valid list of mount point shape types.
- getMountPointSides() API to get a valid list of mount point sides.
- getMountPointUnits() API to get a valid list of mount point units.
- getMountPointBoundaries() API to get a valid list of mount point boundaries.
- getMountPointChassisMaterials() API to get a valid list of mount point chassis materials.



Sherlock-optiSLang Connection

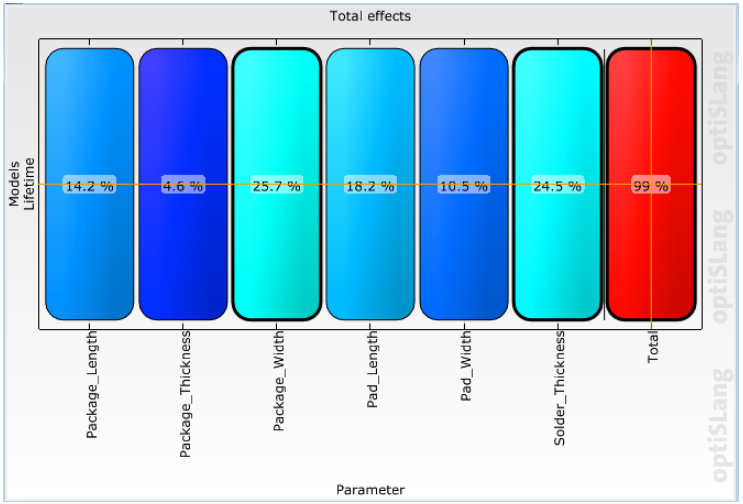
Ansys Sherlock-optiSLang Studies

- Ansys Sherlock APIs allow users to programmatically study the effects of parameterizing key variable, such as:
  - Part Properties, Component Locations, and Stackup Information on key output, such as Time-to-Failure.
- Scripts can be incorporated as a part of Sensitivity Studies and Optimization routines in Ansys optiSLang.
- Examples:
  - Determine the influence of key Part Properties on Cycles-to-Failure for components subjected to Thermal Cycling.
  - Rapidly evaluate the trade-offs associated with moving components on a board subjected to Mechanical Shock, Random Vibration, etc.



Sample Ansys Sherlock Code Snippet introducing Part Property Parameters

```
Solder_Thickness = 0.0635
  Pad_Length = 1.45
  Pad_Width = 3.5
  Pad_Pitch = 6.40
  Package_Length = 6.3
  Package_Width = 3.15
  Package_Thickness = 0.6

newData = []
with open('D:\\Backedup\\ACT\\PartsList.csv','r') as f:
  reader = csv.reader(f)
  for l in csv.reader(f, quotechar='"', delimiter=',',
quoting=csv.QUOTE_ALL):
    newData.append(l)


newData[48][125] = Solder_Thickness
newData[48][101] = Pad_Length
newData[48][104] = Pad_Width
newData[48][102] = Pad_Pitch
newData[48][95] = Package_Length
newData[48][100] = Package_Width
newData[48][98] = Package_Thickness
```

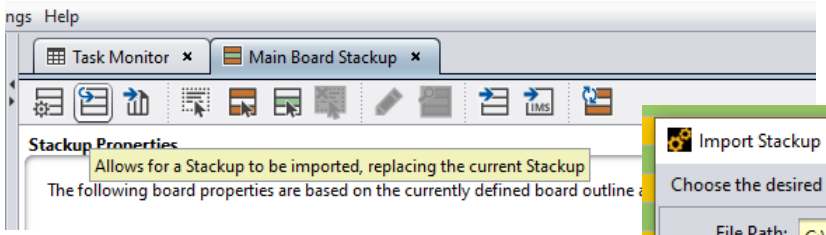


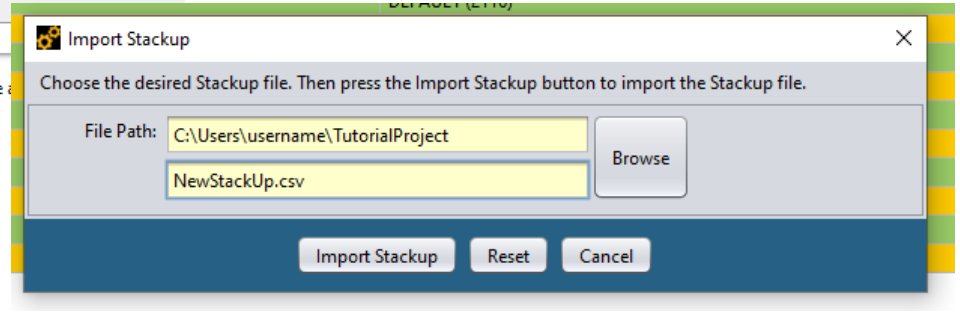
Sample Sensitivity Study Results using Ansys optiSLang

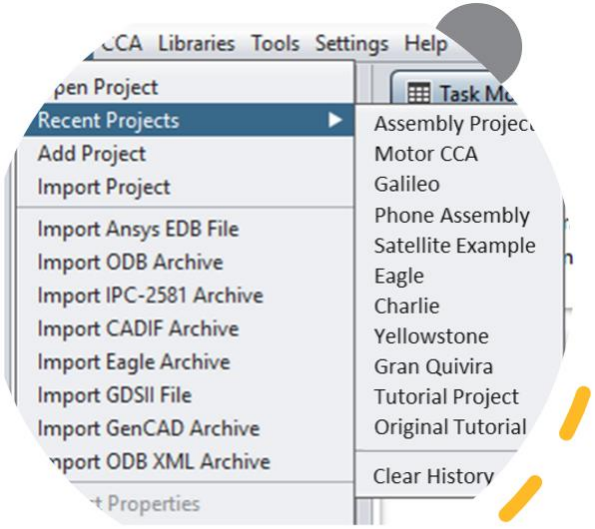
Sherlock General Enhancements

New in 2022 R1: Workflow Enhancements

- Added the ability to import a Stackup from CSV and XLS files.



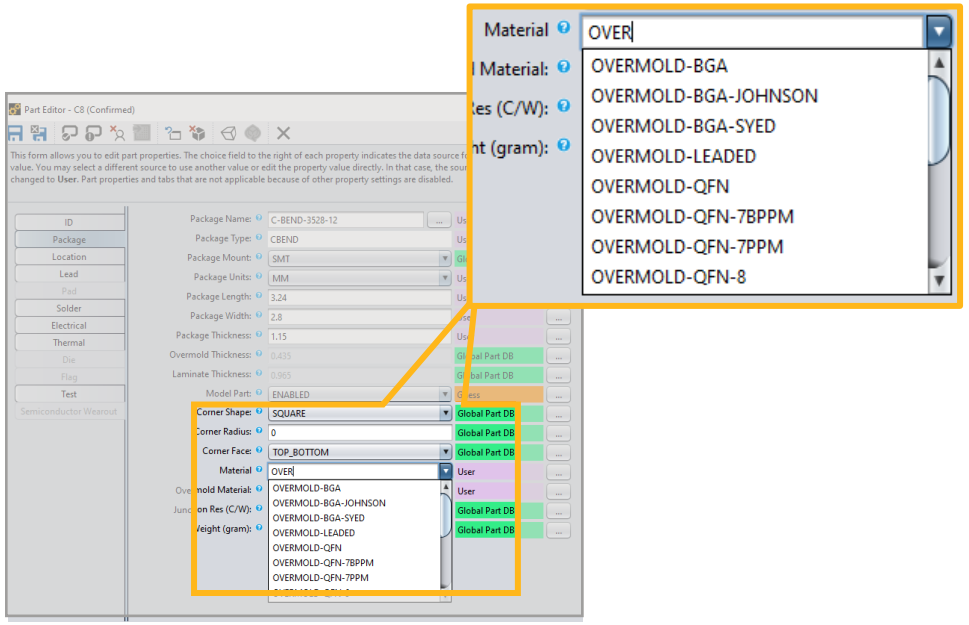


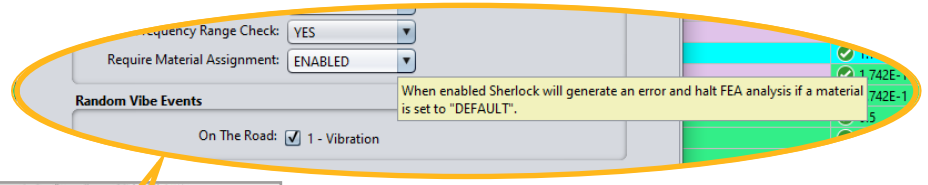


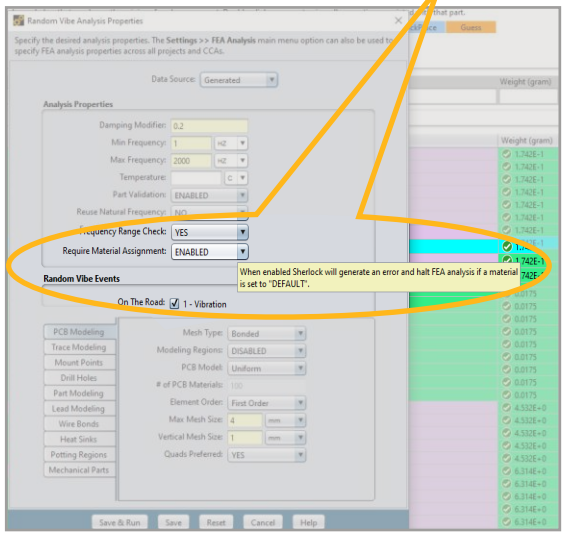
- In the ECAD import dialogs, the ten most recently imported ECAD files of the given file type are now displayed
- Similarly, the ten most recently imported FEA result files are presented in the FEA Analysis dialogs.

New in 2022 R1: Improved User Experience

- Improved the look and feel of icons and fonts for 4K displays.
- All material selectors in forms are now searchable.







- Added a **Require Material Assignment** option in an FEA analysis to check if any material is unassigned.
- Sherlock saves console output in a log file for each session and is included with the project export.

Additional Sherlock Enhancements in 2022 R1

- Added option in Advanced Settings to display and use part body weight to update part body's material density in FEA analyses only. Refer to Sherlock Part Properties in the Sherlock User's Guide.
- Sherlock now provides the ability to edit the specific heat property of materials in the Materials Library. Some materials have a specific heat property set already. This is useful for exporting to Electronics Desktop so that it can be used for Icepak simulations.
- Improved initial loading time of the Part Viewer and editor dialogs.
- Updated package model forms to hide the package thickness field for BGA packages. BGA packages use the **Overmold** and **Laminate Thickness** fields instead.
- Added a **Require Material Assignment** option in an FEA analysis to check if any material is unassigned. If this option is enabled, the FEA analysis will stop as soon as it finds a material that is not assigned. If this option is not enabled and Sherlock finds material that is not assigned, a message appears in the Results Viewer.
- Added information tooltips to the **Stackup Properties** and **Stackup** table.
- Updated the **Parts List** and **Part Editor** so they no longer update the pad properties of QFN packages when selecting **Update Pad Properties**.
- Updated the Part Editor to disable the solder tab if a BGA package is selected.
- Updated the Package Editor so that when a BGA package is added, the package does not have to be saved before adding ball parameters

Additional Sherlock Enhancements in 2022 R1

- In the ECAD import dialogs, the ten most recently imported ECAD files of the given file type are displayed, a helpful shortcut when reimporting a file. In addition, users can now see the ten most recently imported FEA result files in the FEA Analysis dialogs. The displayed files are specific to the combination of the CCA name and the FEA Analysis name.
- The context menu from the Stackup Table allows the associated layers to be opened in the Layer Viewer. The Layer Viewer may also be opened from the Stackup for selected layers by clicking the **View Stackup** icon in the toolbar.
- Added a context menu for copper layers in the Layer Viewer which allows for Trace Modeling options such as **Generate Trace Model** and **Export Trace Model**. Added a context menu to the **Copper Layers** folder in the Layer Viewer which allows for Trace Model generation, viewing, and clearing of all copper layers on the circuit card.
- Added **Quartz Controlled**, **Voltage Controlled**, **Temperature Controlled**, and **Oven Controlled** as sub-types for **MISC CRYSTAL** in failure class (helpful for the Sherlock-medini workflow).
- Sherlock no longer displays repeated warnings about contact elements during random vibration results input (summary warning provided).
- When performing a Random Vibration analysis where no zero-crossing frequency is found, Sherlock will include this information in the analysis errors to help provide insight into why the analysis was not complete.
- For BGA's, Sherlock uses the ball material defined on the **Ball** tab of the Parts List instead of using the solder material defined on the **Solder** tab when performing Solder Joint Fatigue analysis.
- And more – Please refer to the Ansys Sherlock 2022 R1 Release Notes.

Documentation Updates in Ansys Sherlock 2022 R1

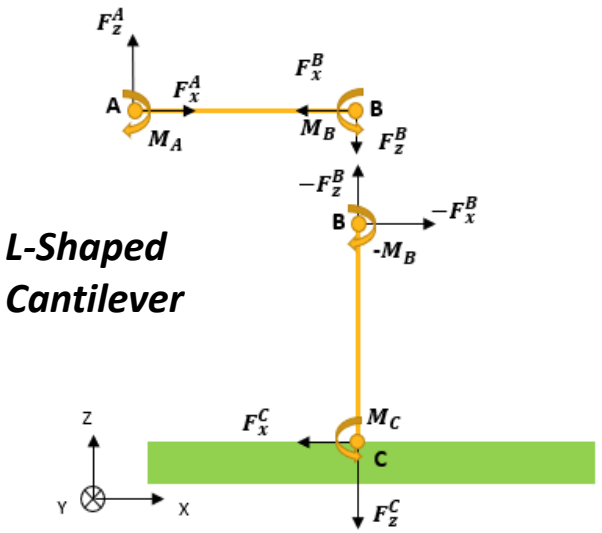
- Available Theory Manuals
  - Solder Fatigue Caused by Thermal Cycling
    - BGA Model
    - Quad Flat No-Lead (QFN) Package
  - Thermal Mech
  - In-Circuit Testing (ICT)
  - Random Vibration
  - Mechanical Shock
- User Guide Updates
  - Overhauled ‘Ansys Workbench Integration’ Chapter to include the latest updates.
  - Improved ‘Importing Projects and ECAD Archives’ sections.

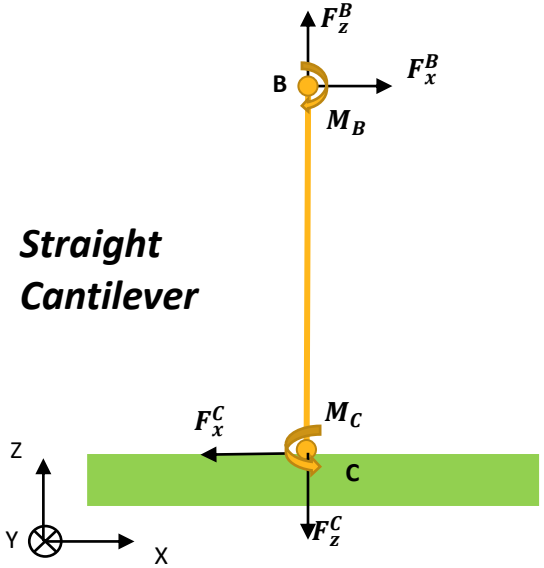
Sherlock Beta Features

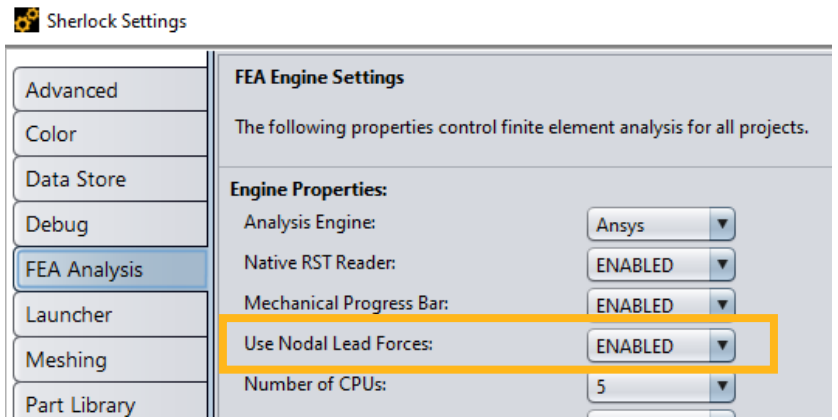
New in 2022 R1: Beta Feature –

Analytical Beam Approach for TTF Predictions for Leaded Components

- Feature is in Beta in Ansys 2022 R1
  - Available for ICT and Random Vibe
  - Debug Settings Flag needed (enableLeadNodalForces)
- Updates to minimize the impact of strain singularities on TTF predictions
  - Implemented an analytical beam model from the extracted forces and moment.



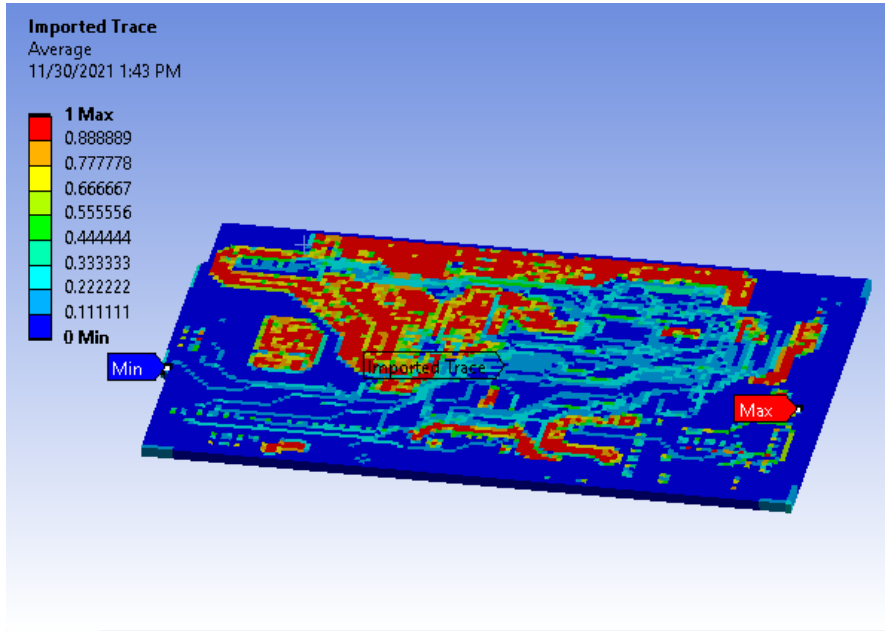




Ansys Mechanical

Support for Solid-Shell Elements in Trace Mapping

- Geometries meshed with Solid-Shell (SOLSH) elements are now supported in the trace mapping workflow.
- The behavior is like that of Shell Trace Mapping, where trace data is controllable on a per-layer basis.
- Dielectric and Trace materials can be defined on a per-layer basis.
- This approach should provide users with the accuracy they are targeting with less computational expense compared to Solid Element Trace Mapping.



Geometry | Worksheet

Data View

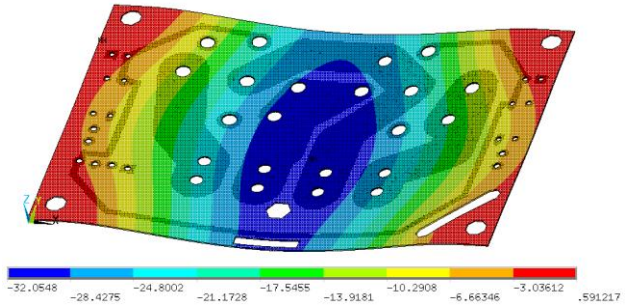
Imported Trace

	Layer	Thickness (mm)	Trace Material	Dielectric Material	Mechanical Layer No.	Active
1	top	0.03556	Copper Alloy	FR-4	11	☑
2	Dielectric_1	0.06731	Copper Alloy	FR-4	10	☑
3	pwr	0.03556	Copper Alloy	FR-4	9	☑
4	Dielectric_2	1.27	Copper Alloy	FR-4	8	☑
5	lyr_1	0.03556	Copper Alloy	FR-4	7	☑
6	Dielectric_3	0.2032	Copper Alloy	FR-4	6	☑
7	lyr_2	0.03556	Copper Alloy	FR-4	5	☑
8	Dielectric_4	0.2032	Copper Alloy	FR-4	4	☑
9	gnd	0.03556	Copper Alloy	FR-4	3	☑
10	Dielectric_5	0.06731	Copper Alloy	FR-4	2	☑
11	bottom	0.03556	Copper Alloy	FR-4	1	☑

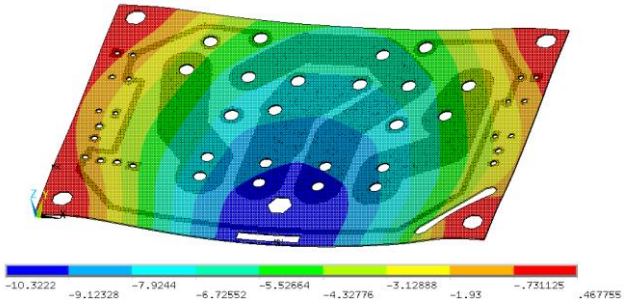
New in 2022 R1: Bending Stiffness for Smeared Reinforcement

- Greatly improves the solution accuracy with 3D smeared reinforcing (REINF) models
- Eliminates the need to use multiple REINF layers to capture the bending stiffness
- Enhances the REINF modeling usability in the new PCP/Chip simulation workflow

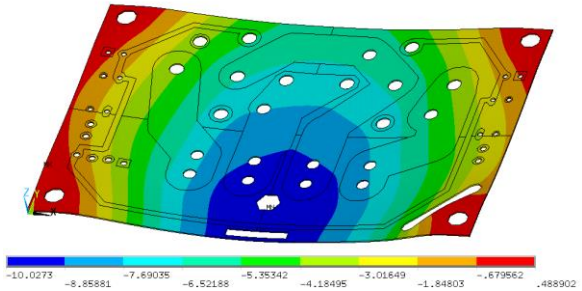
Accurate simulation results: REINF with bending vs. full 3D model



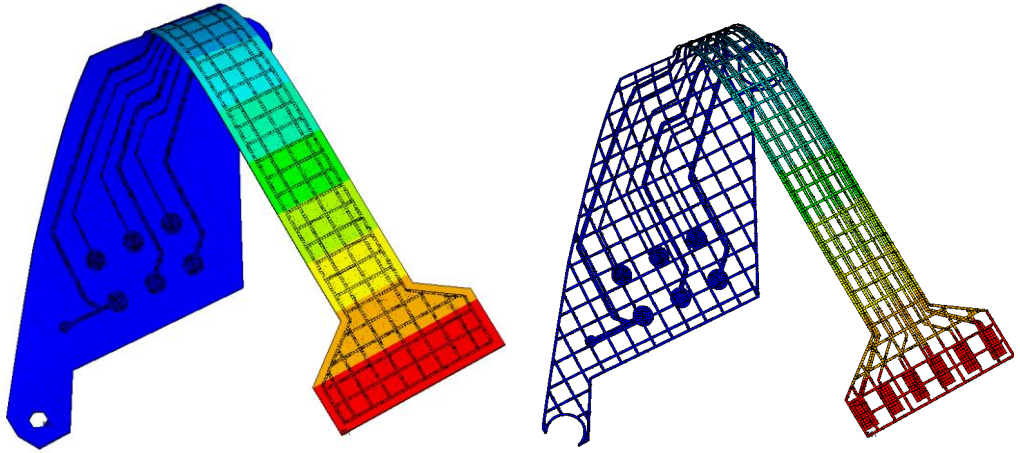
Reinforcing w/o bending stiffness



Reinforcing with bending stiffness



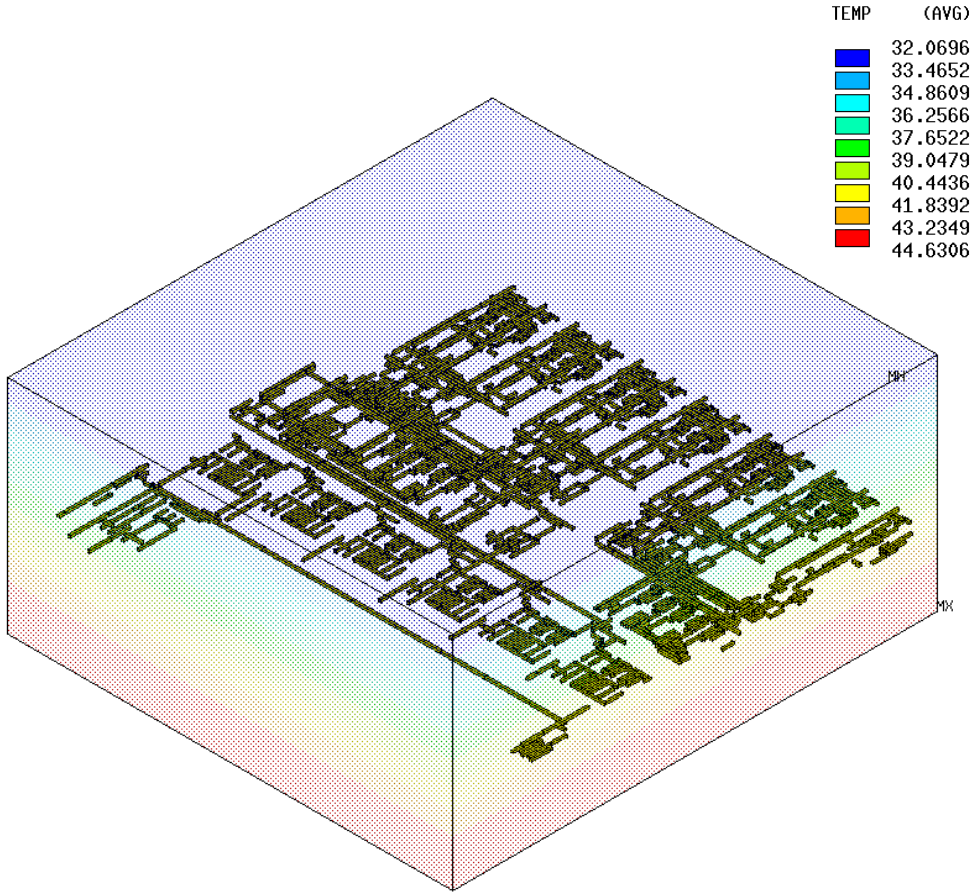
Full 3D model (trace modeling)



Robust and accurate simulation of flexible PCBs under large deformation

New in 2022 R1: Reinforcing Performance Enhancements

- Motivated by the requirements to account for large models (full PCB and chip models)
- Improved performance in pre-processing
  - Allows large number of reinforcing members in one base element
  - Reduces time needed for load mapping
- Improved solution efficiency
- Improved performance in post-processing
  - Significantly reduced time for querying min/max member results
  - Improved inter-member result smoothing



Ansys LS-DYNA

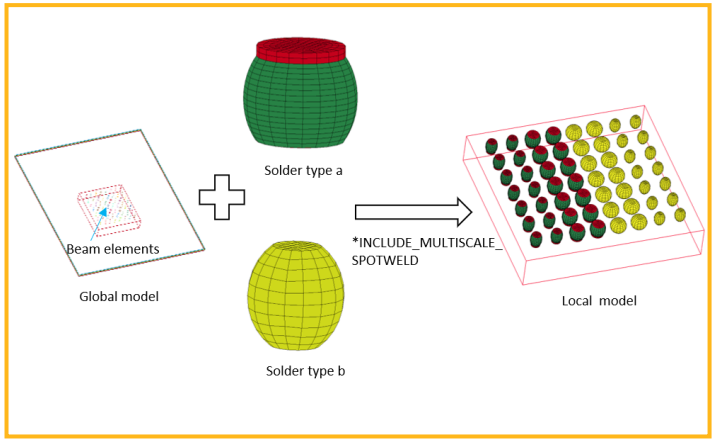
Ansys LS-DYNA Updates in 2022 R1

Please refer to the Ansys LS-DYNA 2022 R1 Update Presentation for additional details.

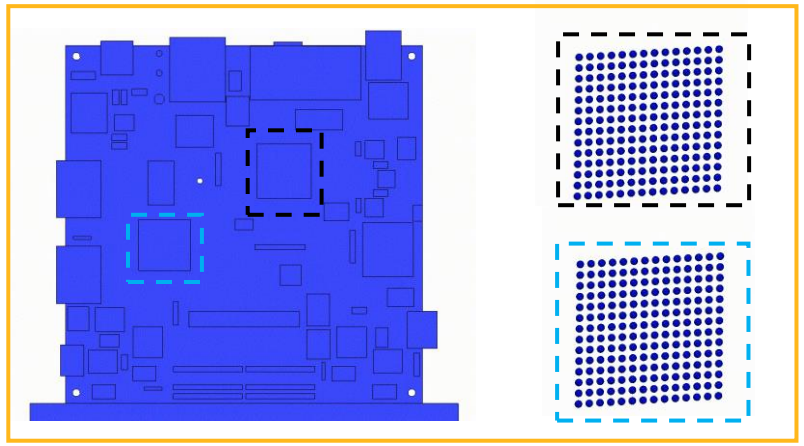
- **Ansys Mechanical Integration:**
  - Imported Displacement
  - Localization (multiple language support)
  - Cyclic Symmetry Support
  - Fluent to LS-DYNA Thermal Transfer(1 way)
  - Restart Improvements Displacements and Remote Displacement can now be modified in a small restart calculation. In addition, initial velocities can be modified for parts in small restarts and full restarts.
  - Foam Material Support New Material Models have been added in support of foam applications
  - Support for Additional Contact Scoping Options
  - Additional Properties for Interference Contact
  - New Solver Version
- **Meshing for Explicit**
- **LS-DYNA Solver R13:**
  - Iso-geometric Analysis (IGA)
  - Materials and Element Enhancements
  - Element Free Galerkin Enhancements
  - Smoothed Particle Galerkin (SPG) Features
  - Smooth Particle Hydrodynamics Enhancements
  - XFEM, Peridynamics
  - Multi-scale Modeling
  - Contact
  - Stamping
  - Acoustics
  - Fatigue
  - Thermal
  - ALE
  - Electro-magnetics (EM)
  - Incompressible Computational Fluid Dynamics (ICFD)
  - Miscellaneous enhancements

Two-scale Co-simulation Approach in Electronics Applications

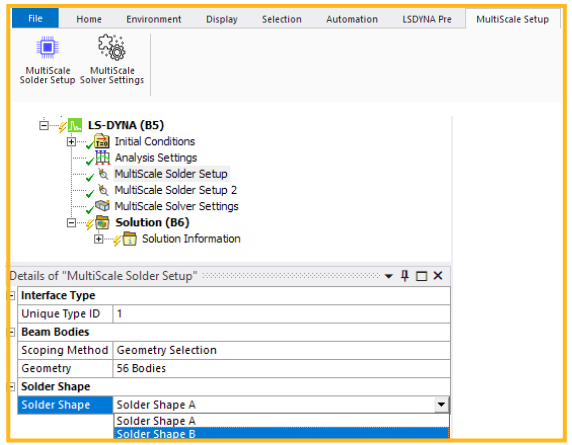
- New keywords: *INCLUDE_MULTISCALE, *DEFINE_MULTISCALE
  - The two-scale co-simulation couples the mesoscale model and the macroscale model using the non-matching discretization to co-simulate the structural response.
  - The beam element can be replaced by solid elements for the solder ball modeling.
  - This solder joint model will be modularized, duplicable, and numerically immersed in a meso-scale chip model automatically.
- ACT on Ansys Workbench (work with LST ACE team)



Automated process of replication solder joint



Trace mapped shell PCB with detailed solder joints model in drop test

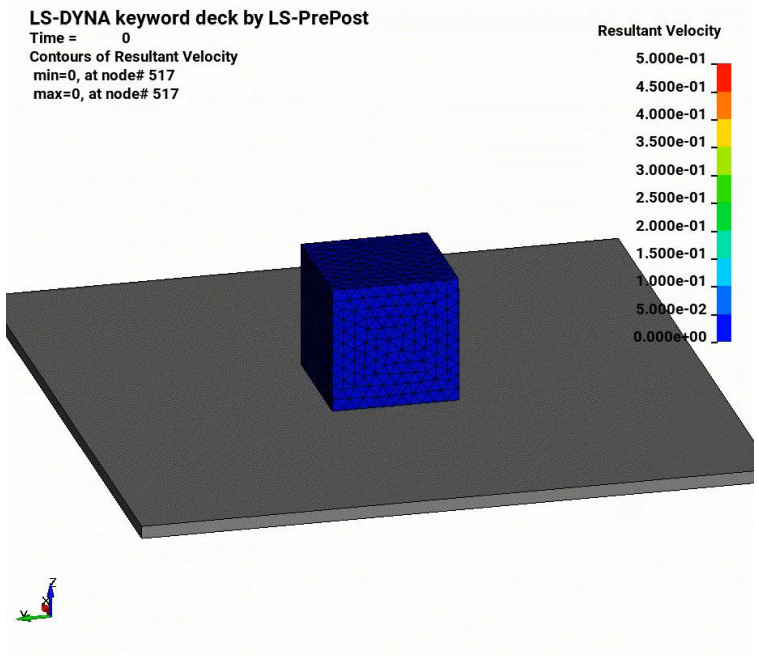


ACT on Ansys Workbench

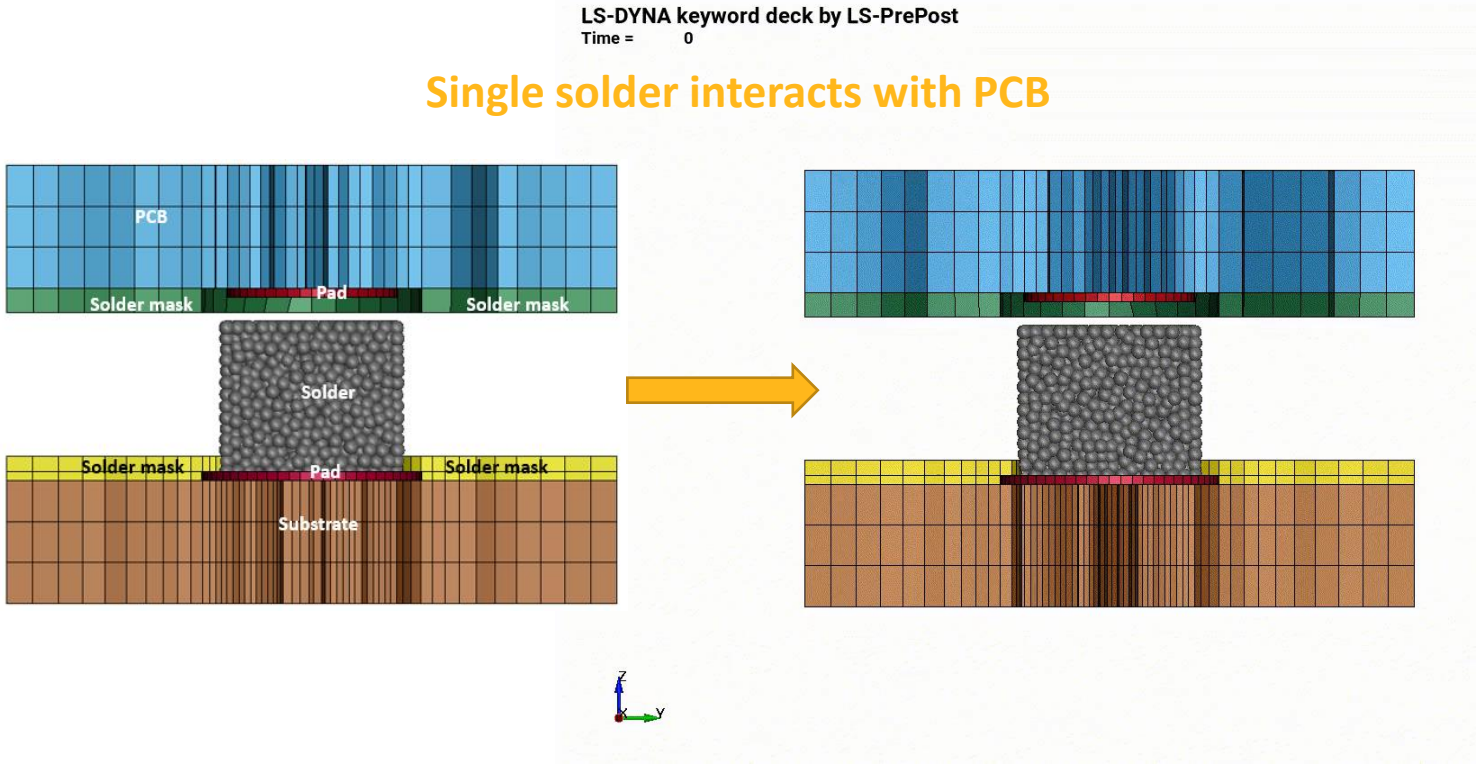
Fully Implicit Incompressible SPG (ISPG) Formulation

- Fully implicit ISPG formulation
  - A new Lagrangian Navier-Stoke solver
  - Can handle the surface tension and wall adhesion accurately and efficiently
  - Can simulate the solder reflow with complex models with the solder mask defined (SMD) pad and NSMD

Large deformation of droplet



Single solder interacts with PCB

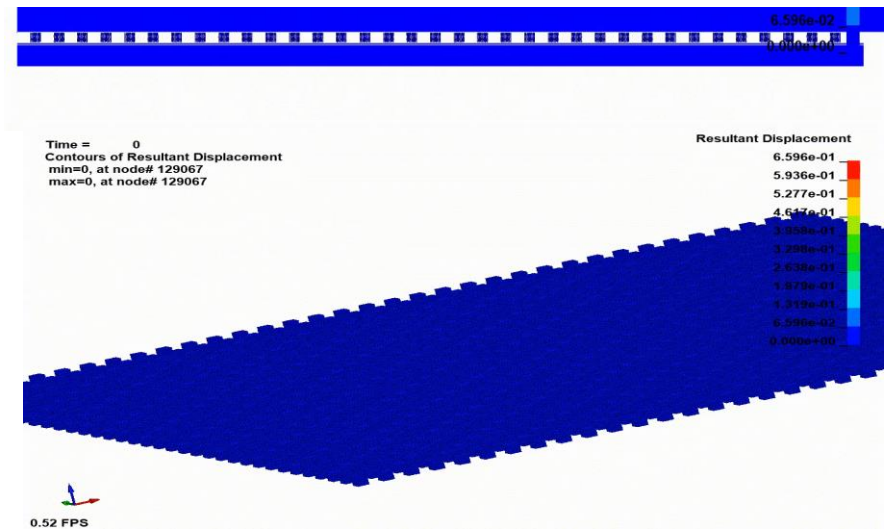


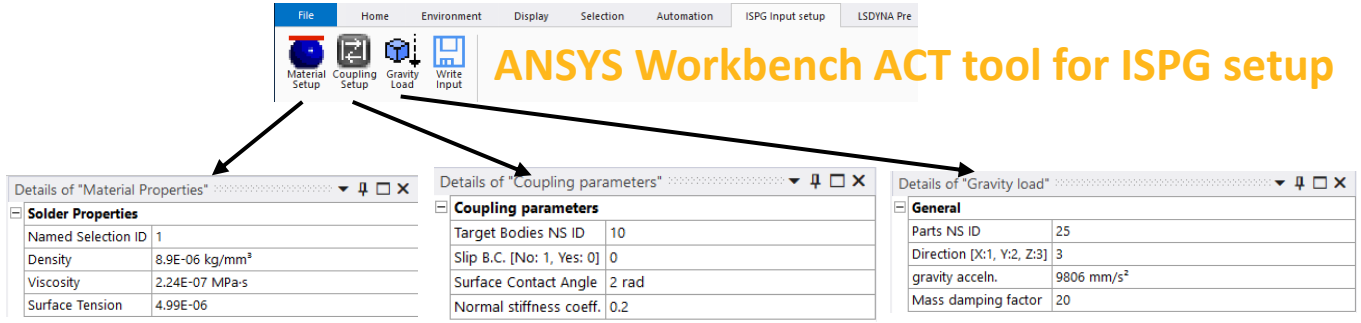
Fully Implicit ISPG Formulation

- Coupled with implicit thermal and structure solvers
  - for large scale thermal-mechanical PCB and solder reflow analysis (considering PCB warpage effect; SMP&MPP)

- ACT on Ansys Workbench available
  - Capable to define all necessary keywords

1225 solders (1.02M nodes) on PCB solved with MPP solver on 64 CPU cores (Run time – 7.5hrs)



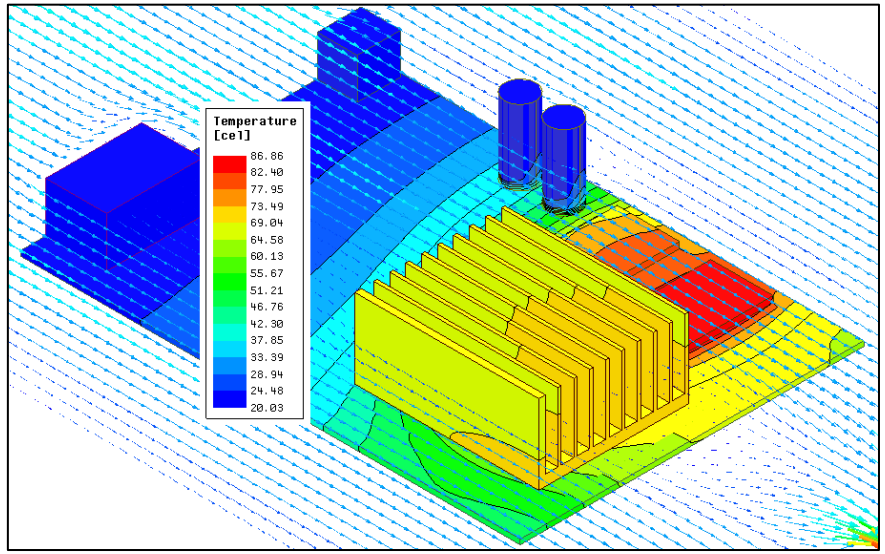


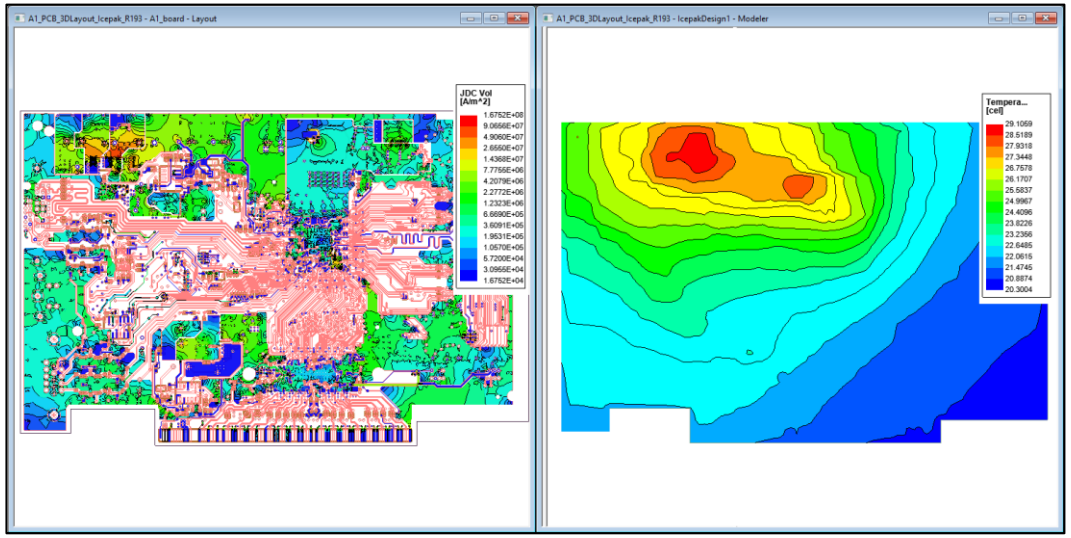
Ansys Icepak

Ansys Icepak 2022 R1 Highlight Summary

Please refer to the Ansys Icepak 2022 R1 Update Presentation for additional details:

- **Reduced Order Modeling (ROM)**
  - Redhawk CTM 2-Way Co-Simulation
  - Delphi Network Creation
- **Advanced Modeling**
  - Blower Model
- **ECAD Import**
  - Wirebond import
  - IDX Import
- **Maxwell 2D – Icepak EM Loss Coupling**
- **Mesher Enhancements**
  - 2.5D Meshing improvements
- **User Experience**
  - Streamlines
  - Validation Enhancements
- **Migration**
  - Improve speed of TZR conversion
  - Network Schematic enhancements
  - Toolkit enhancements
    - PCB, Package parameterization

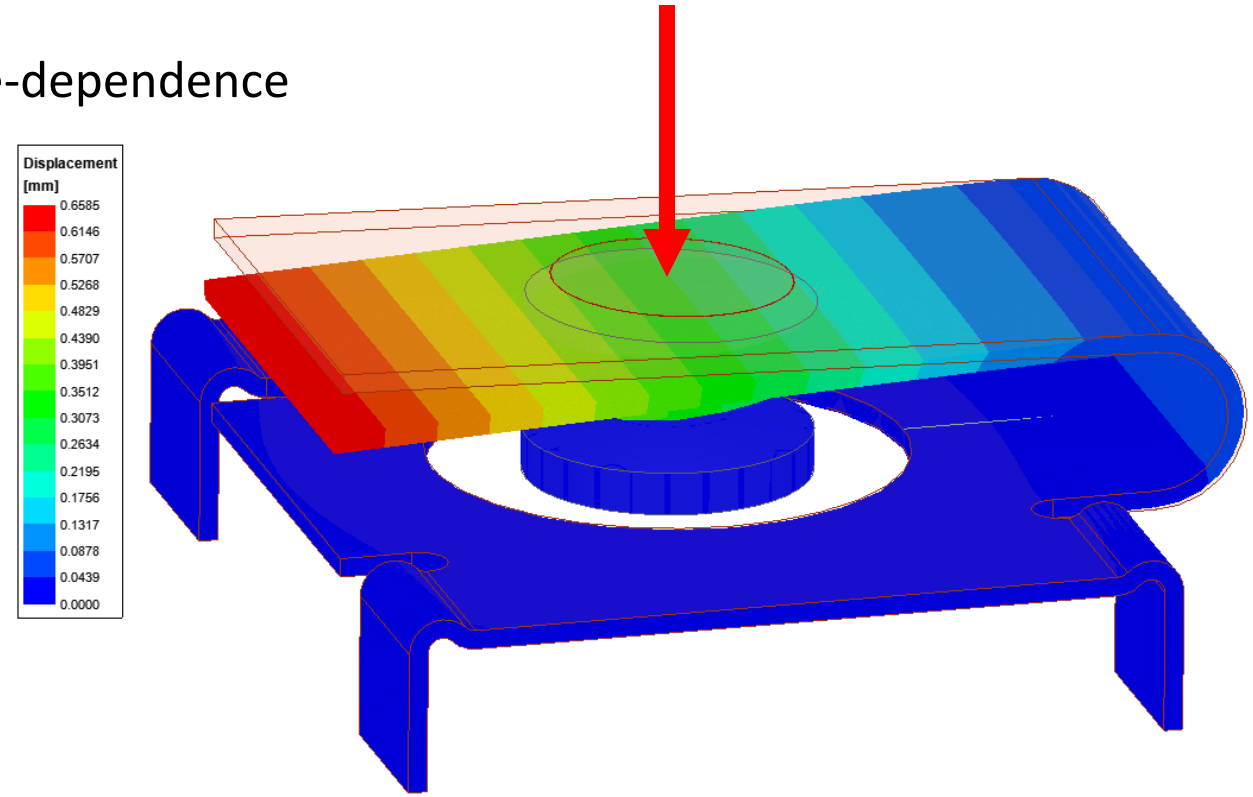




Ansys AEDT Mechanical

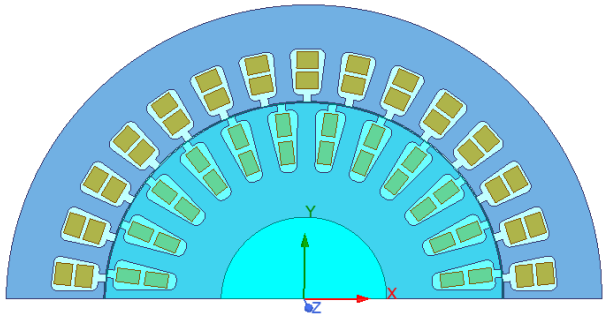
AEDT Mechanical 2022 R1 Highlights

- Coupling
  - Maxwell 2D – Thermal EM Loss Coupling
- Materials
  - General expression support for temperature-dependence
- Structural - Beta
  - Boundaries
    - Displacement
    - Pressure/Force
  - Coupling
    - Mechanical Thermal-Structural Link
    - EM Force – Structural Coupling
- Meshing - Beta
  - Thermal Slider bar Meshing
- Reporting
  - Fields Summary

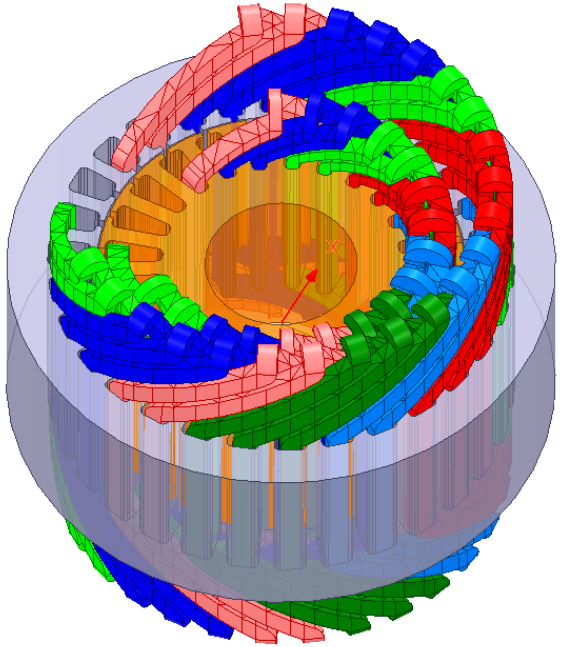


Coupling: Maxwell 2D – Thermal EM Loss Coupling

- Support EM Loss Import from Maxwell 2D
  - Extruded geometries of 2D representations
    - Support both +ve and –ve extrusions in XY
    - Can be partial geometries
  - Coupling projects 3D mesh points onto 2D geometry
  - Limitations
    - Extrusions need to be along Z axis
    - Losses not conservative
    - 2-way coupling not supported

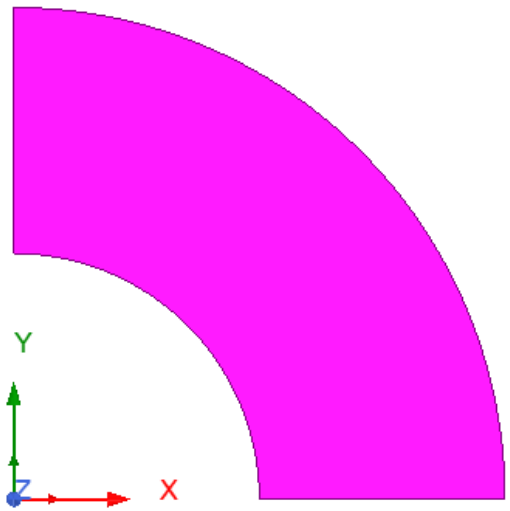


Maxwell 2D Geometry

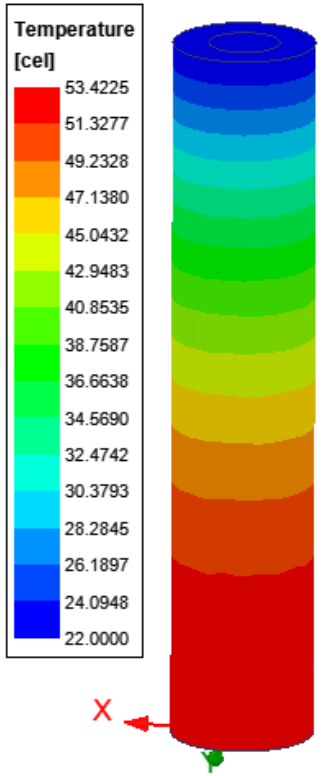


Thermal 3D Geometry

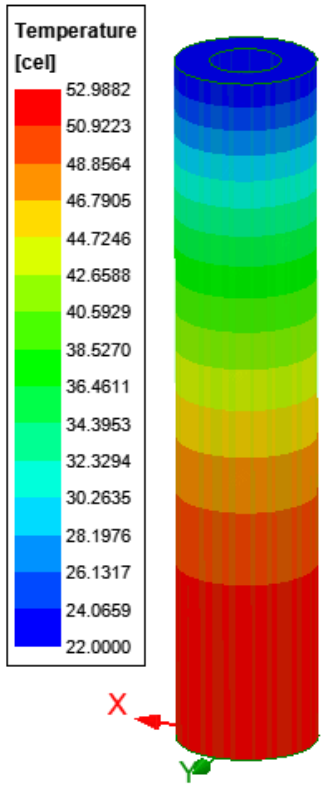
Coupling: Maxwell 2D – Thermal EM Loss Coupling



Maxwell 2D Geometry



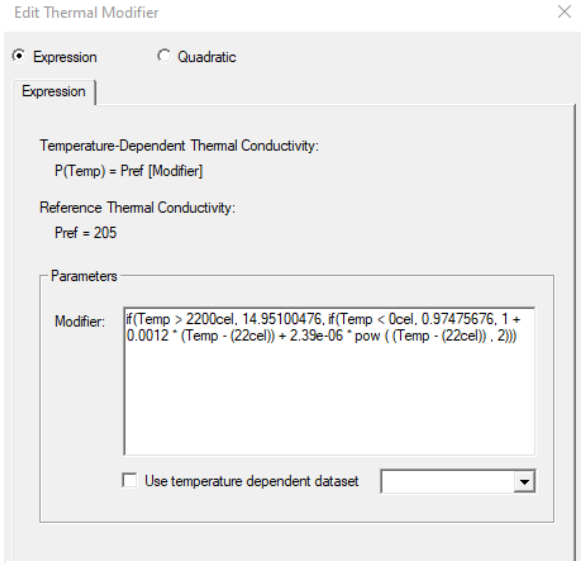
Maxwell 3D Coupling

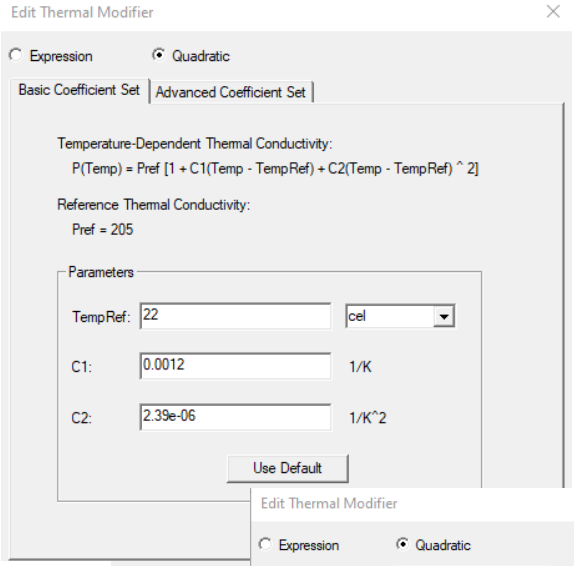


Maxwell 2D Coupling

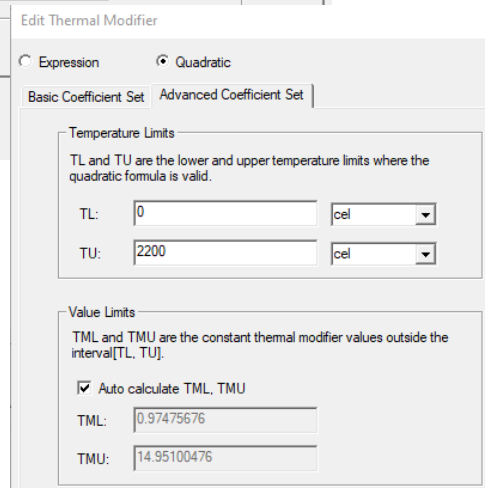
Materials: Temperature-Dependent Materials

- Temperature Dependent Materials
  - General expression support
  - Quadratic expressions
    - Advanced coefficient support
  - Converted to datasets for solver
  - Thermal & Structural



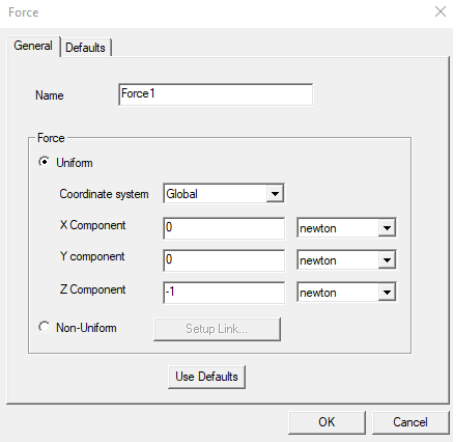


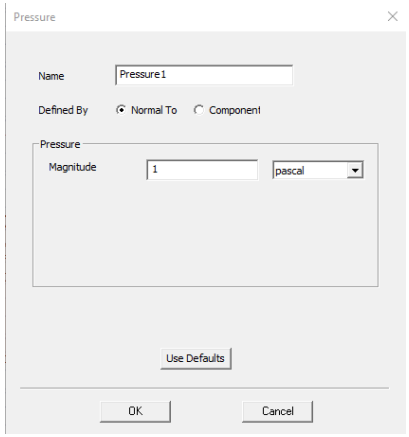

```
MPTEMP,1,0
MPTEMP,2,20
MPTEMP,3,40
MPTEMP,4,60
MPTEMP,5,80
MPTEMP,6,100
MPTEMP,7,120
MPTEMP,8,140
MPTEMP,9,160
MPTEMP,10,180
MPTEMP,11,200
MPTEMP,12,300
MPTEMP,13,400
MPTEMP,14,500
MPTEMP,15,600
MPTEMP,16,700
MPTEMP,17,800
MPTEMP,18,900
MPTEMP,19,1000
MPDATA,KXX,1, ,199.8251358,204.5099598,209.5867438,215.0554878,220.9161918,227.1688558,
MPDATA,KXX,1, ,233.8134798,240.8500638,248.2786078,256.0991118,264.3115758,311.2532958,
MPDATA,KXX,1, ,367.9940158,434.5337358,510.8724558,597.0101758,692.9468958,798.6826158,
MPDATA,KXX,1, ,914.2173358, ! W m^-1 C^-1
MPTEMP,,,,,,,,
```

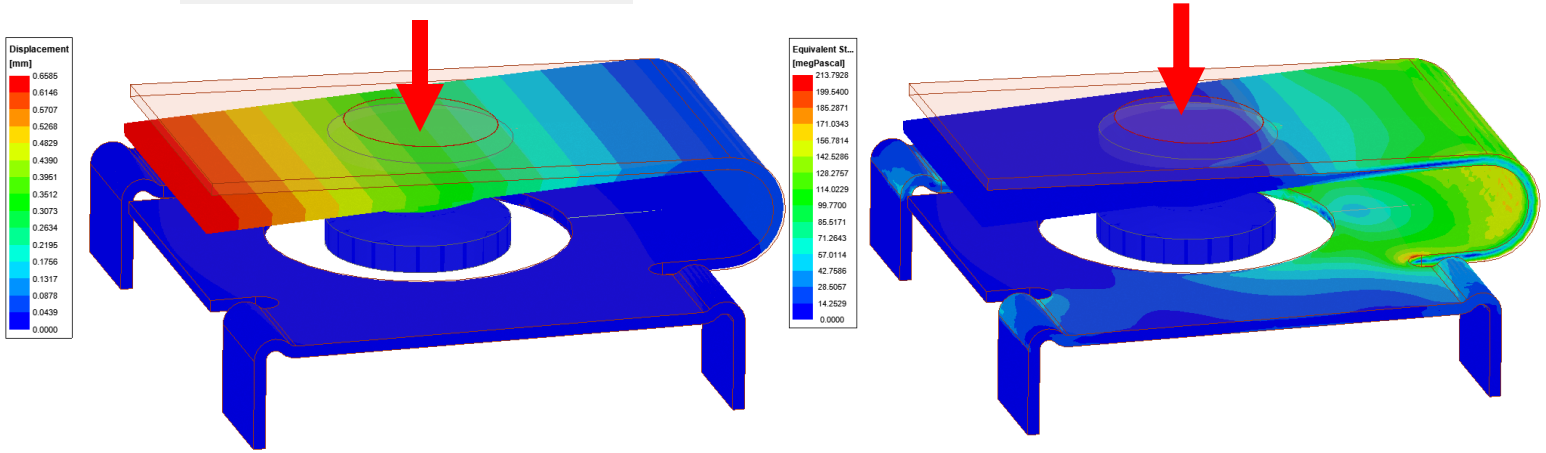


Structural: Pressure/Force Excitations

- Force Excitation
  - Face and Object assignment
  - Uniform and Non-uniform Force options
    - Uniform (face): X, Y, Z components
    - Non-uniform via Setup Link to HFSS/Maxwell
- Pressure Excitation
  - Face assignment
  - Normal To or Component options
    - Normal To: Magnitude
    - Component: X, Y, Z components
  - Support curved faces



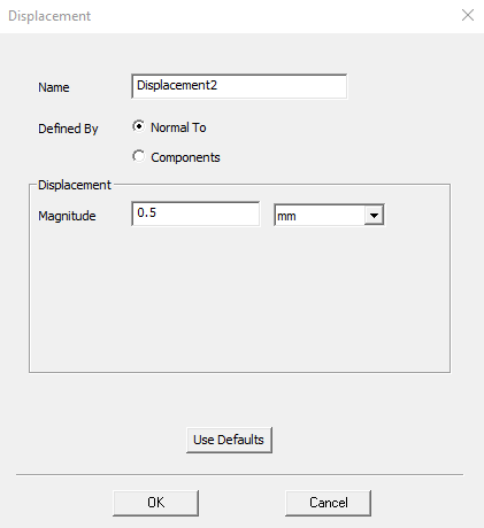


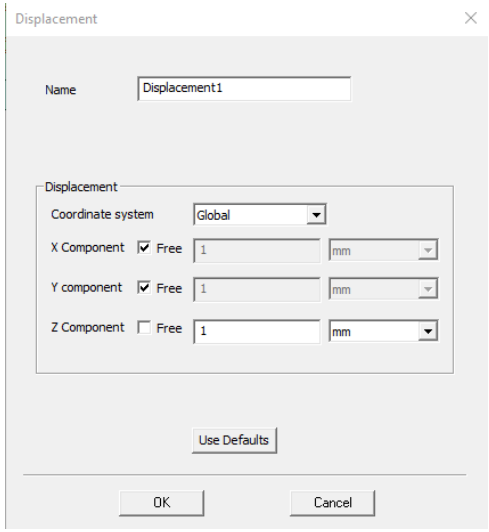


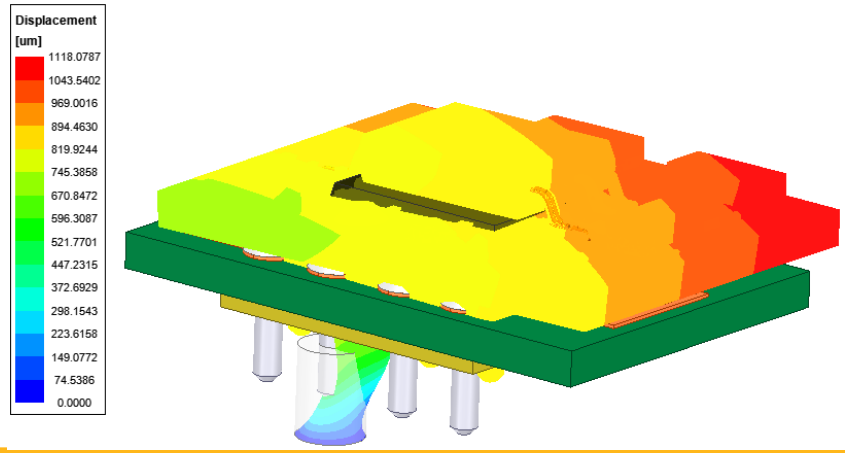
Assembly under 1N vertical force

Structural: Displacement Excitation

- Displacement Excitation
  - Assignment: Faces and Edges
  - Normal To (faces)
    - Magnitude
  - Components (faces and edges)
    - X, Y, Z components
    - Each component can be fixed magnitude or free



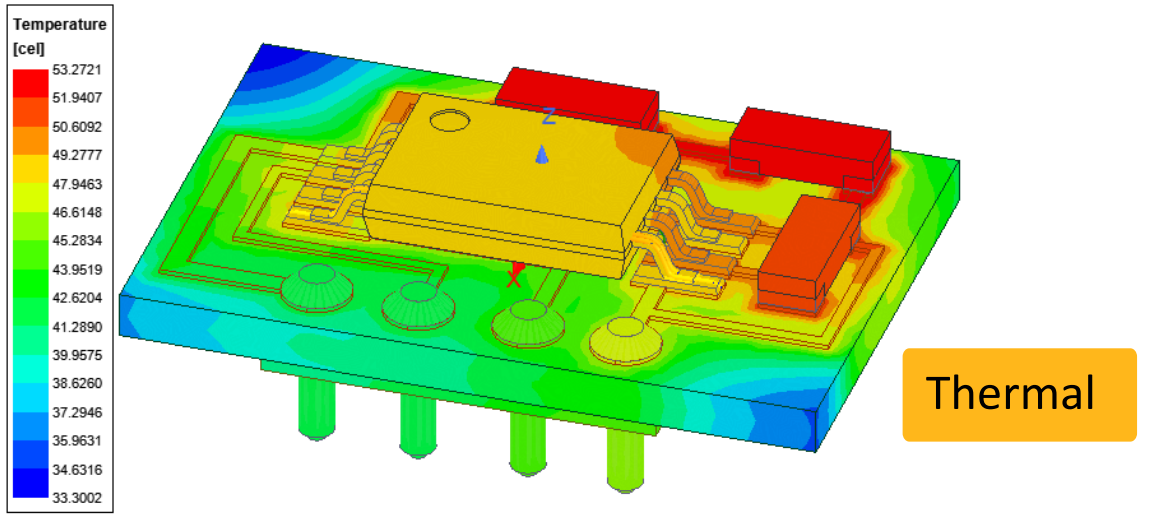




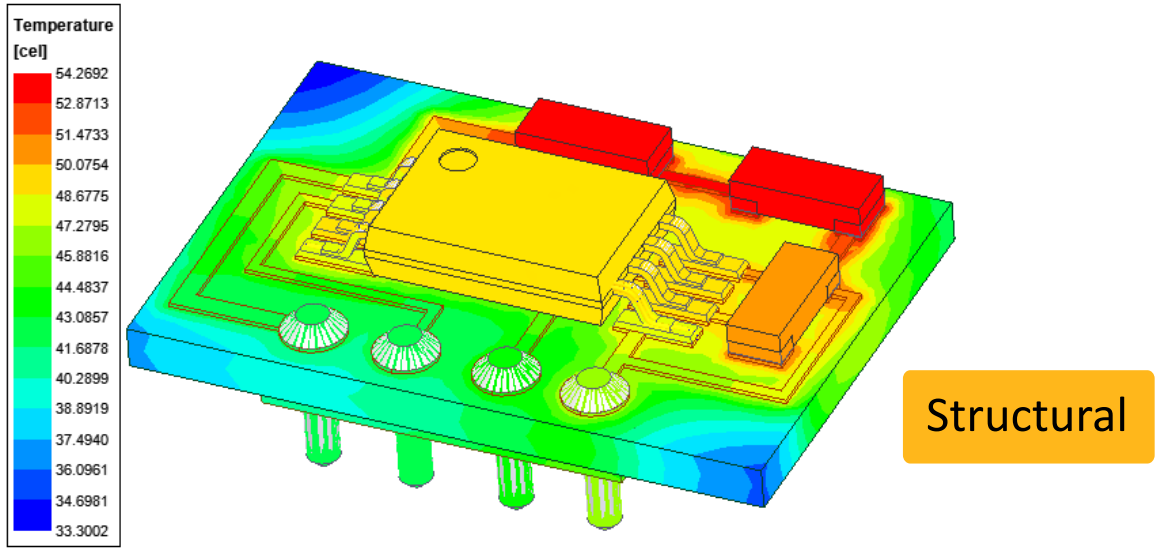
PCB Assembly with Y, Z displacements along edge

Thermal Stress Analysis: Link to Mechanical Thermal

- Coupled Thermal Stress Analysis
  - Linked to Thermal design
    - Thermal condition excitation
  - Temperatures imported for objects
    - System Coupling mapper
    - Radial Basis Functions (RBF) Algorithm
  - Temperature field plots



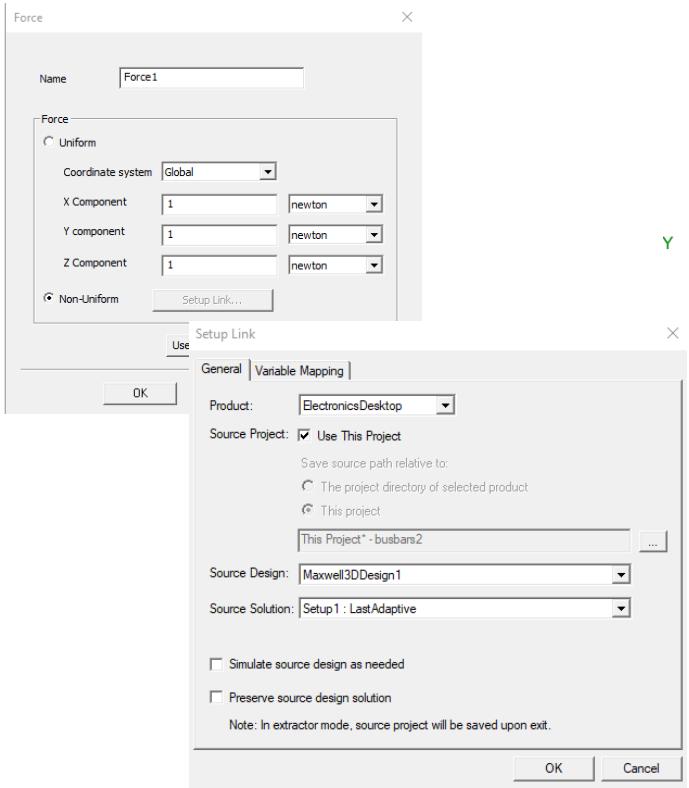
Thermal

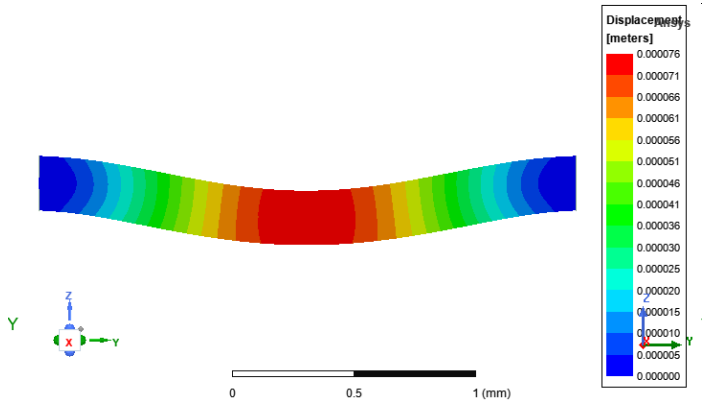


Structural

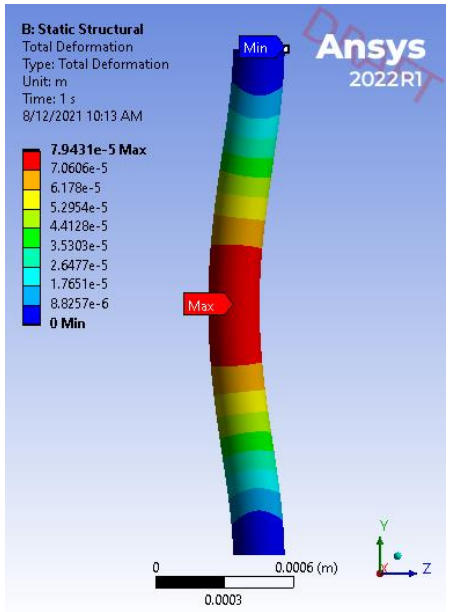
EM – Structural Coupling: EM Force

- Coupled EM Force - Structural Analysis
  - Linked to HFSS
    - Surface assignment
  - Linked to Maxwell 3D
    - Surface and Volume assignment
  - Assignment: Faces and Objects
  - 1-way coupling support





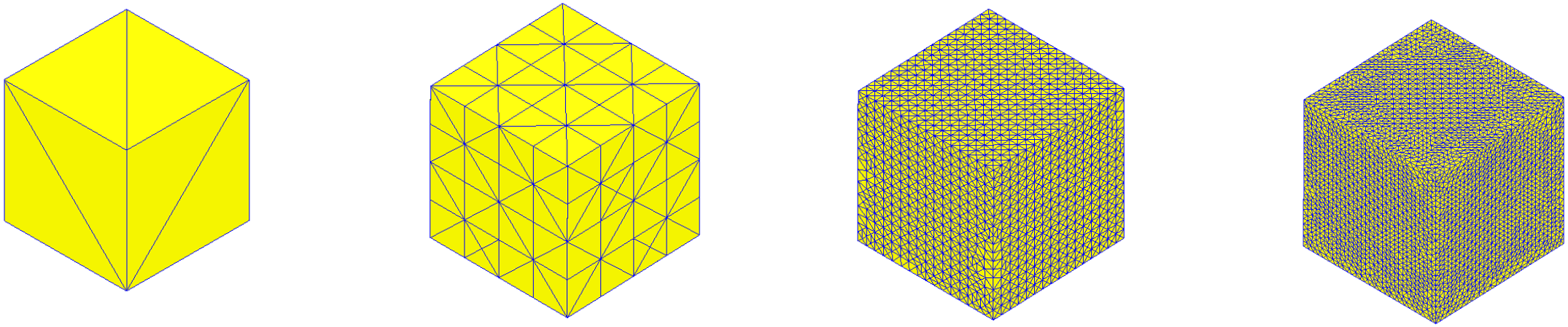
AEDT



Workbench

Thermal Slider Meshing: Automated Refinement (Beta)

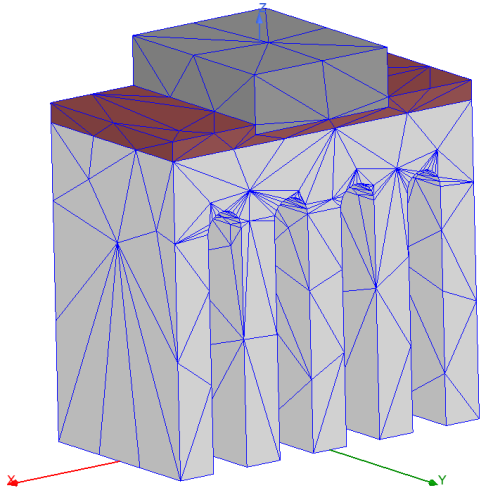
- Automated refinement based on slider position
  - Length-based refinement inside and on surfaces of all objects
  - Refinement tailored to curvilinear and rectilinear geometries
- Restrict the need for user-defined mesh operations
- Improved solution accuracy



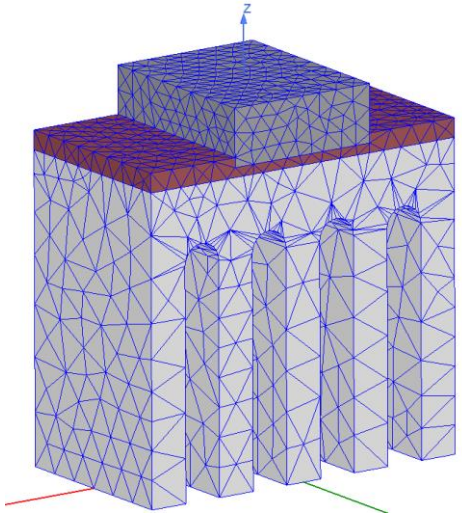
2021R2 (All slider positions)

2022R1 Auto refinement

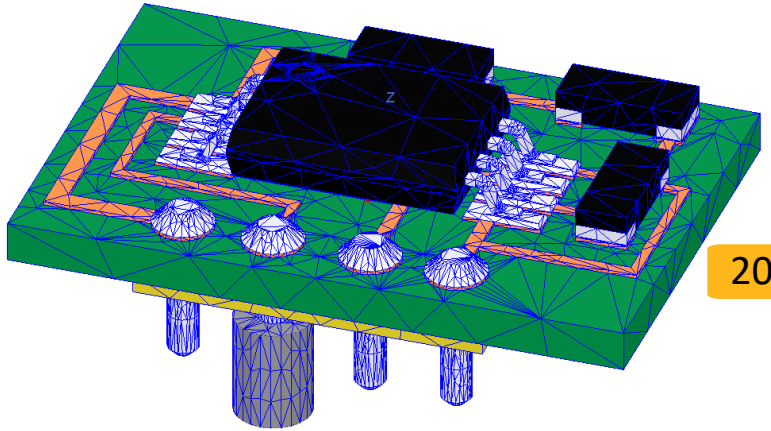
Thermal Slider Meshing: Comparison



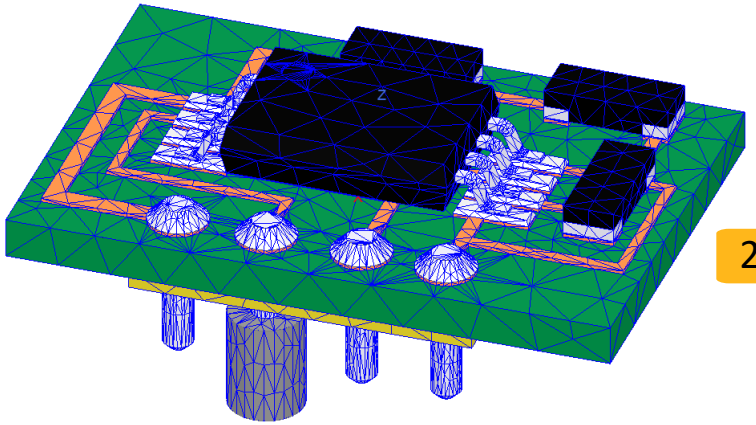
2021R2 (Count: 2k)



2022R1 (Count 11k)



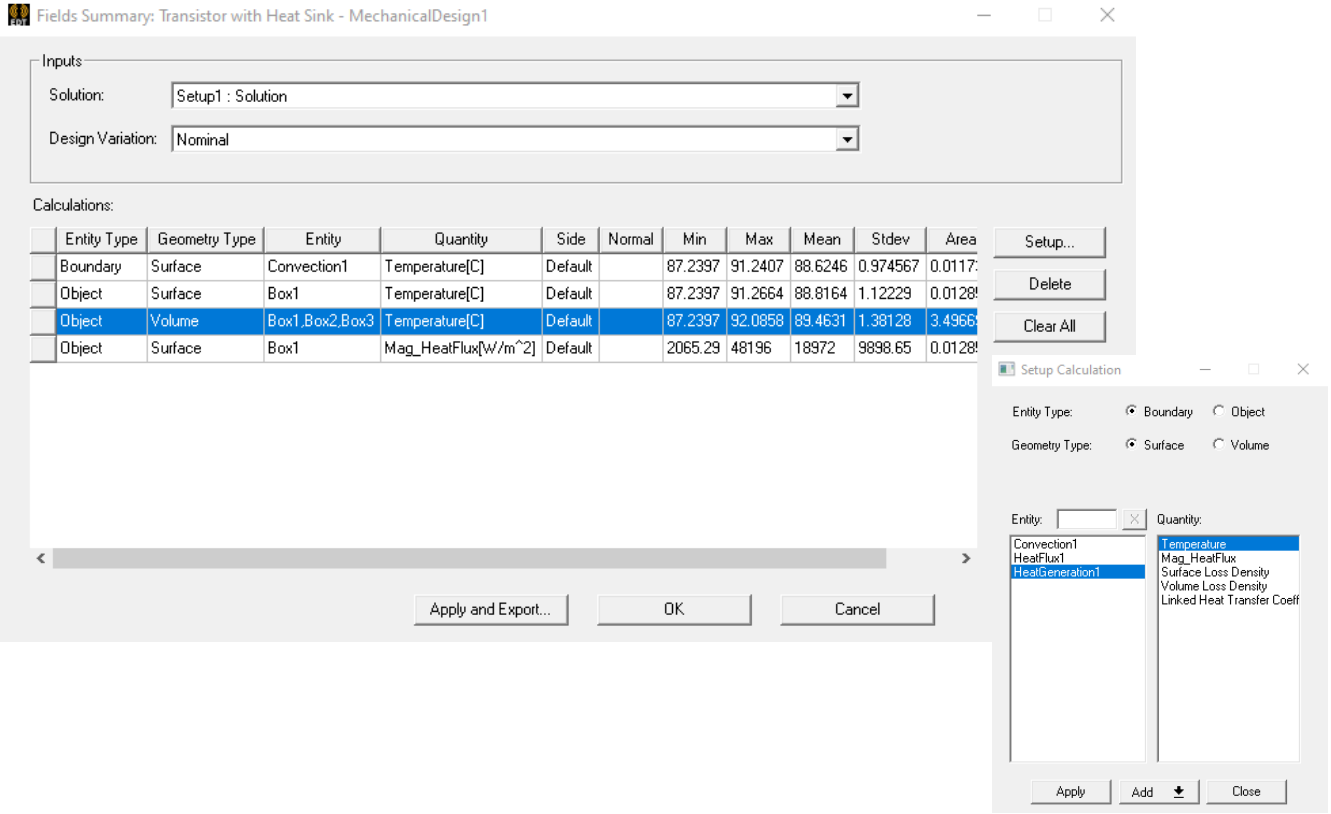
2021R2 (Count: 13k)



2022R1 (Count 14k)

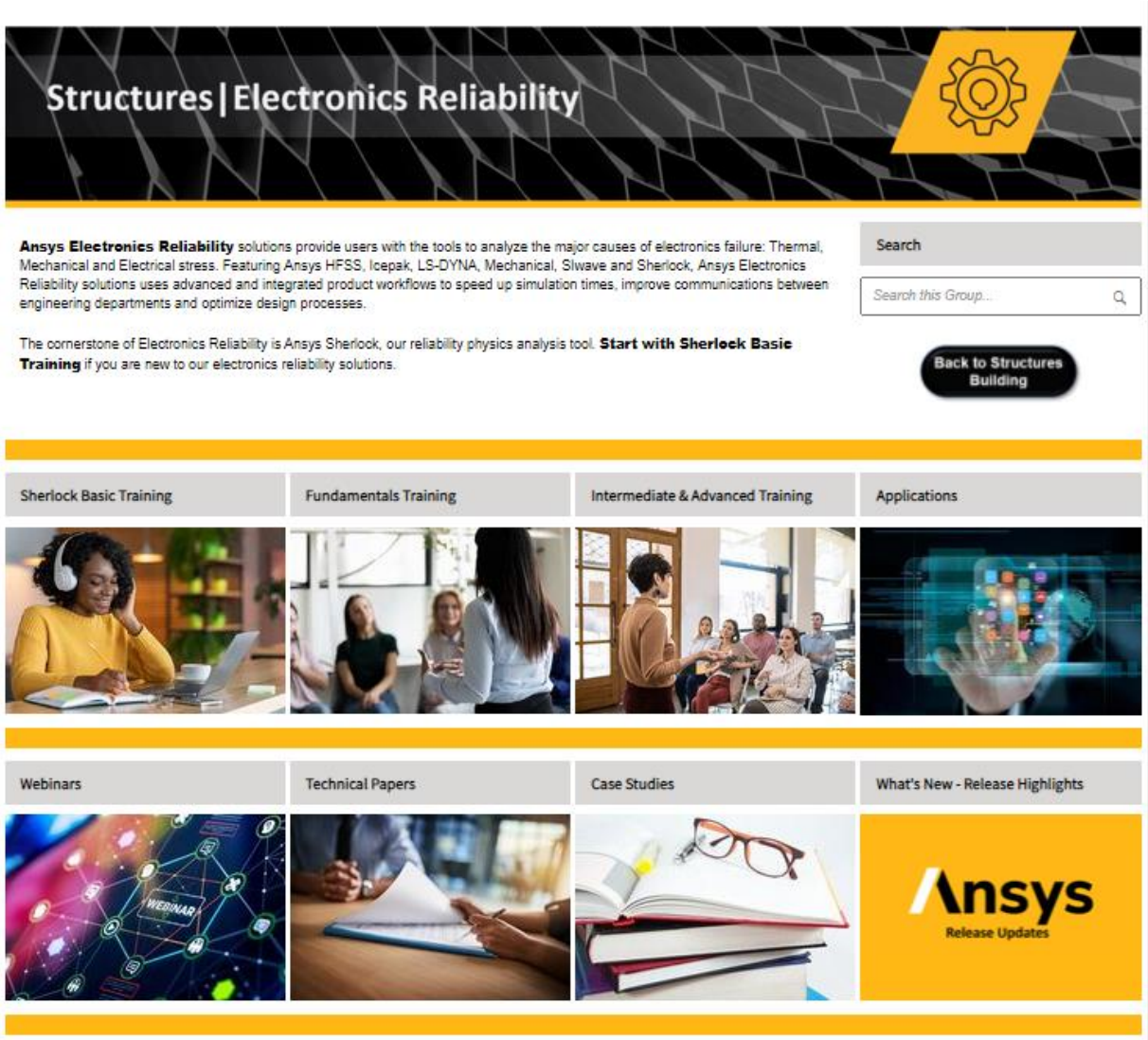
Reporting: Fields Summary

- Fields Summary: User-friendly report calculation capability
  - Supports all Fields Calculator variables
  - Boundary and Object selection
  - Surface and Volume calculations
  - Min, Max, Mean, Standard Deviation, Total
  - Multi-select and multiple calculations
  - Export to CSV format
  - Heat Flow Rate not available yet



Additional Resources

Ansys Learning Hub (ALH) Electronics Reliability Learning Room



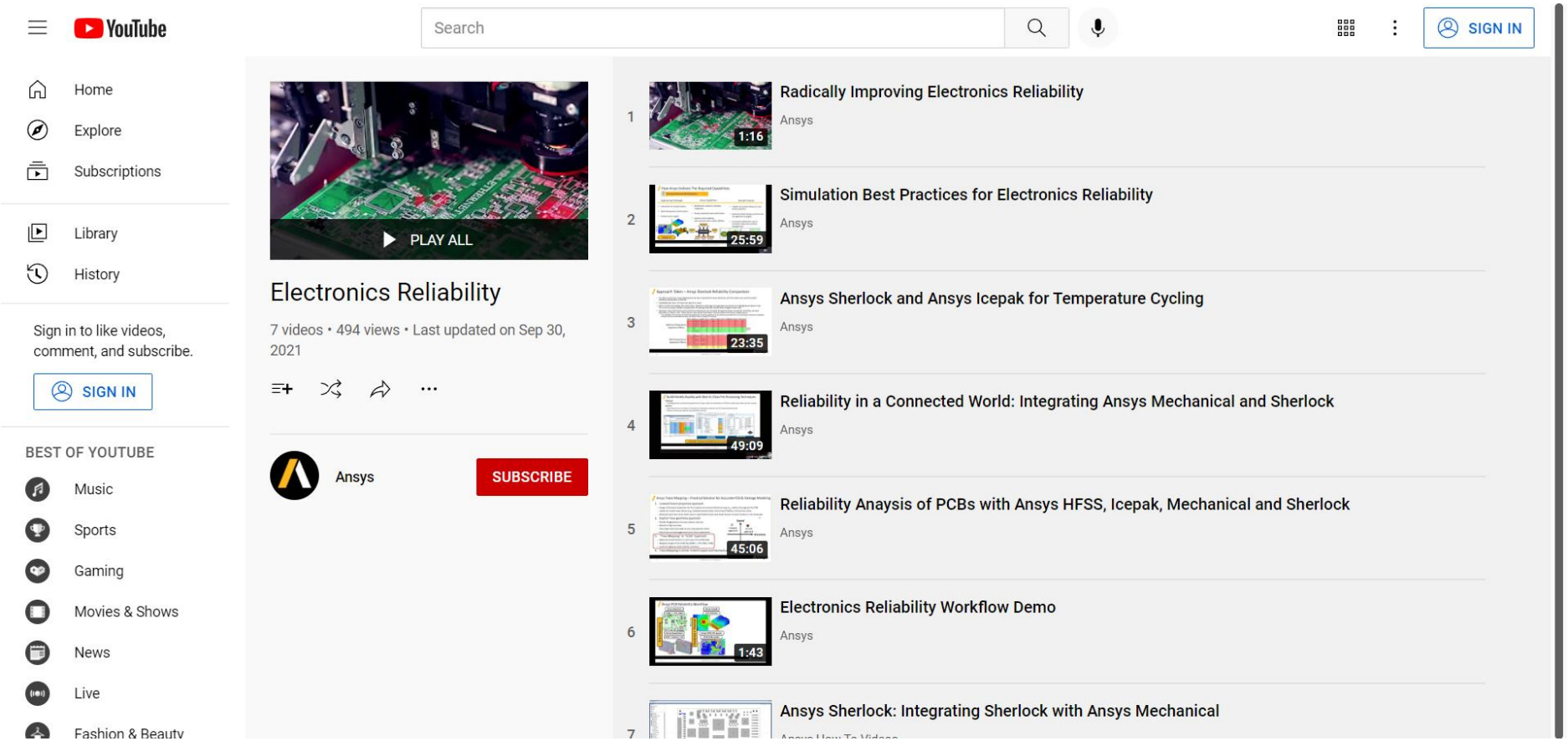
- New and **improved user experience**
- **3-pronged learning paths** including Basic, Fundamentals, Intermediate & Advanced Training
- Video Walk-throughs, on-demand webinars, **technical papers, and more**
- Ask questions directly to **Ansys experts**

Login here:
https://catalog.ansys.com/ALH.cshtml

Direct Link (ALH Access Required):

https://jam8.sapjam.com/groups/QxhZIS5hvjR1EWlg4pCOD2/overview_page/owCBFHDqvFQ01u7FsvRRcx

Ansys Electronics Reliability YouTube Page



https://www.youtube.com/playlist?list=PLQMtm0_chcLzeB8zCeGmvBkFpT3oG7kFN

Ansys