



*ANSYS solutions to  
Lead Free Package  
Design Challenges*

**Kamal Karimanal  
Sheldon Imaoka  
Vamsi Krishna**

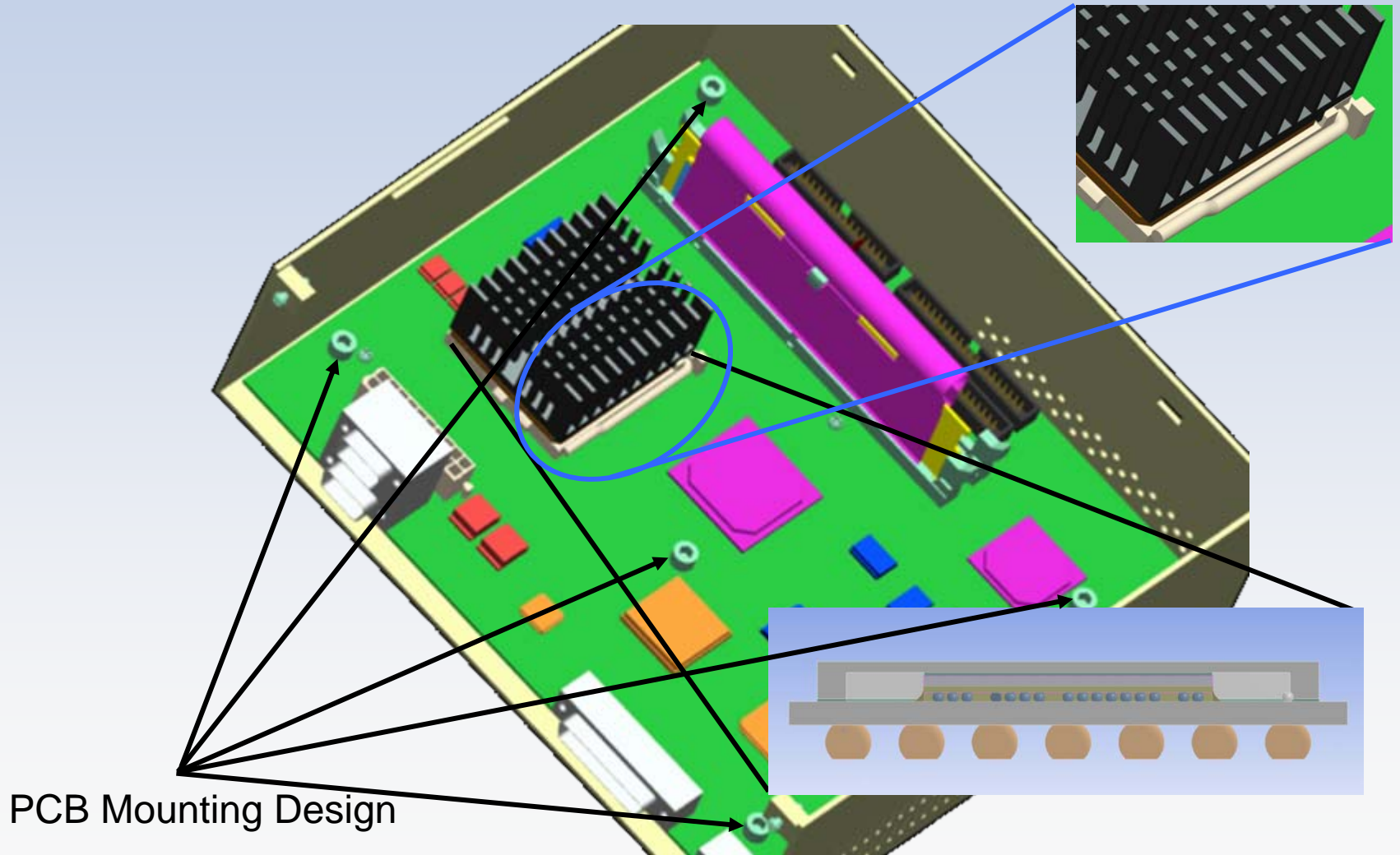


*Today's Topic*

*ANSYS Solutions to  
Electronics System Design  
Challenges*

- **Shipping and handling exposes electronics to shock and drop events**
  - Can result in catastrophic failure even from a single event
- **Even normal, recommended end use conditions subject electronics to temperature fluctuations due to power on and off**
  - Thermo mechanical stresses incrementally damage components with every cycle
  - Failure is not about “if”, its about “When”
    - Need for life prediction capabilities

# Design Implication of the Lead Free Change

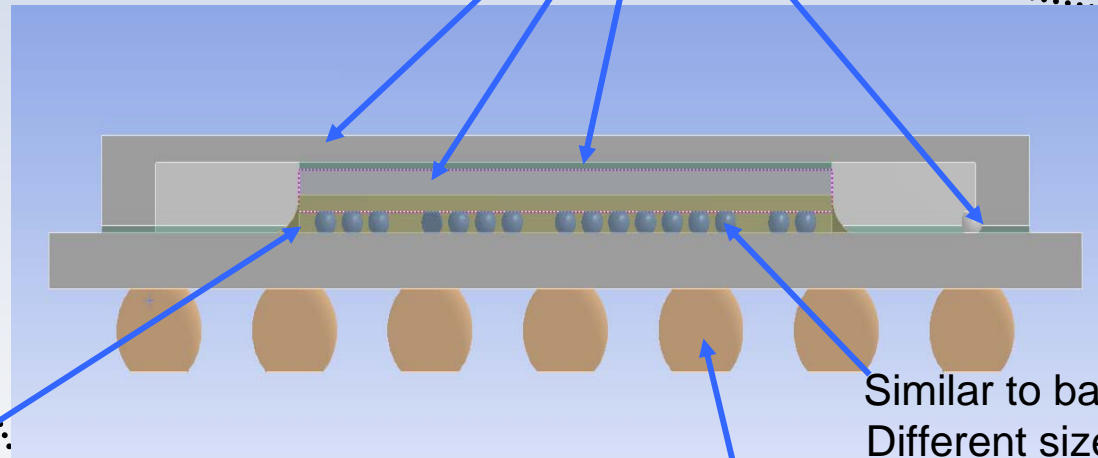
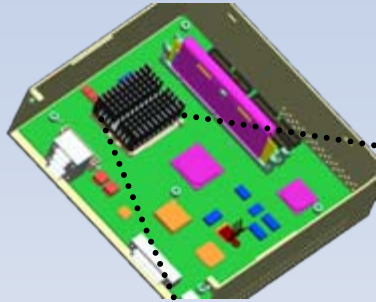


PCB Mounting Design

# Design Implication of the Lead Free Change at the Package Level



Silicon, IHS, Attach Material selection & Design are completely coupled. Larger Die = more functionality as well as reliability issues



Underfill -  
Material & thickness are coupled with process as well as product designs

Similar to balls but entirely Different size scale issues. Coupled with underfill delam.

Solder Balls – size, pitch, material PCB & Substrate pad config, mask design criteria

# ANSYS Technologies for Lead Free Challenges at the Package Level

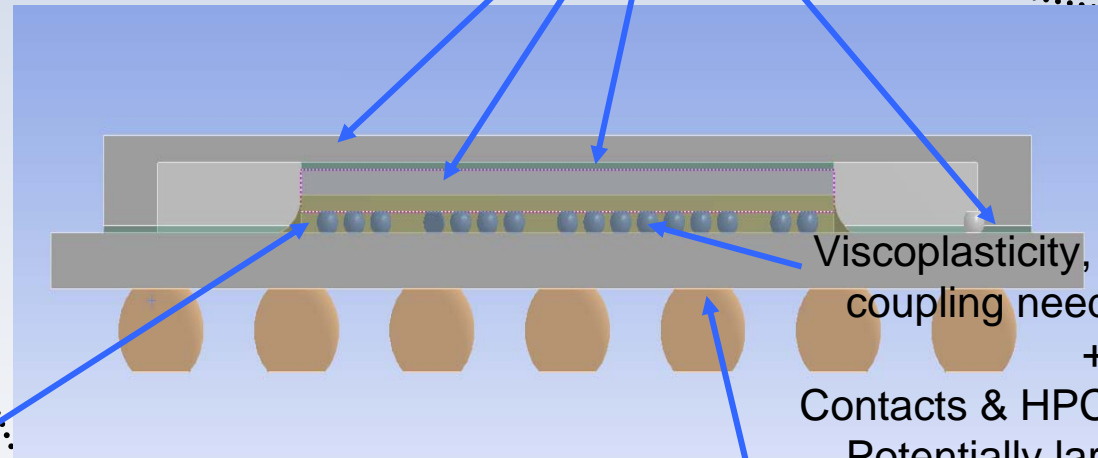
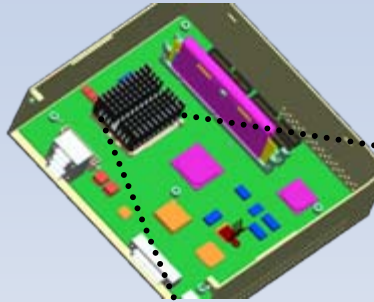


CZM in ANSYS Mechanical

+

Implicit as well as explicit solver

Technologies for shock & drop reliability



Viscoplasticity, creep, physics coupling needs & scripting

+

Contacts & HPC technology for Potentially large models in ANSYS Mechanical

Underfill -

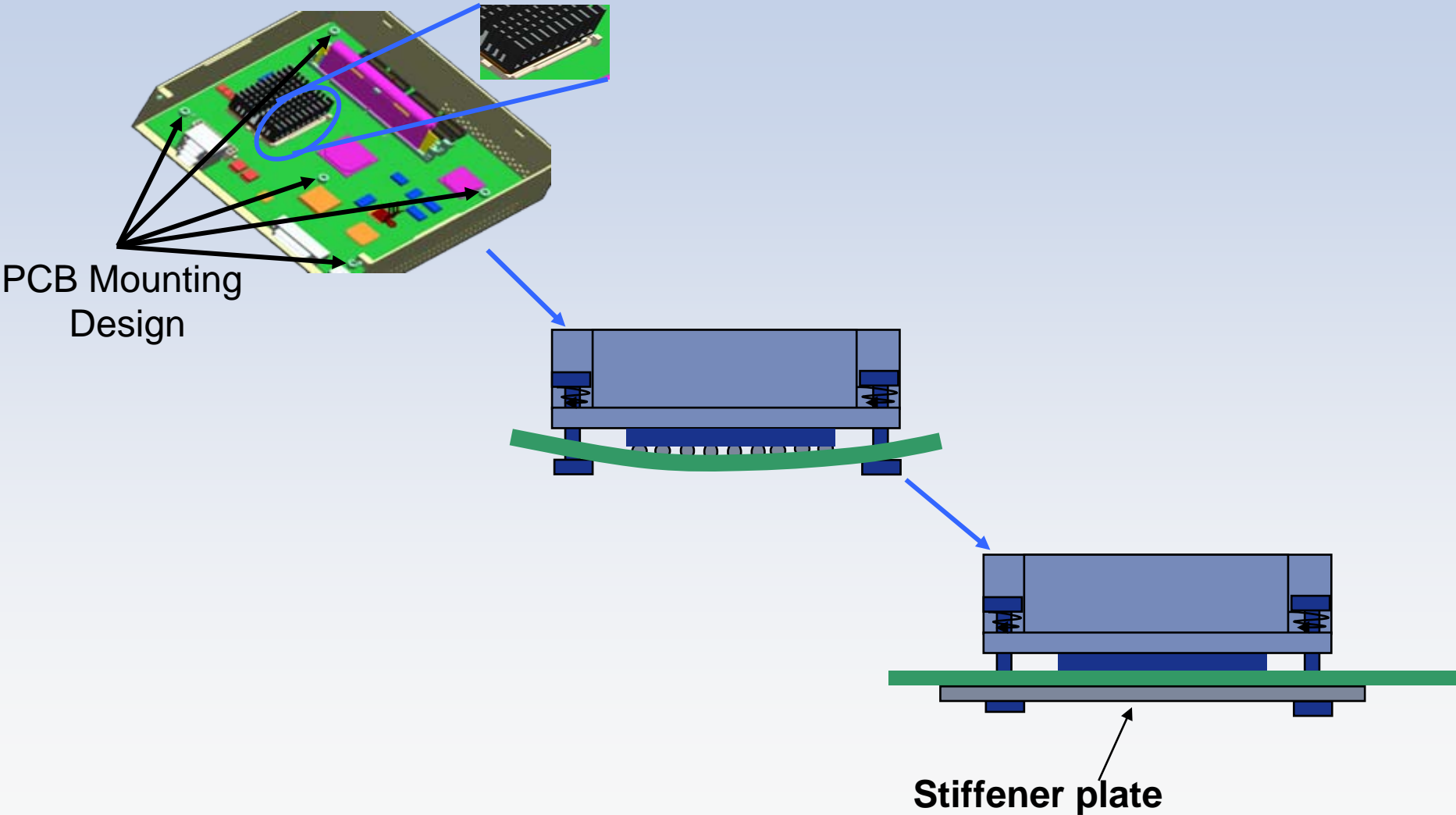
Visco elasticity and Cohesive Zone Models in ANSYS Mechanical for product design.

+

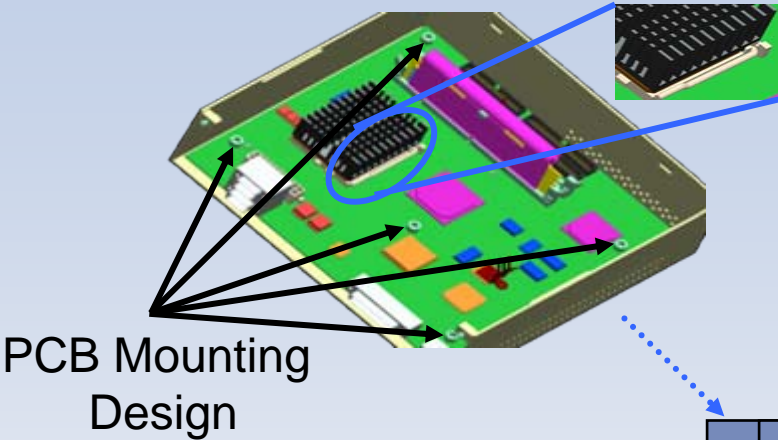
VOF modeling capabilities in ANSYS CFD to model Capillary driven underfill flow for process design

Solder Balls – non-linear Visco Plasticity & Creep models, thermo-mechanical coupling, If necessary for power cycling analysis. Scripting Capabilities to aid automatic estimation of Fatigue Life in ANSYS Mechanical

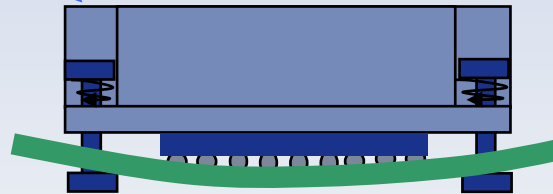
# Design Implication of the Lead Free Change at the System Design Stage



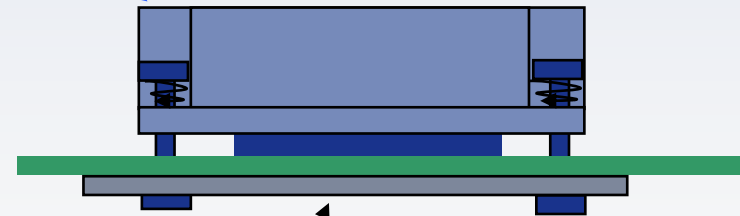
# ANSYS Technologies for the Lead Free Challenges at the System Design Stage



Linear/non-linear flexible dynamics using implicit as well as explicit solver technologies in the WB environment for shock & drop reliability



Linear Static Structural, Modal and Random Vibration solver technologies in ANSYS Mechanical



Stiffener plate

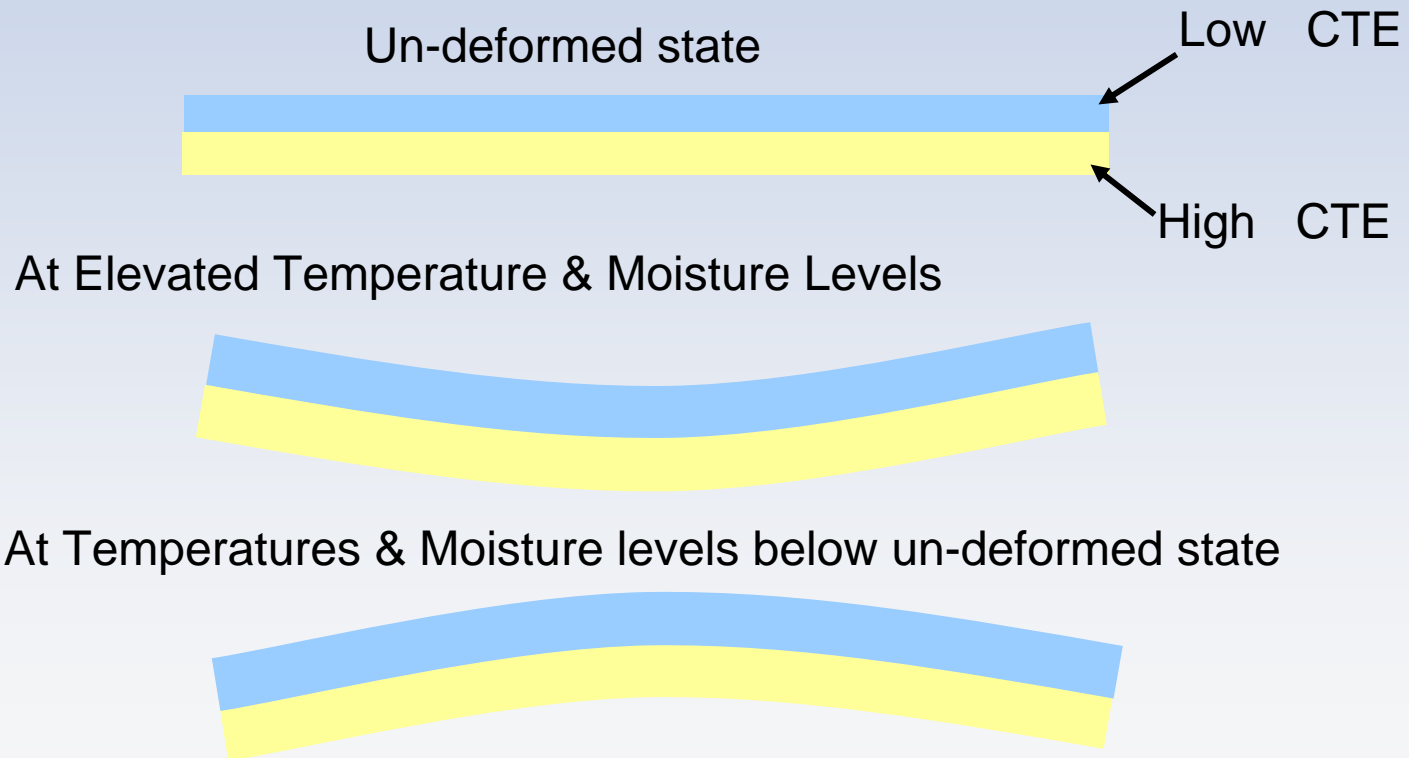
The background features a central globe of the Earth, rendered in shades of blue and white. The globe is surrounded by a complex network of glowing, ethereal lines in blue and orange, resembling magnetic field lines or energy trails. The ANSYS logo is prominently displayed in the center of the globe.

*Today's Focus:*

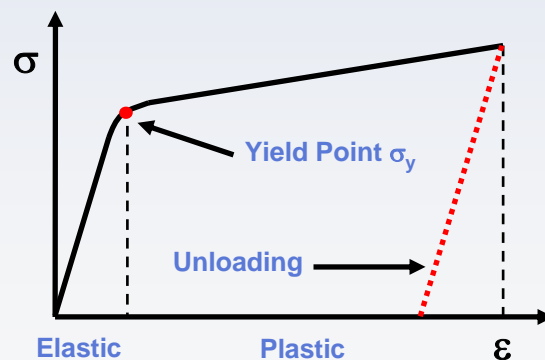
*Solder Joint Reliability  
Under Thermal Cycling  
Fatigue*

**ANSYS®**

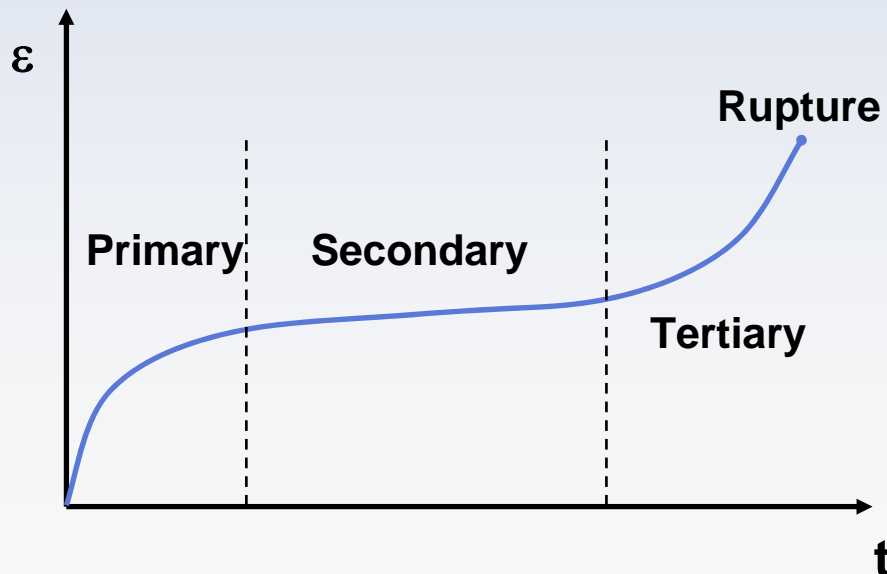
$$\frac{dV}{dT} = \alpha V, \quad \alpha - CTE$$



- **Plasticity models describe inelastic behavior that develops when the stress level goes beyond the yield strength of the material**
  - Can be rate-independent or rate-dependent or coupled with creep



- **Materials tend to undergo creep at temperature higher than their Homologous temperature**
  - Homologous temperature = 0.5 X Melting point (in Kelvin)
- **Secondary creep has been known to cause Failure of Solder Joints**



$$\dot{\epsilon}_{cr} = f_1(\sigma)f_2(\epsilon)f_3(t)f_4(T)$$

- No Permanent damage for perfectly elastic materials
- Among the IC package parts that undergo plastic deformation and/or creep, solder joints are the most susceptible to failure
- For Most Consumer electronics, thermal cycling is the main cause for fatigue failure

- **For solder joint reliability, prediction of life in ANSYS is based on two Choices:**
  - Choice of fatigue model
  - Choice of constitutive model
    - This is a material model that to be used in FEA that captures

$$\dot{\varepsilon}_{cr} = f_1(\sigma)f_2(\varepsilon)f_3(t)f_4(T)$$

# Fatigue model chosen for this work



- Following taken from R. Darveaux, “Effect of Simulation Methodology on Solder Joint Crack Growth Correlation”, ECTC 2000

- Crack Initiation: 
$$N_0 = K_1 \Delta W_{avg}^{K_2}$$

- Crack growth: 
$$\frac{da}{dN} = K_3 \Delta W_{avg}^{K_4}$$

- Characteristic life: 
$$\alpha_w = N_0 + \frac{a}{\frac{da}{dN}}$$

Here,  $K_1$  through  $K_4$  are parameters estimated by correlating  $\Delta W_{avg}$  from simulation predictions to test data of  $N_0$  and crack growth rate

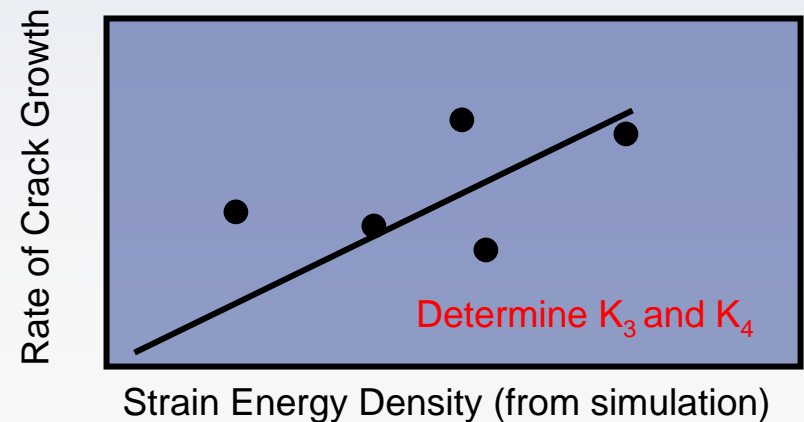
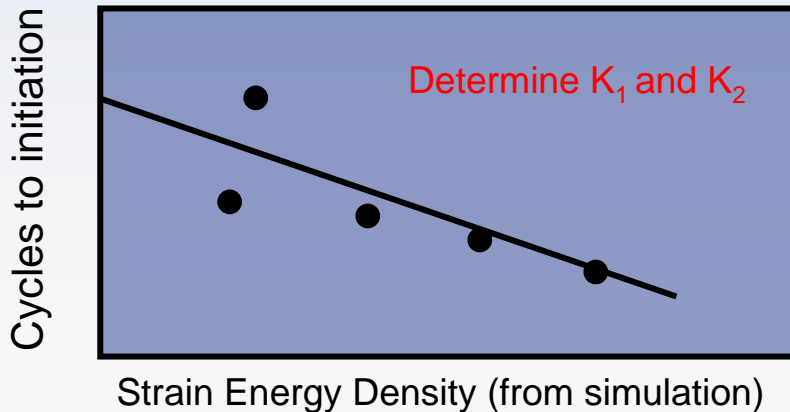
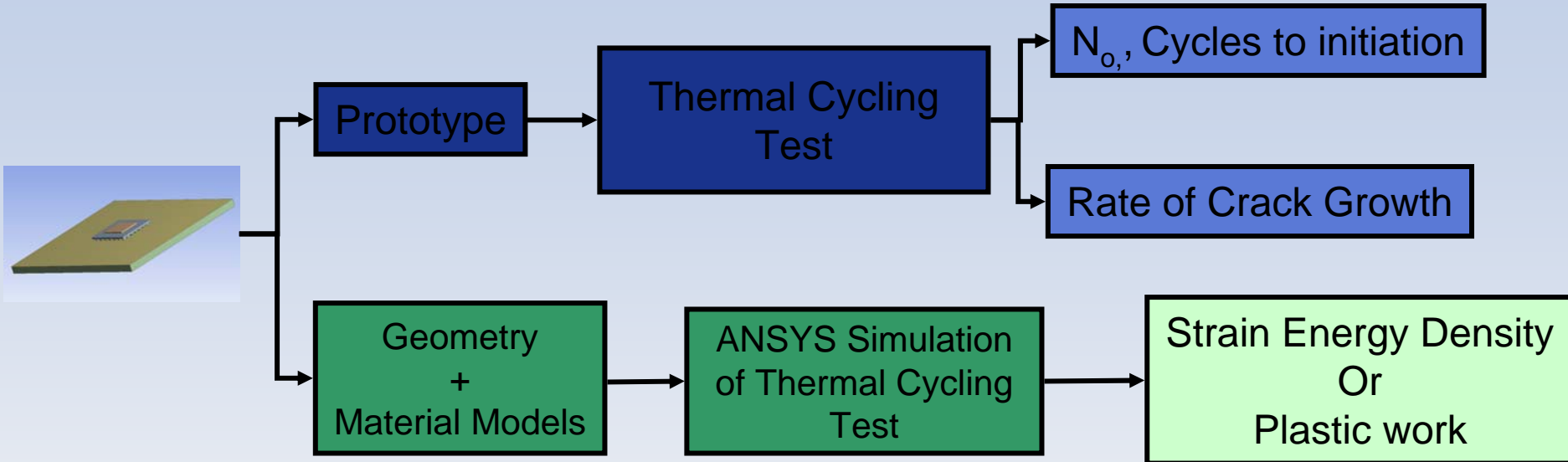
$a$  is the joint diameter at the interface (‘final crack length’)

$\Delta W$  is the plastic work per cycle

- The characteristic life  $\alpha_w$  is then related to the failure free life and cycles to first failure for failure prediction

- **Choice of constitutive model is tied to the model used to determine  $K_1$  through  $K_4$**
- **In this work we will use  $K_1$  through  $K_4$  determined by Darveaux using Anand Constitutive Model**
- **Hence Anand Constitutive model is used in this work**

# Characterization Methodology (Darveaux, 2000)



The background features a central blue globe with the ANSYS logo overlaid. The globe is surrounded by a complex network of glowing blue and orange energy trails that radiate outwards, creating a sense of dynamic motion and high-tech energy.

*Solder Joint Fatigue  
Simulation in ANSYS  
Workbench Environment*

**ANSYS®**

# ANSYS Workbench Workflow for Fatigue Simulation



Detailed BGA  
from Macro

ANSYS  
DesignModeler

ANSYS Unified  
Meshing Platform

ANSYS Workbench

Material props, Solver Settings

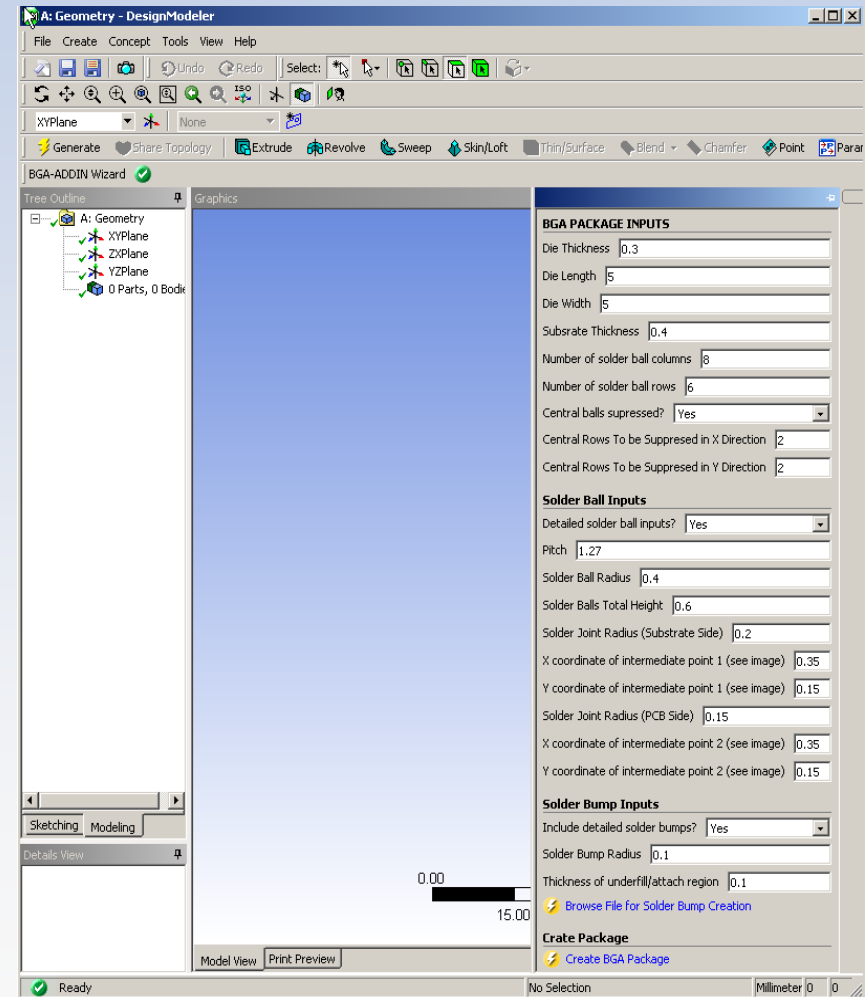
Ansys Mechanical  
for  
Thermal Cycling Stress  
Simulation

Post processing APDL  
for Cycles to failure

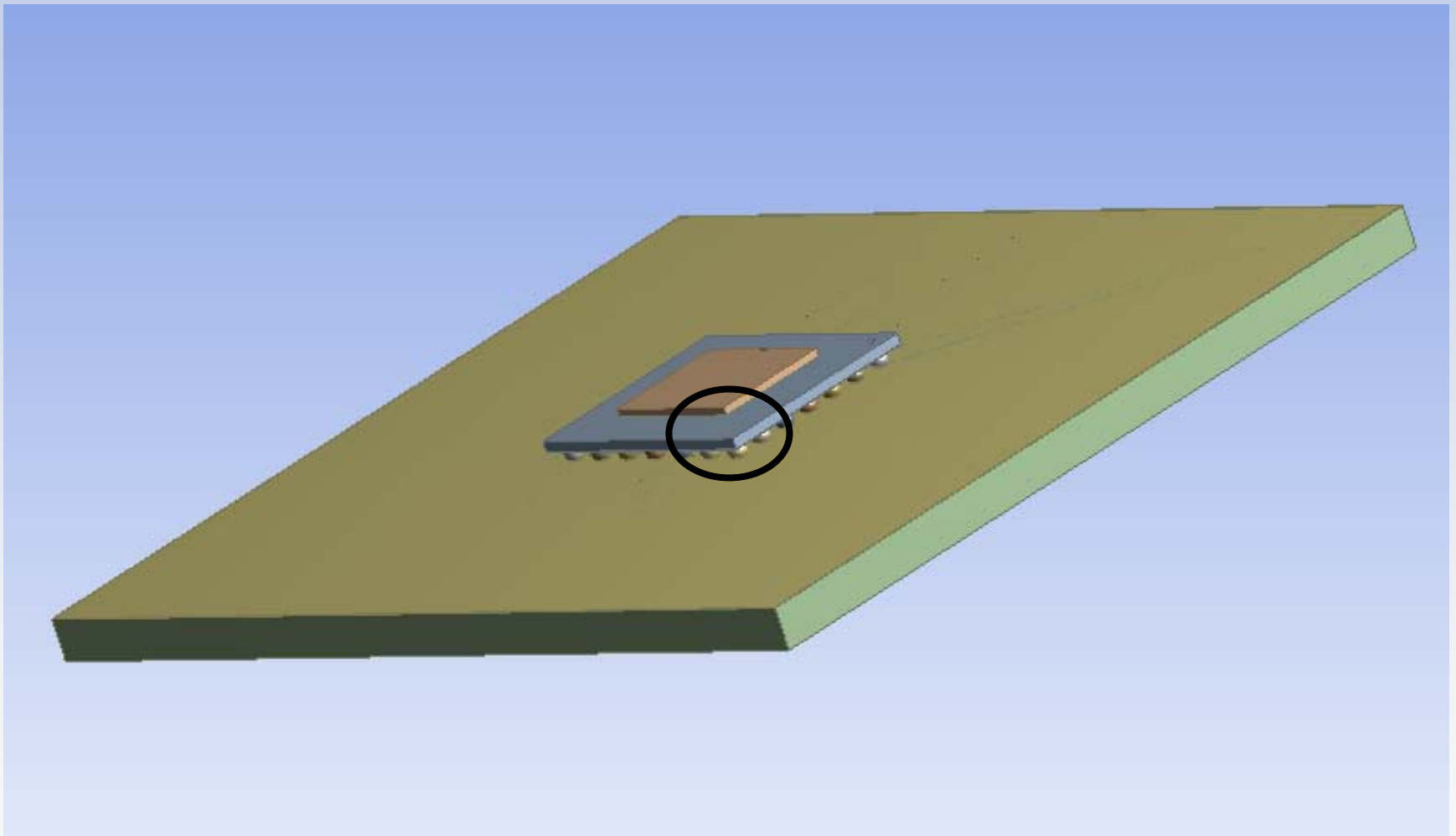
# BGA Geometry Creation Macro



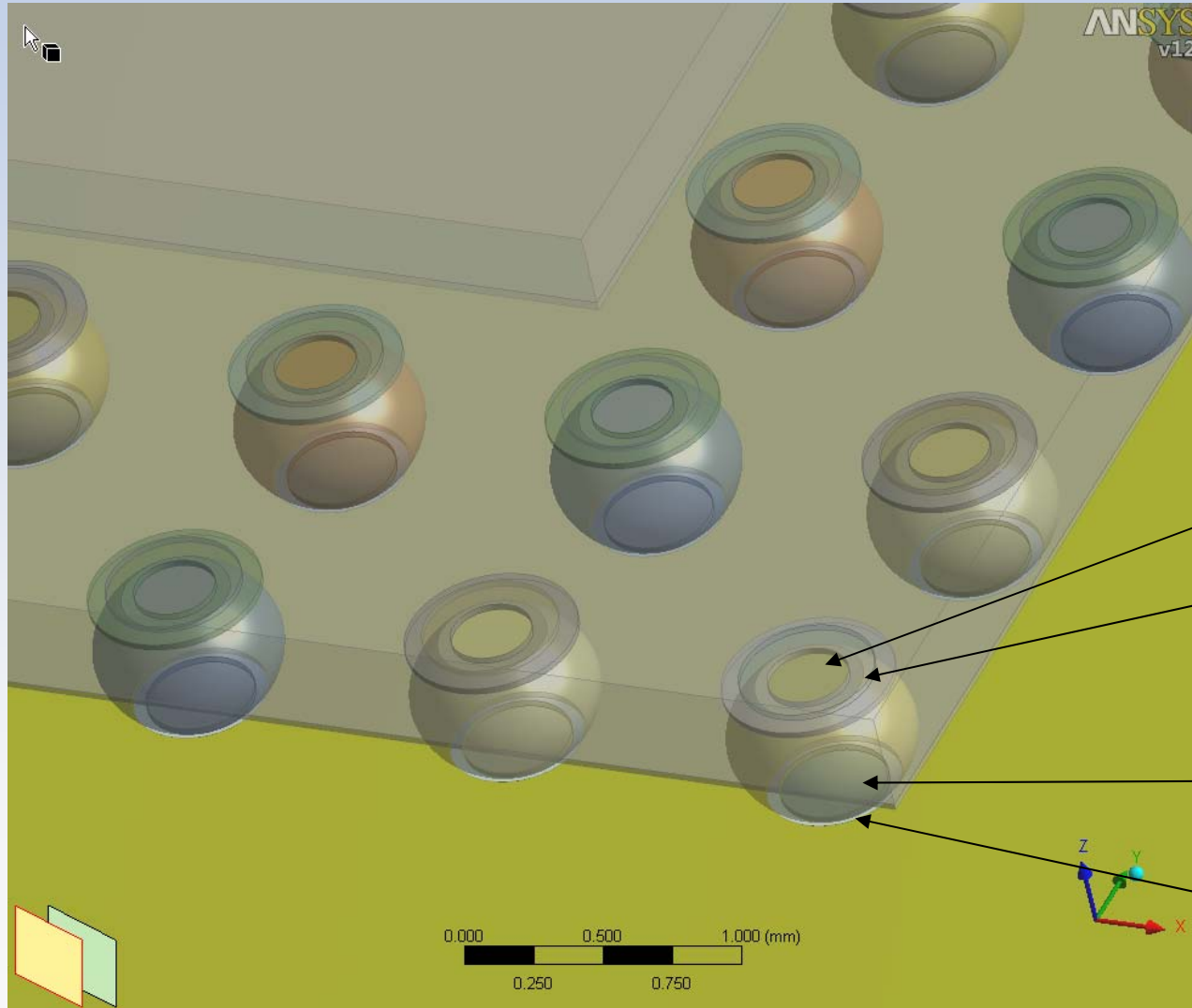
- **Ball Grid Arrays (BGA)** type Packages are well suited for customization because of their standardized geometry



# Geometric Details and Simulation Intent



# Geometric Details and Simulation Intent



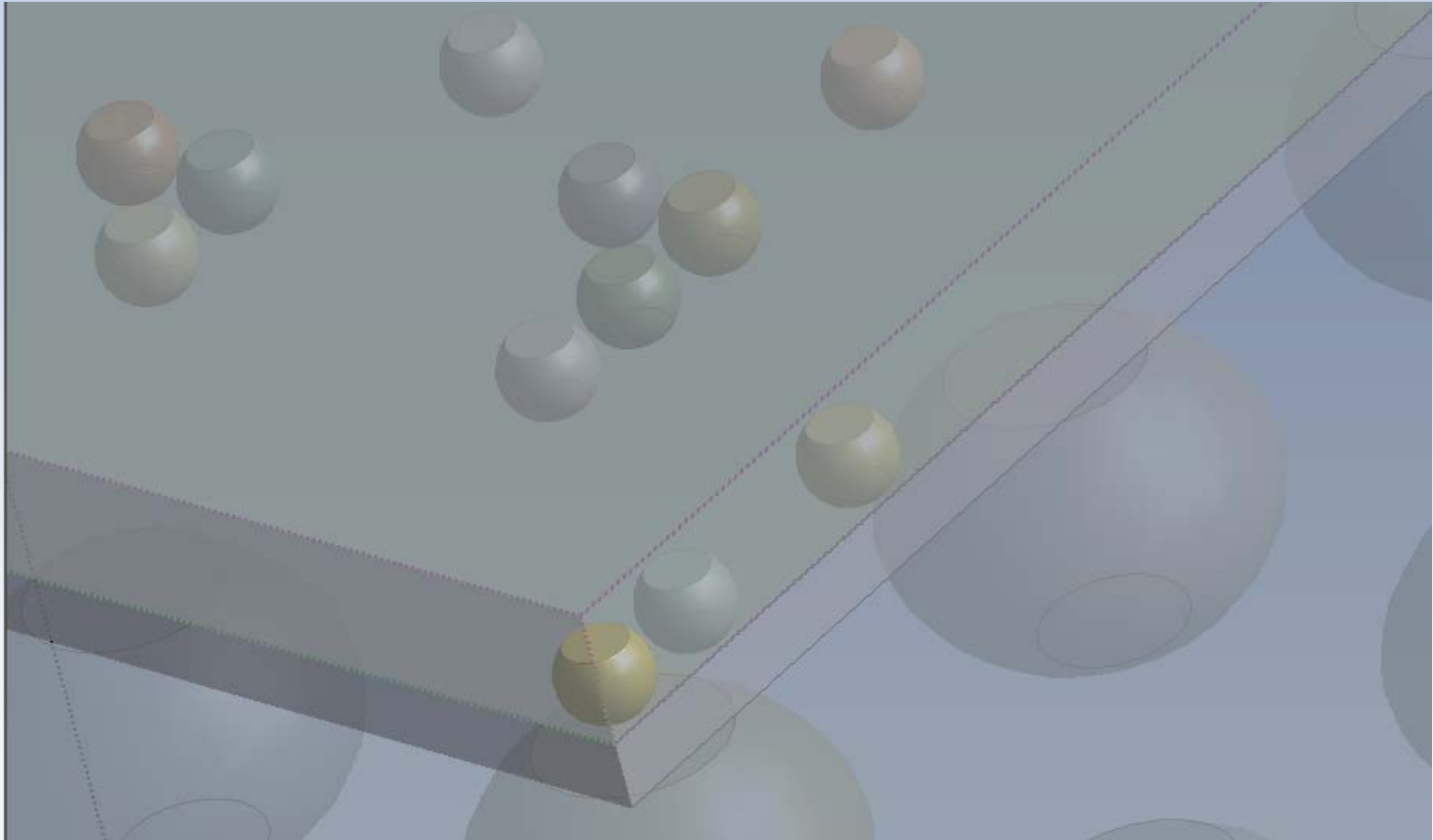
Substrate  
PAD

SMD Mask

NSMD Pad

PCB mask Opening  
(NSMD)

# *Geometric Details and Simulation Intent*



# Geometry - Possible Simplifications



M : Targeted\_SMD\_NSMD - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help | Solve Validate (Beta) |

Symmetry Symmetry Region Periodic Region

Outline

- Project
  - Model (B4)
    - Geometry
      - Coordinate Systems
        - Global Coordinate System
        - octant\_sym\_plane
        - quadrant\_for\_symmetry
      - Symmetry
        - Symmetry Region
        - Symmetry Region 2
    - Connections
    - Mesh
    - Named Selections
    - Static Structural (B5)
      - Analysis Settings
      - Thermal Condition
      - Displacement
      - Anand\_model
      - Toffset
      - bt\_substrate\_CTE
      - PCBoard\_CTE
    - Solution (B6)
      - Solution Information
      - Equivalent Stress
      - Total Deformation

ANSYS v12

Symmetry Region  
8/31/2009 9:06 AM

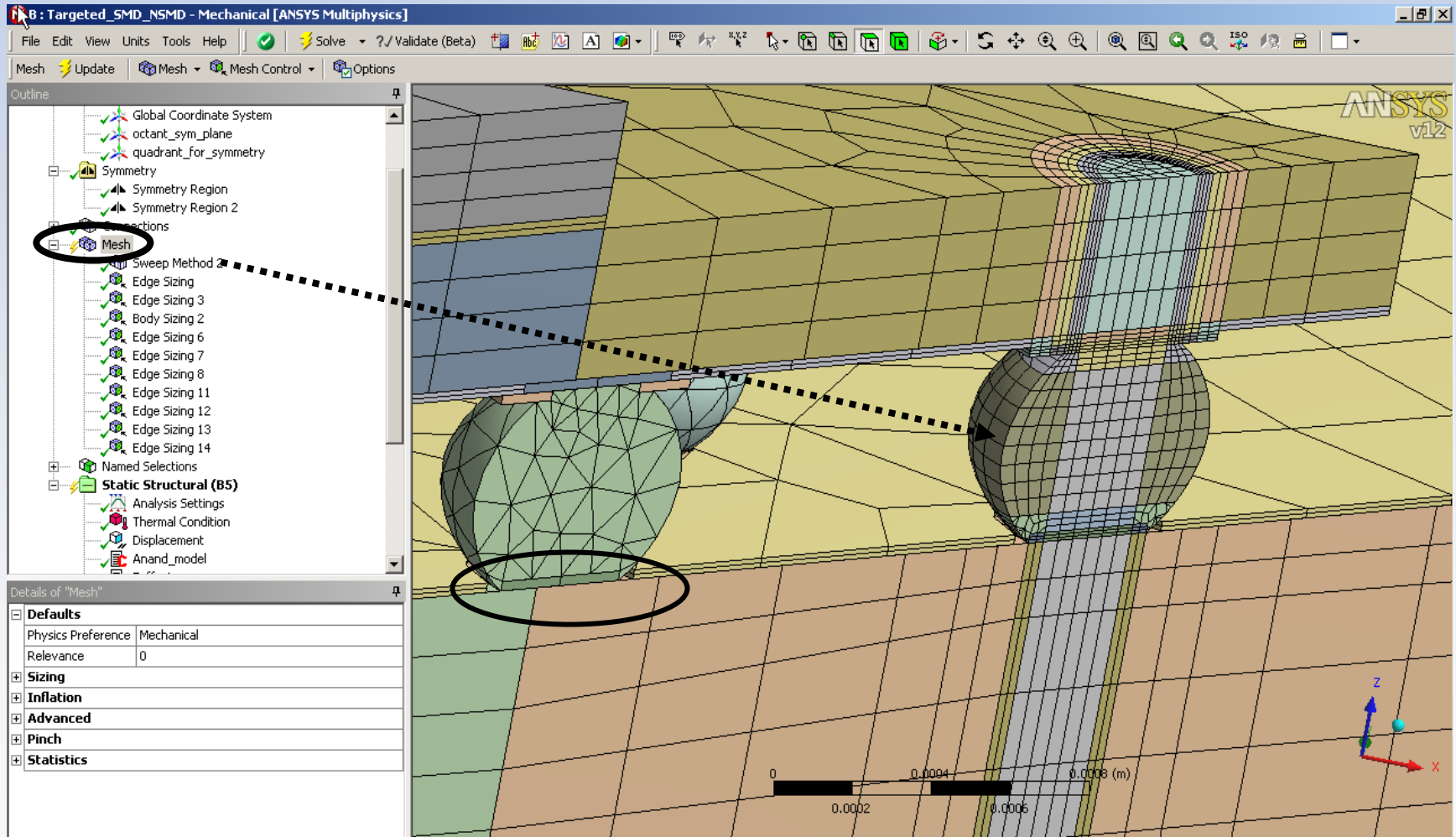
Symmetry Region

0 0.003 0.006 (m)  
0.0015 0.0045

Details of "Symmetry Region"

Scope	
Scoping Method	Named Selection
Named Selection	Symmetry:octant_sym_plane
Definition	
Scope Mode	Manual
Type	Symmetric
Coordinate System	octant_sym_plane
Symmetry Normal	Z Axis
Suppressed	No

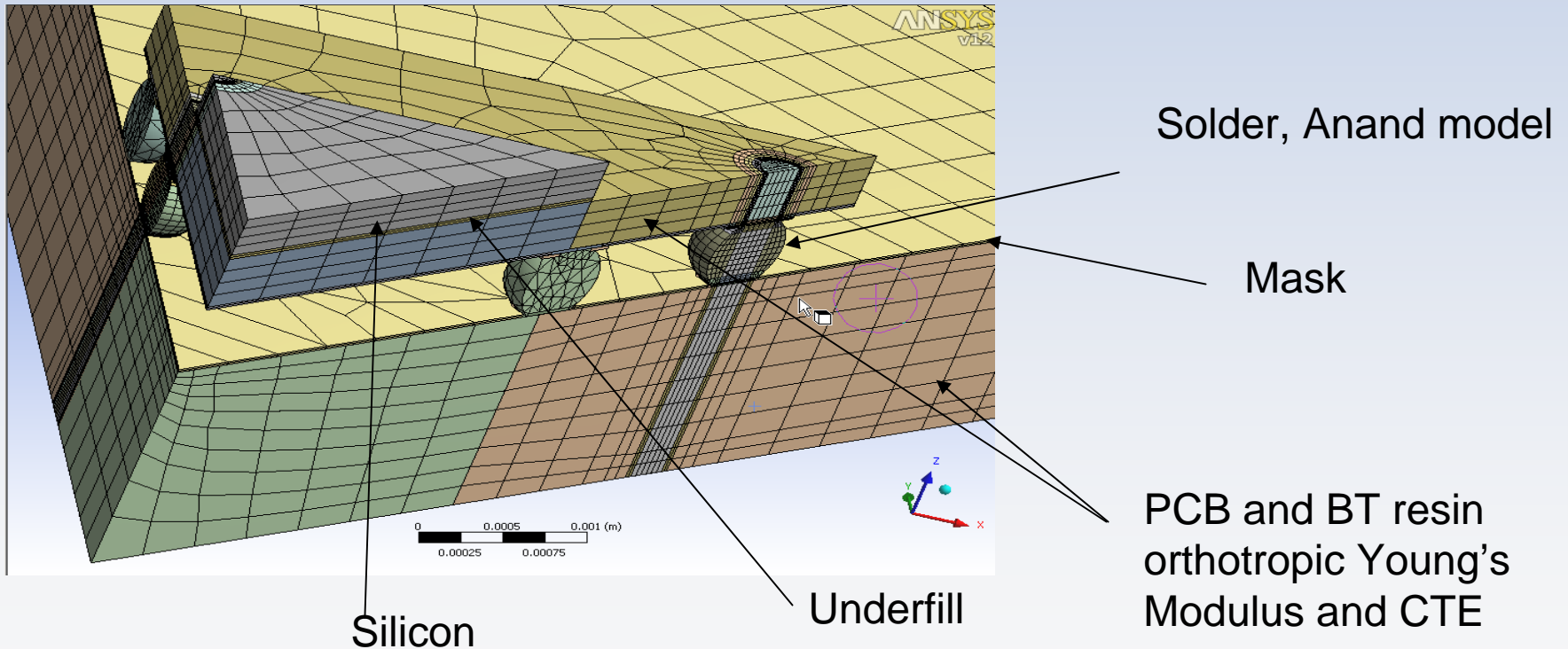
# Simulation Intent, WB Mesh Control and Contacts Technology



# Material Models and Material Properties Used for Sample Case



All materials are linear elastic except the solder ball, which uses Anand rate Dependent plasticity model

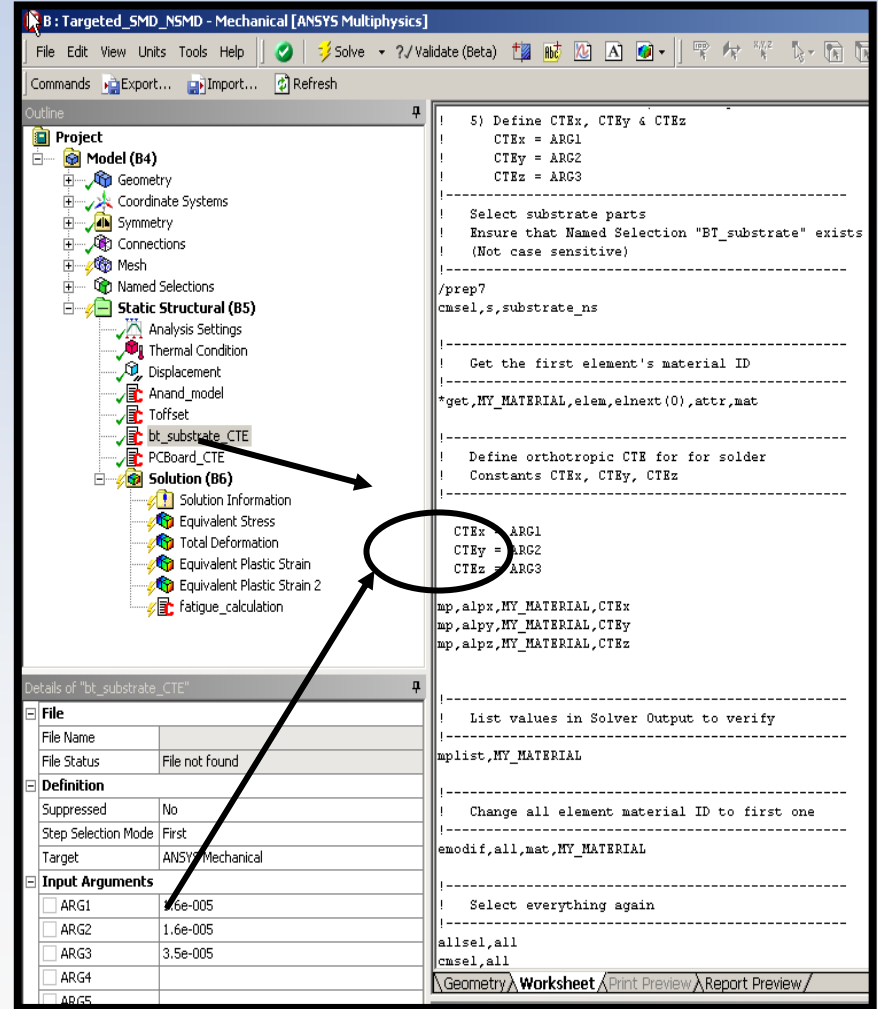
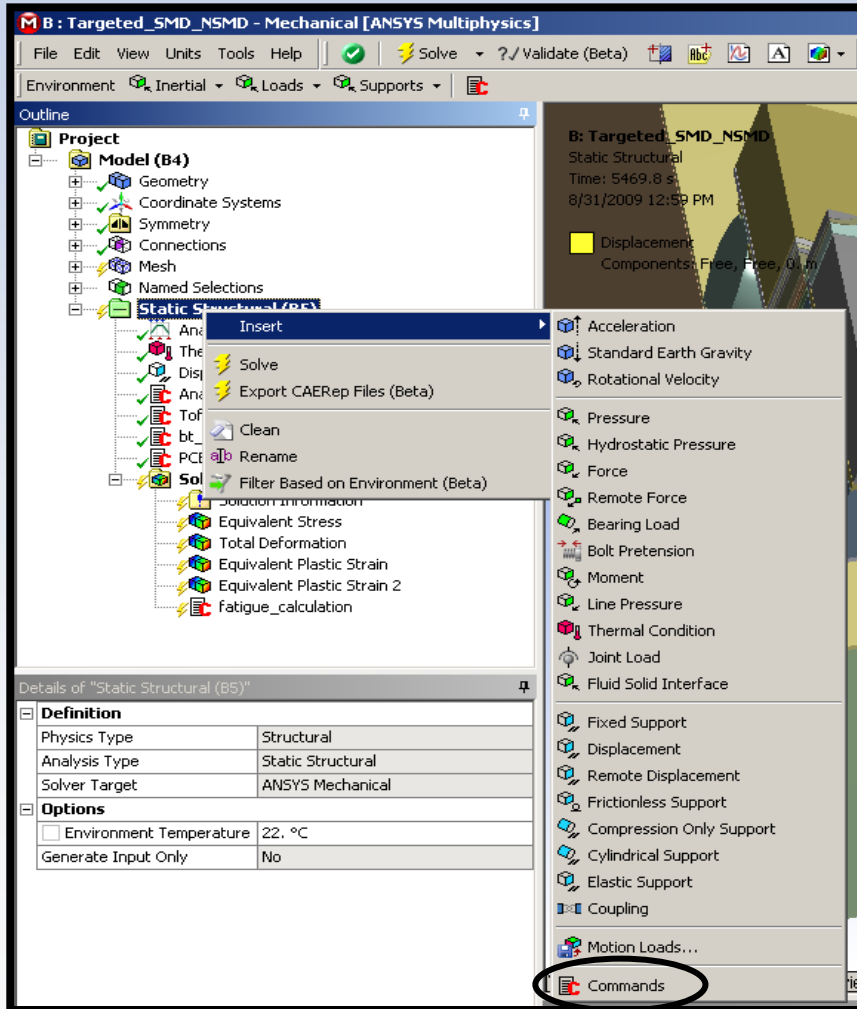


- **PCB, BT, mask, Silicon, and Solder property values were used from:**
  - “Impact of ball via configurations on solder joint reliability in tape-based, chip-scale packages” Zahn Bret A. , 52nd electronic components & technology conference : ( San Diego CA, 28-31 May 2002 )

# Commands Object for Orthotropic CTE and Plasticity Models



Parameters for Commands objects can be input from Details window in the GUI



# Commands Object for Anand Model



- Variable names ARG1, ARG2 etc in the APDL instruct WB to take inputs from the Details window for the corresponding commands object.

The screenshot displays the ANSYS APDL interface. The main window shows the Commands window with the following APDL code:

```
! C1=ARG1
! C2=ARG2
! C3=ARG3
! C4=ARG4
! C5=ARG5
! C6=ARG6
! C7=ARG7
! C8=ARG8
! C9=ARG9
!-----
! Select solder parts
! Ensure that Named Selection "SolderAll" exists
! (Not case sensitive)
!-----
/prep7
cnsel,s,SolderAll
!-----
! Get the first element's material ID
*get,MY_MATERIAL,elem,e1next(0),attr,mat
!-----
! Define Anand Model for solder
! Constants C1-C9
!-----
C1=ARG1
C2=ARG2
C3=ARG3
C4=ARG4
C5=ARG5
C6=ARG6
C7=ARG7
C8=ARG8
C9=ARG9
!-----
cb,Rate,MY_MATERIAL,,ANAND
tbddata,1,C1
tbddata,2,C2
tbddata,3,C3
tbddata,4,C4
tbddata,5,C5
tbddata,6,C6
tbddata,7,C7
tbddata,8,C8
tbddata,9,C9
!-----
! List values in Solver Output to verify
tlist,Rate,MY_MATERIAL
!-----
! Change all element material ID to first one
!-----
emodif,all,mat,MY_MATERIAL
!-----
! Select everything again
!-----
allsel,all
cnsel,all
/solu
```

The Details of "Anand\_model" window shows the following input arguments:

Input Argument	Value
ARG1	1.24e+007
ARG2	9400.
ARG3	4.e+006
ARG4	1.5
ARG5	0.303
ARG6	1.378e+009
ARG7	1.38e+007
ARG8	7.e-002
ARG9	1.3

# Thermal Cycle Input



**B: Targeted\_SMD\_NSMD - Mechanical [ANSYS Multiphysics]**

File Edit View Units Tools Help Solve Validate (Beta) ...

Environment Inertial Loads Supports

Outline

- Project
  - Model (B4)
    - Geometry
    - Coordinate Systems
    - Symmetry
    - Connections
    - Mesh
    - Named Selections
  - Static Structural (B5)
    - Analysis Settings
    - Thermal Condition
    - Displacement
    - Anand\_model
    - Toffset
    - bt\_substrate\_CTE
    - PCBoard\_CTE
  - Solution (B6)
    - Solution Information
    - Equivalent Stress
    - Total Deformation
    - Equivalent Plastic Strain
    - Equivalent Plastic Strain 2
    - fatigue\_calculation

Details of "Thermal Condition"

**Scope**

Scoping Method	Geometry Selection
Geometry	97 Bodies

**Definition**

ID (Beta)	599
Type	Thermal Condition
Magnitude	Tabular Data
Suppressed	No

**Tabular Data**

Independent Variable	Time
----------------------	------

**B: Targeted\_SMD\_NSMD**  
Thermal Condition  
Time: 0. s  
8/31/2009 4:06 PM

Thermal Condition: 22. °C

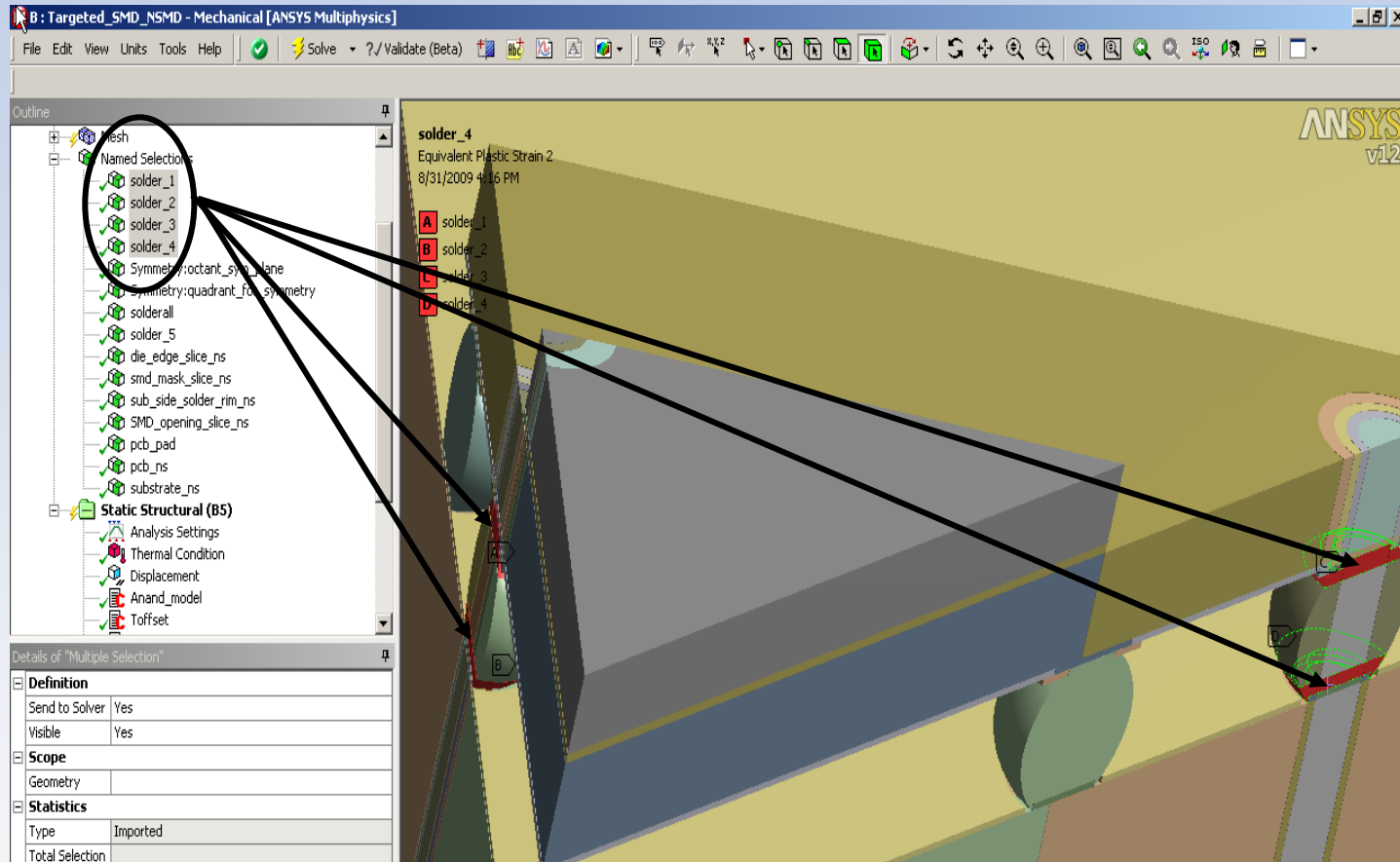
0 0.003 0.006 (m)  
0.0015 0.0045

Graph

Tabular Data

Steps	Time [s]	Temperature [°C]	
1	0.	22.	
2	1	300.	-40.
3	2	2100.	-40.
4	3	2400.	85.
5	4	4200.	85.
6	5	4500.	-40.
7	6	6300.	-40.
8	7	6600.	85.
9	8	8400.	85.
10	9	8700.	-40.
11	10	10500.	-40.
12	11	10800.	85.
13	12	12600.	85.

# User Choice for Performing Fatigue Calculations



Named selections for sections of the solder joint expected to experience highest plastic strain fluctuations

# Fatigue Post-Processing Calculations



The screenshot displays the ANSYS Multiphysics interface. The left pane shows the project tree with 'Fatigue\_calculation' selected. The bottom-left pane shows 'Input Arguments' and 'Results'. The main window shows the APDL script with annotations.

**APDL inserted as command Object in Solution branch**

**Results are calculated after Solution is completed**

**"My\_..." variables in the script make results appear in the Details window**

Input Argument	Value
ARG1	22400
ARG2	-1.52
ARG3	5.8e-007
ARG4	0.90
ARG5	1.45e-004
ARG6	4.
ARG7	4200.
ARG8	4.e-004
ARG9	

Result	Value
my_solder_1_wdiff_3_2	0.43518
my_solder_1_wdiff_4_3	0.41324
my_solder_2_wdiff_3_2	0.6067
my_solder_2_wdiff_4_3	0.56321
my_solder_3_wdiff_3_2	1.5117
my_solder_3_wdiff_4_3	1.4421
my_n0_solder1	83968
my_nprop_solder1	63649
my_n0_solder2	93608
my_nprop_solder2	47659
my_n0_solder3	12841

- Sample macros will be available for download at ANSYS Customer portal



## *Results and Discussion*

# Cycles to failure calculation

M B : Targeted\_SMD\_NSMD - Mechanical [ANSYS Multiphysics]

File Edit View Units Tools Help Solve Validate (Beta)

Commands Export... Import... Refresh Search Parameters

Outline

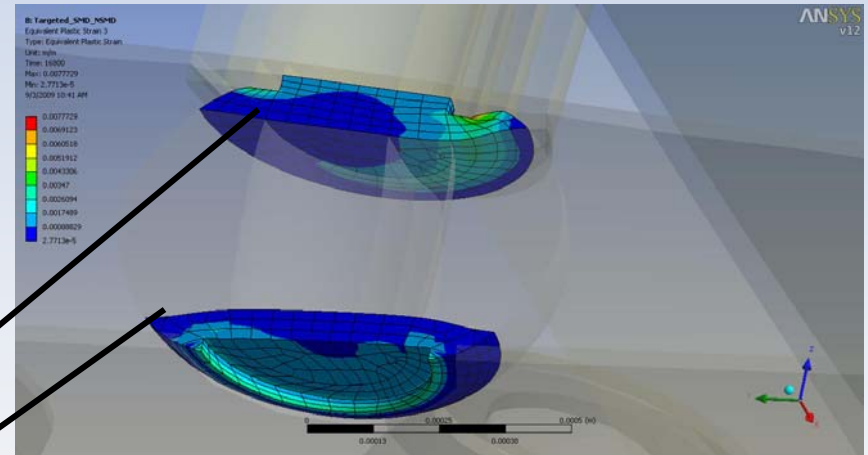
- Anand\_model
- Toffset
- bt\_substrate\_CTE
- PCBoard\_CTE
- Solution (B6)
  - Solution Information
  - Equivalent Stress
  - Total Deformation
  - Equivalent Plastic Strain
  - Equivalent Plastic Strain 2
  - fatigue\_calculation

Details of "fatigue\_calculation"

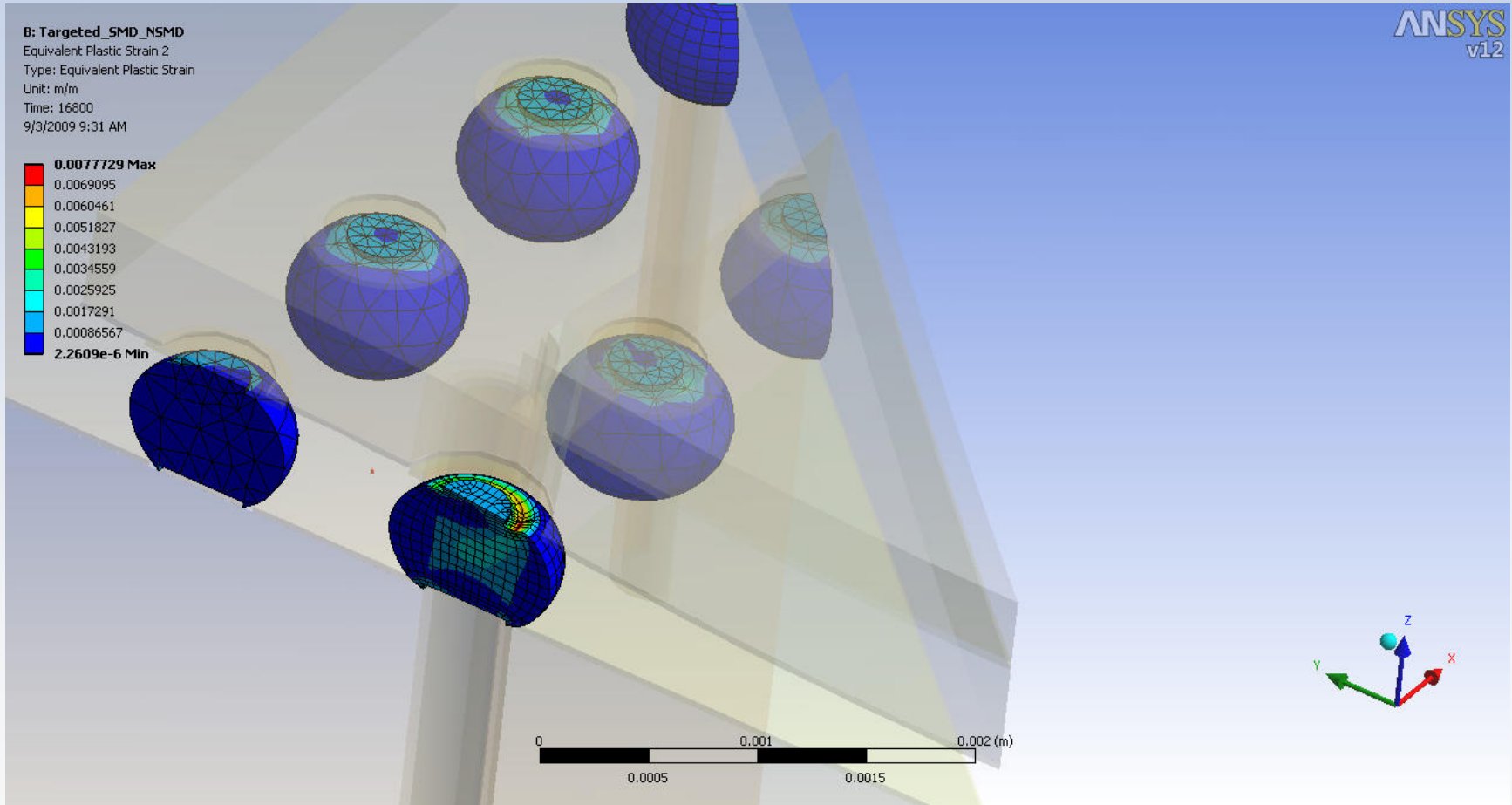
<input type="checkbox"/> ARG1	22400
<input type="checkbox"/> ARG2	-1.52
<input type="checkbox"/> ARG3	5.8e-007
<input type="checkbox"/> ARG4	0.98
<input type="checkbox"/> ARG5	1.45e-004
<input type="checkbox"/> ARG6	4.
<input type="checkbox"/> ARG7	4200.
<input type="checkbox"/> ARG8	4.e-004
<input type="checkbox"/> ARG9	

Results

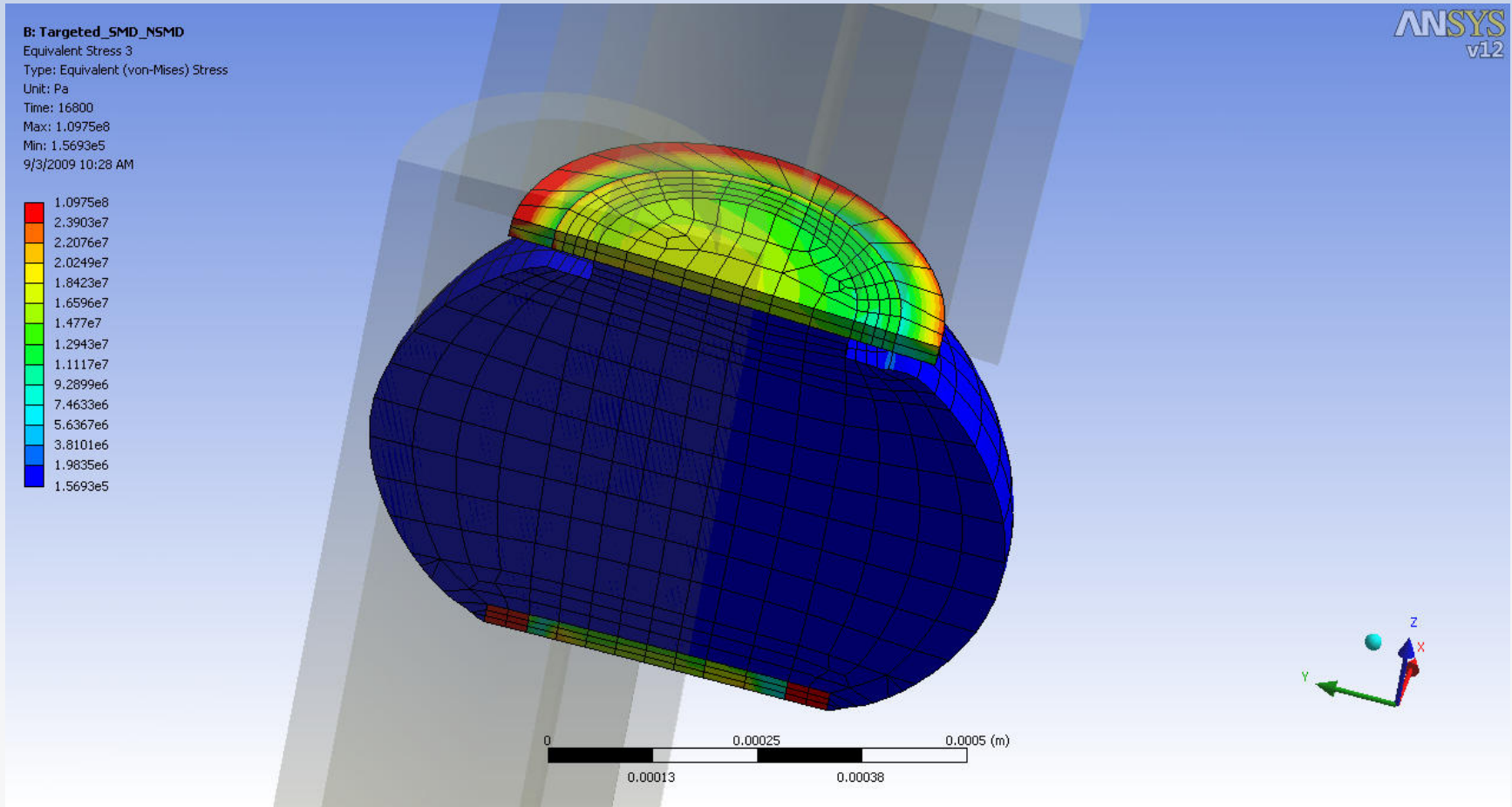
<input type="checkbox"/> my_solder_1_wdiff_3_2	0.3565
<input type="checkbox"/> my_solder_1_wdiff_4_3	0.33773
<input type="checkbox"/> my_solder_2_wdiff_3_2	1.0669
<input type="checkbox"/> my_solder_2_wdiff_4_3	1.0182
<input type="checkbox"/> my_solder_3_wdiff_3_2	0.18217
<input type="checkbox"/> my_solder_3_wdiff_4_3	0.17271
<input type="checkbox"/> my_n0_solder1	1.1663e+005
<input type="checkbox"/> my_nprop_solder1	78668
<input type="checkbox"/> my_n0_solder2	21793
<input checked="" type="checkbox"/> my_nprop_solder2	26675
<input type="checkbox"/> my_n0_solder3	3.2323e+005
<input type="checkbox"/> my_nprop_solder3	1.5178e+005



# Plastic strain on all solders



# Stresses on solder pad



# *ANSYS Software Products Needed*

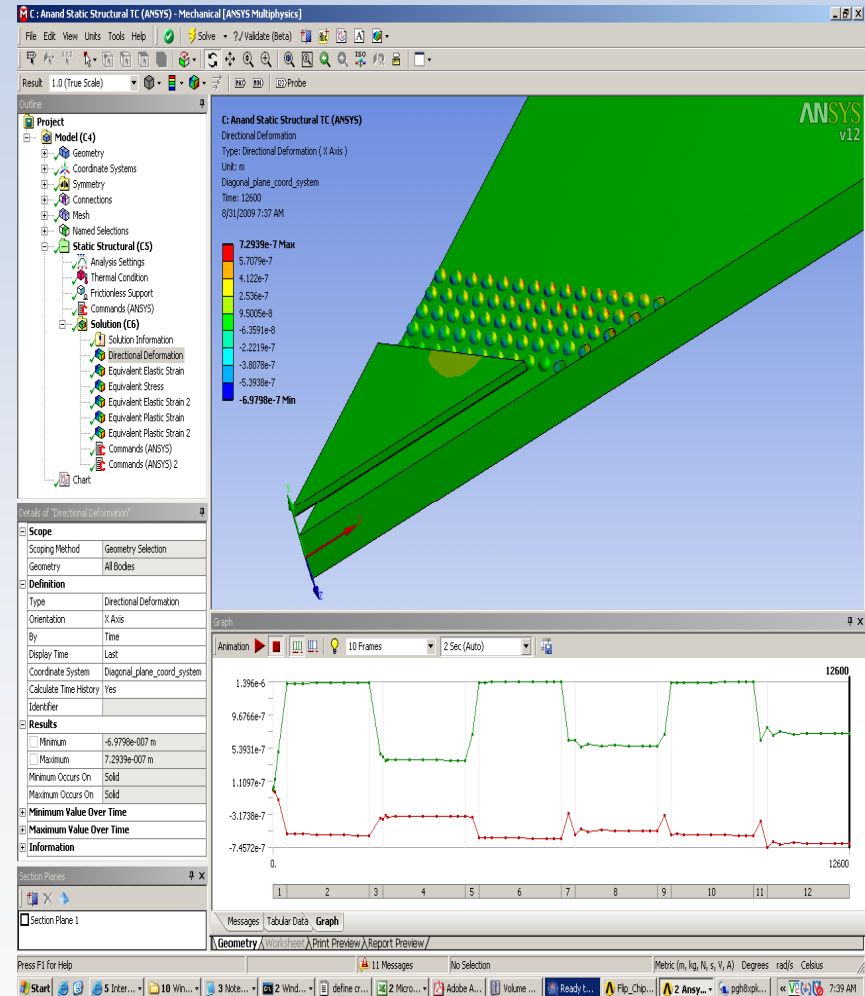


- **ANSYS DesignModeler**
- **ANSYS Mechanical**

# How Can ANSYS Help You?



- **BGA geometry creation macro**
- **Workflow instructions**
  - ANSYS Workbench Capabilities for geometry processing and meshing
  - Material models in ANSYS Mechanical
  - Sample APDL scripts for GUI based user inputs for Material models and Fatigue Model
- **Sample ANSYS Workbench Project**
- **Jump start assistance through service projects**
- **Focused Training?**



- **The following materials have already been uploaded to ANSYS Customer Portal**
  - A workflow Tutorial for performing Fatigue Calculation using Anand constitutive model and Plasticity based TTF prediction
  - A sample model containing more details and finer mesh than the tutorial problem will also be available from ANSYS Customer Portal.

- **We presented an overview of design challenges in electronics due to lead free transition**
  - Focused on package specific challenges
- **Highlighted why ANSYS technology is best suited to address these challenges**
- **Used Solder Joint Fatigue simulation of a BGA package to illustrate workbench based workflow using ANSYS Mechanical software**

- **Example workflow illustrated the following unique strengths**
  - Geometry creation time savings due to DesignModeler Macro for BGA
  - ANSYS Meshing platform for high quality refined mesh where needed
  - Contacts technology for computational cost savings
  - Productivity enhancements through Automatic Contact detection
  - Time tested ANSYS Mechanical solver with advanced non-linear models
    - Viscoplasticity and creep modeling capabilities
  - Workbench platform bridges the power of solver lever manipulation using APDL with user friendliness

*Thank You!*