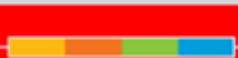
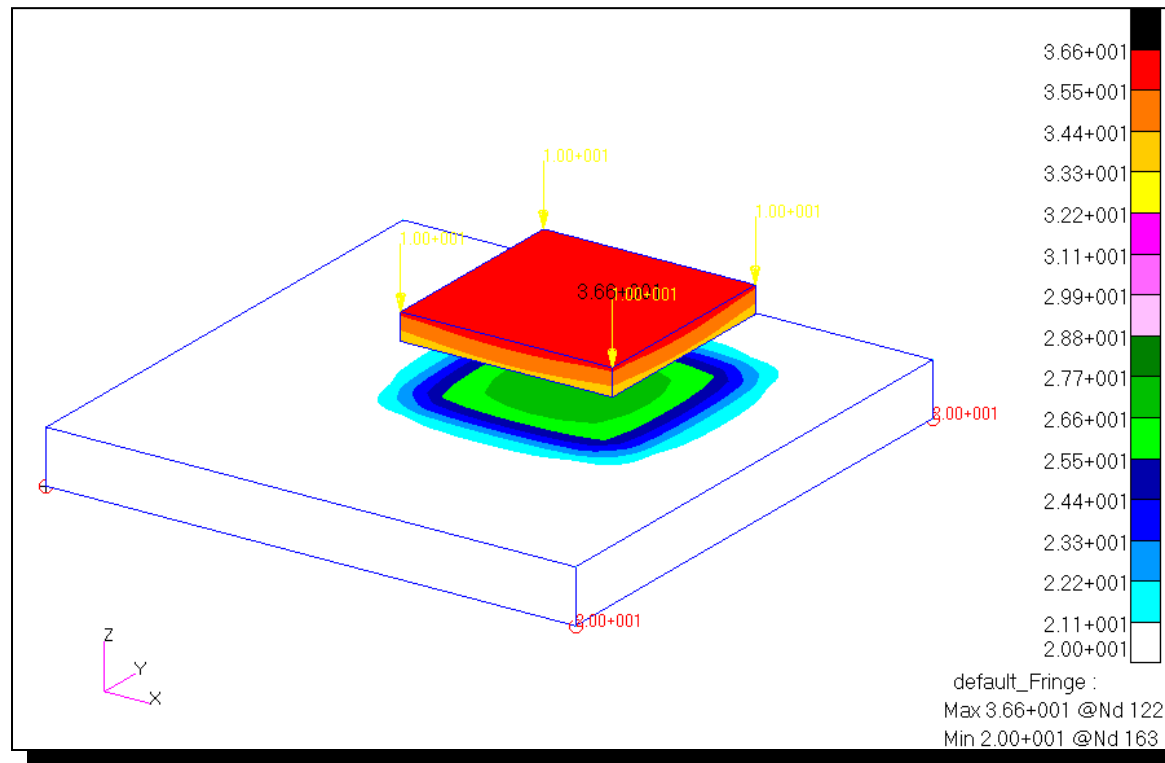


WORKSHOP

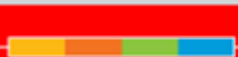
THERMAL CONTACT RESISTANCE

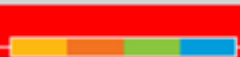
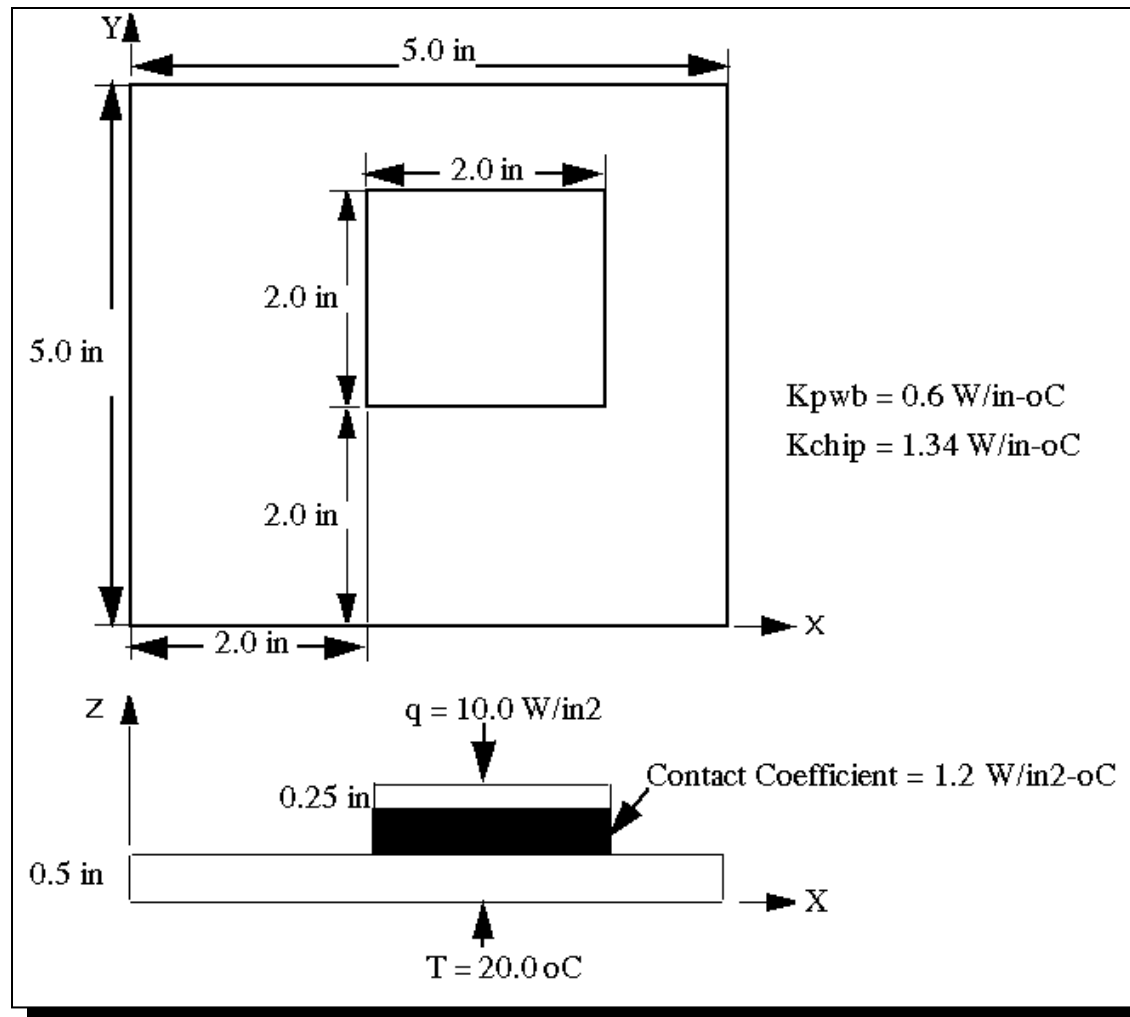




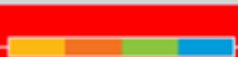
- Problem Description

- In this example, two solid Hex8 meshes are connected using convection with the Option set to Coupled. Instead of connecting the Hex8 elements with conduction elements, the connection is made using contact resistance via convection.
 - In the previous workshops, to model heat transfer across components, the meshes on adjacent parts had to match each other and then were equivalenced together. For some models, this is either impossible due to technical difficulty or time constraints. This technique gives an a way around this challenge, if the user has a good understanding of the coupled convection abilities of MD Nastran. This technique has numerous applications outside of this workshop. Students are encouraged to think up some uses and discuss them with the class.
- Heat flux is applied to the top of the chip and a fixed temperature is specified at the bottom of the PCB.
- For this model, determine the maximum temperature at the top of the chip, and the temperature drop to the bottom of the PCB.

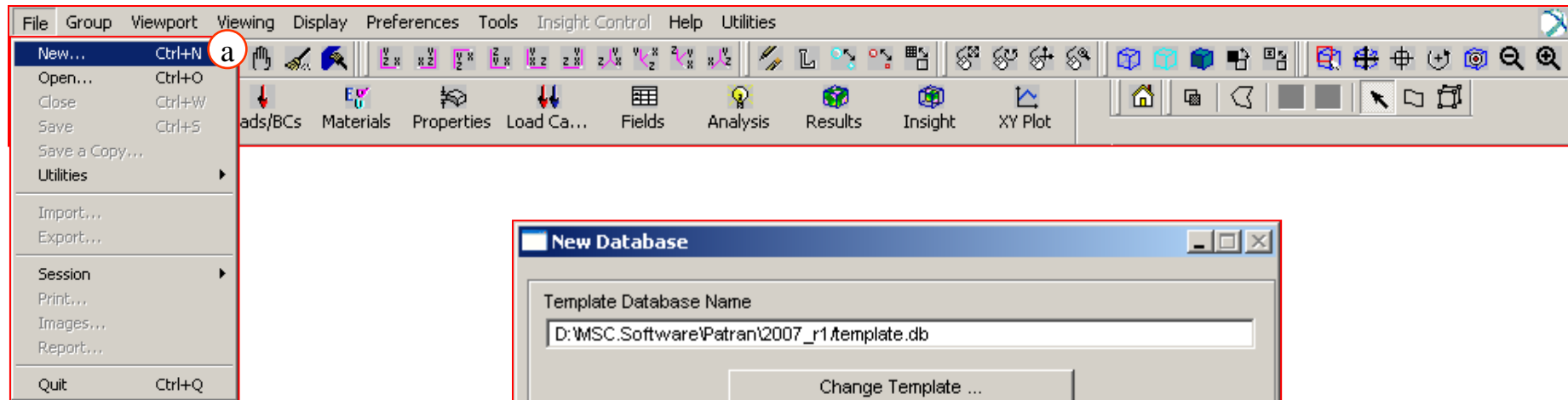




-
- Suggested Exercise Steps
 - Create a new database.
 - Set the preference to MD Nastran thermal.
 - Create solids for IsoMesh with Hex8 elements.
 - IsoMesh the solids.
 - Specify materials.
 - Define element properties.
 - Create convection between the two solid meshes.
 - Apply a heat flux.
 - Apply a temperature boundary condition.
 - Perform steady-state thermal analysis.
 - Attach the results file.
 - Display the temperature results.
 - Quit MD Patran.

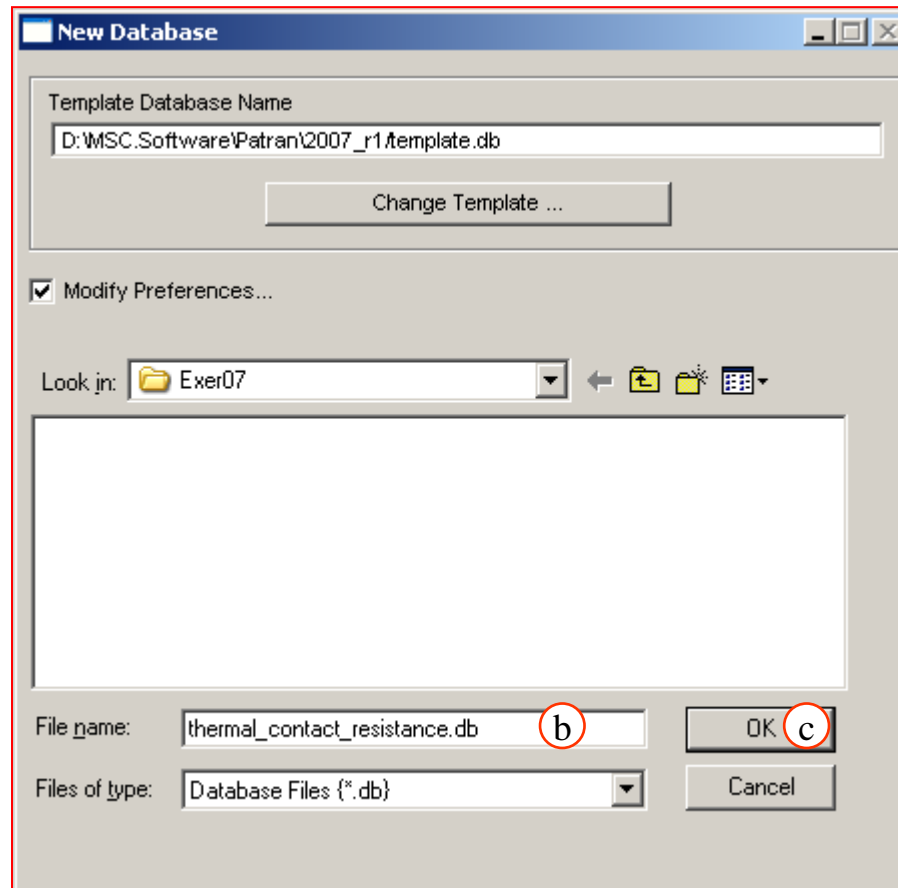


Step 1: Create a New Database

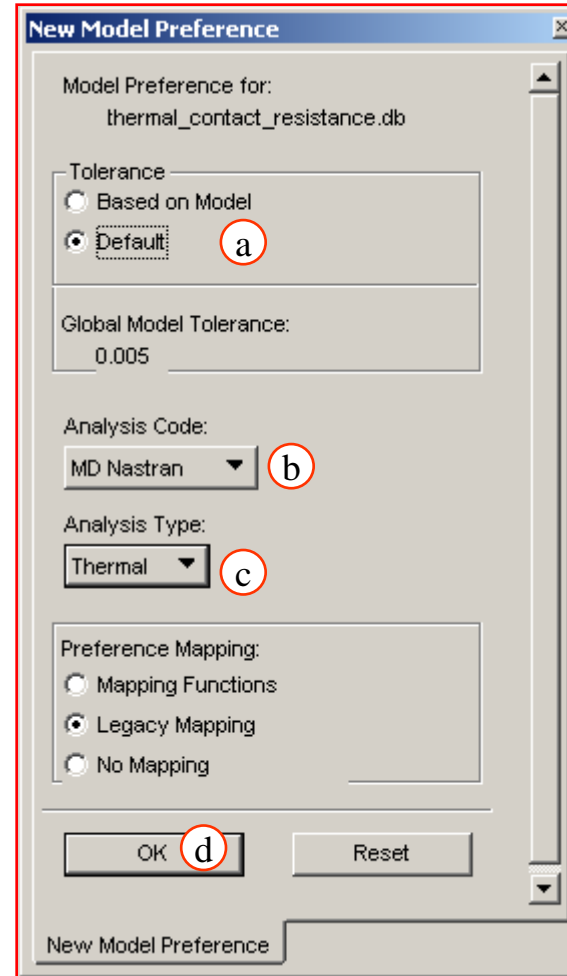


Create a new database:

- a. File: New.
- b. Enter **thermal_contact_resistance.db** for File name
- c. Click **OK**.



Step 2: Use MD Nastran Thermal Solver

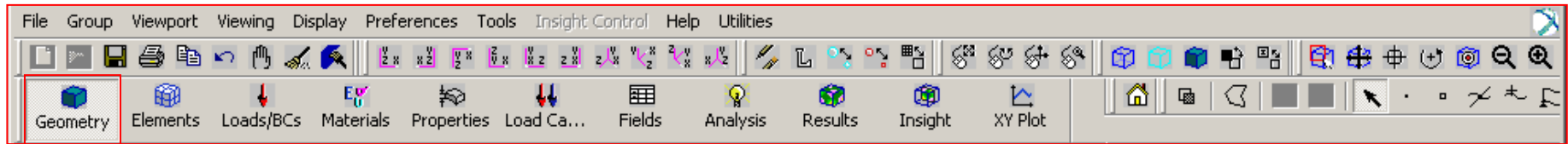


Set the analysis code and analysis type to MD Nastran Thermal:

- a. Select **Default** for Tolerance.
- b. Select **MD Nastran** for Analysis Code.
- c. Select **Thermal** for Analysis Type.
- d. Click **OK**.

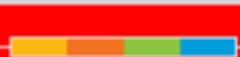
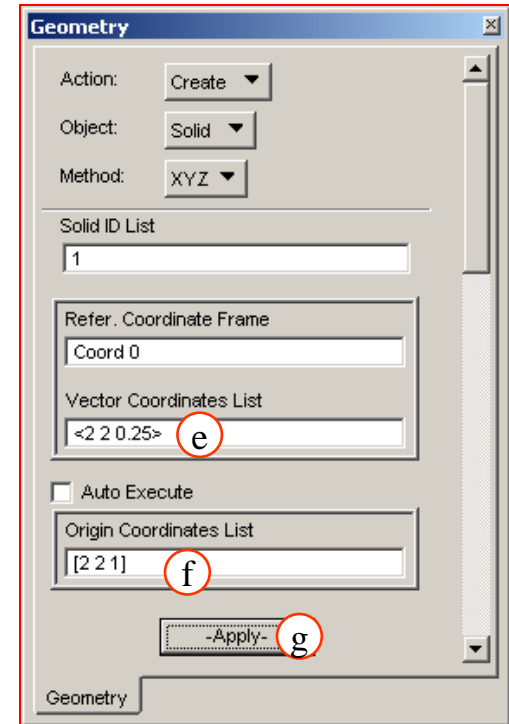
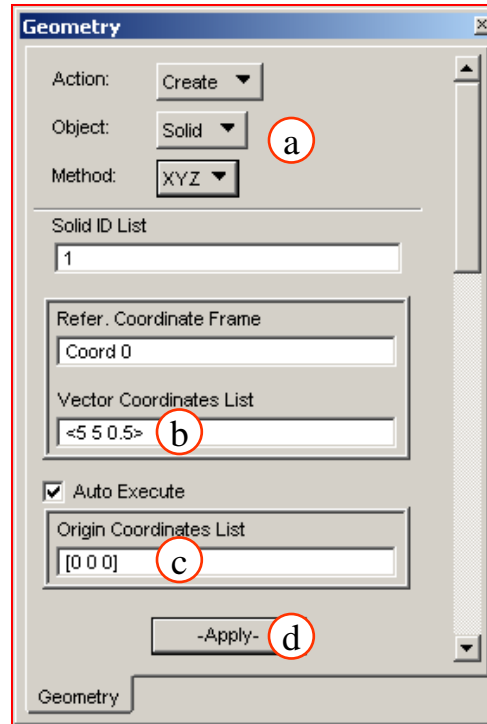


Step 3: Create Solids for IsoMesh With Hex8 Elements

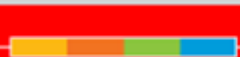
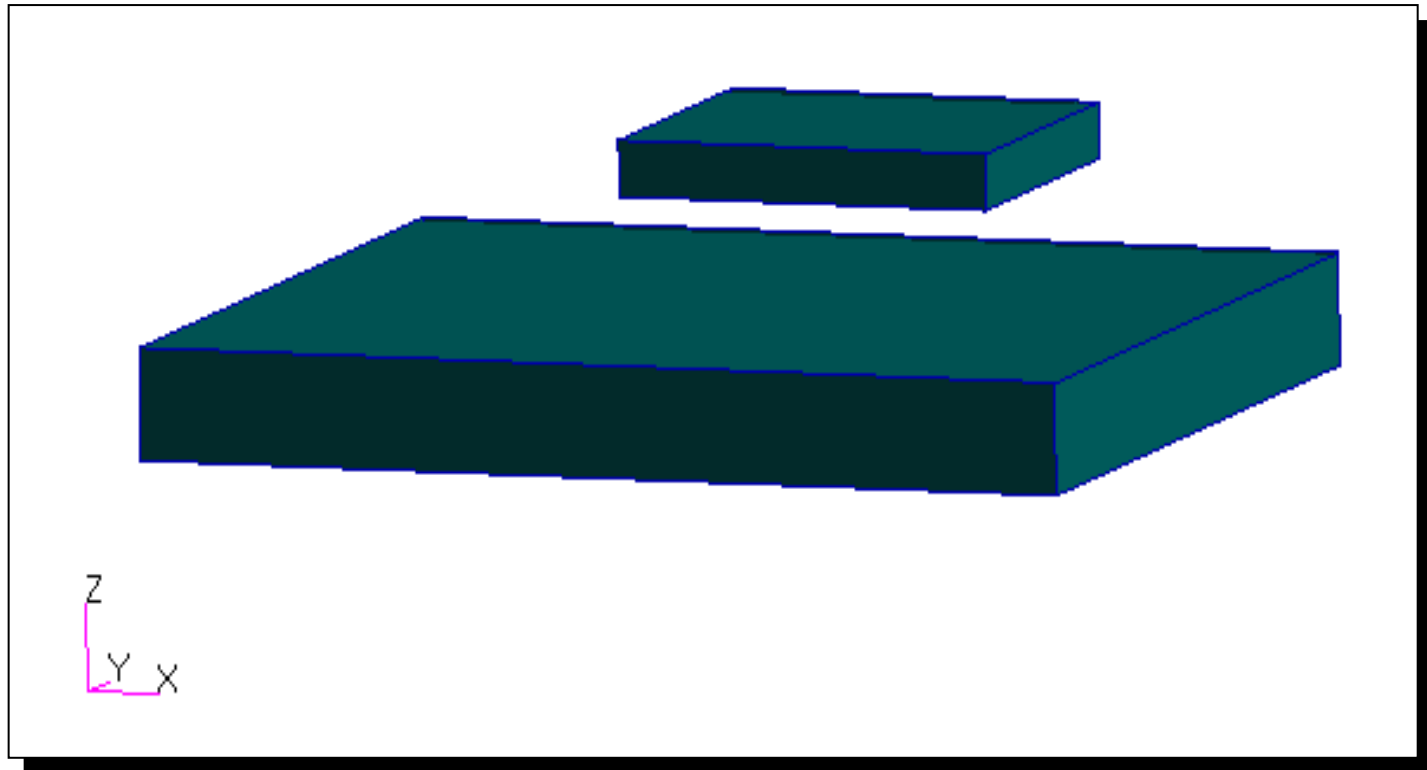


Create the solids representing the PCB and chip:

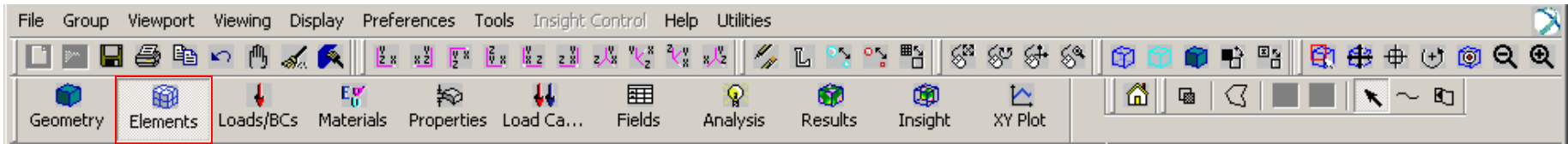
- a. **Geometry : Create / Solid / XYZ.**
- b. Enter **<5 5 0.5>** for Vector Coordinates List.
- c. Enter **[0 0 0]** for Origin Coordinates List.
- d. Click **Apply**.
- e. Enter **<2 2 0.25>** for Vector Coordinates List.
- f. Enter **[2 2 1]** for Origin Coordinates List.
- g. Click **Apply**.



Step 3: Create Solids for IsoMesh With Hex8 Elements (Cont.)

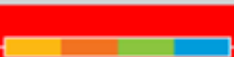
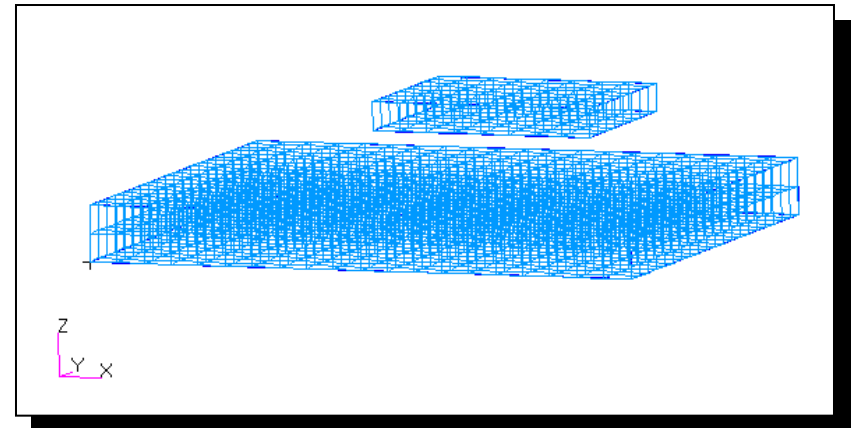
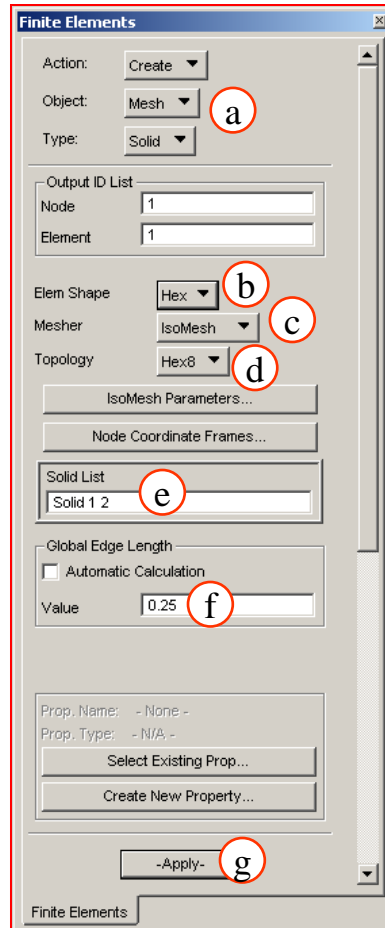


Step 4: IsoMesh the Solids

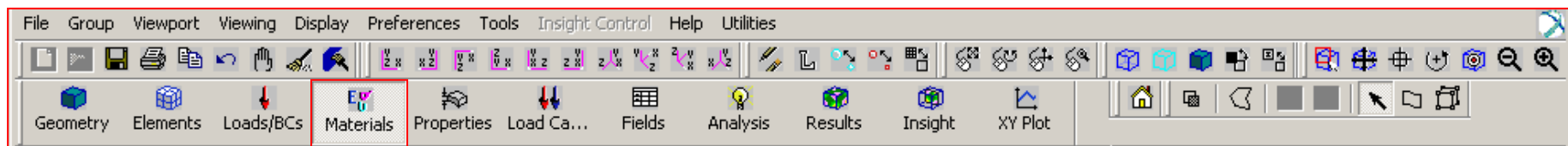


IsoMesh the two solids using Hex8 elements:

- a. **Elements : Create / Mesh / Solid.**
- b. Select **Hex** for Elem Shape.
- c. Select **IsoMesh** for Mesher.
- d. Select **Hex8** for Topology.
- e. Enter **Solid 1 2** for Solid List.
- f. Enter **0.25** for Value of Global Edge Length.
- g. Click **Apply**.

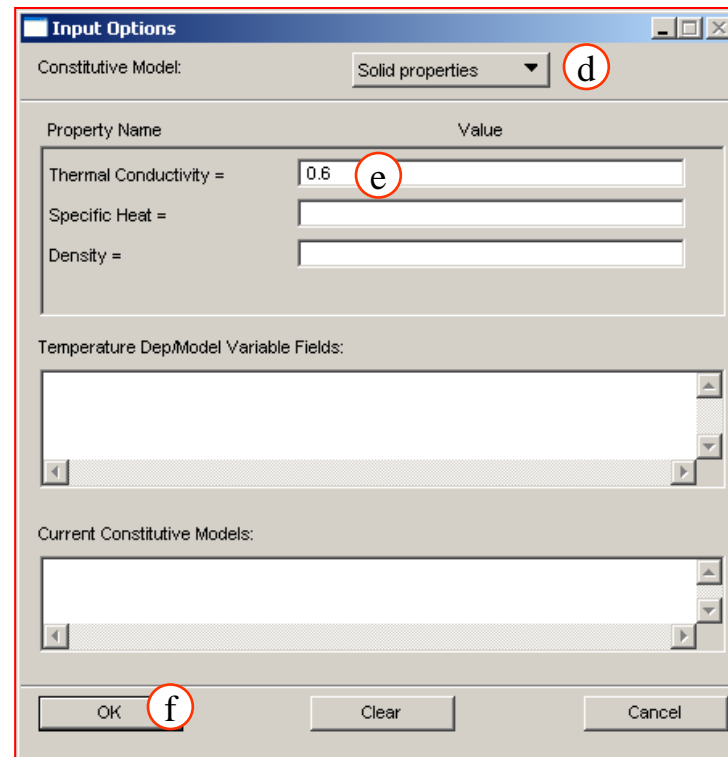
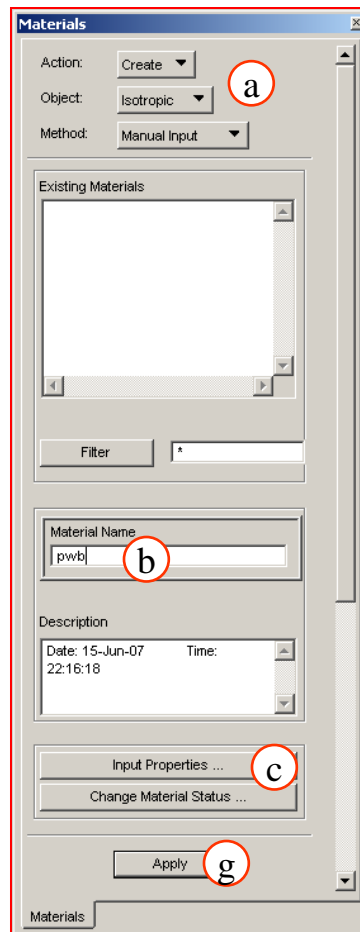


Step 5: Specify Materials

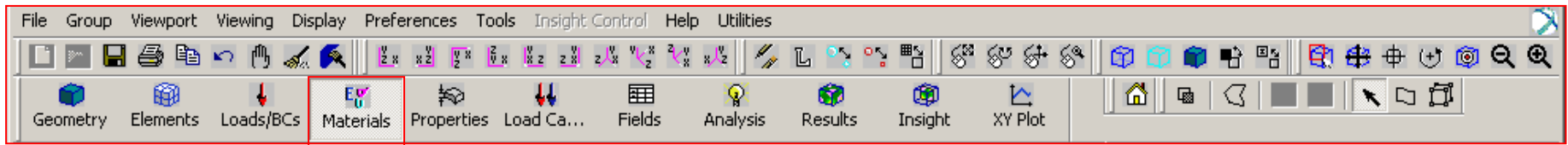


Create the isotropic material property for the PCB:

- a. **Materials : Create / Isotropic / Manual Input.**
- b. Enter **pwb** for Material Name.
- c. Click on **Input Properties**.
- d. Select **Solid properties** for Constitutive Model.
- e. Enter **0.6** for Thermal Conductivity.
- f. Click **OK**.
- g. Click **Apply**.

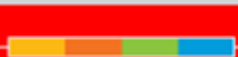
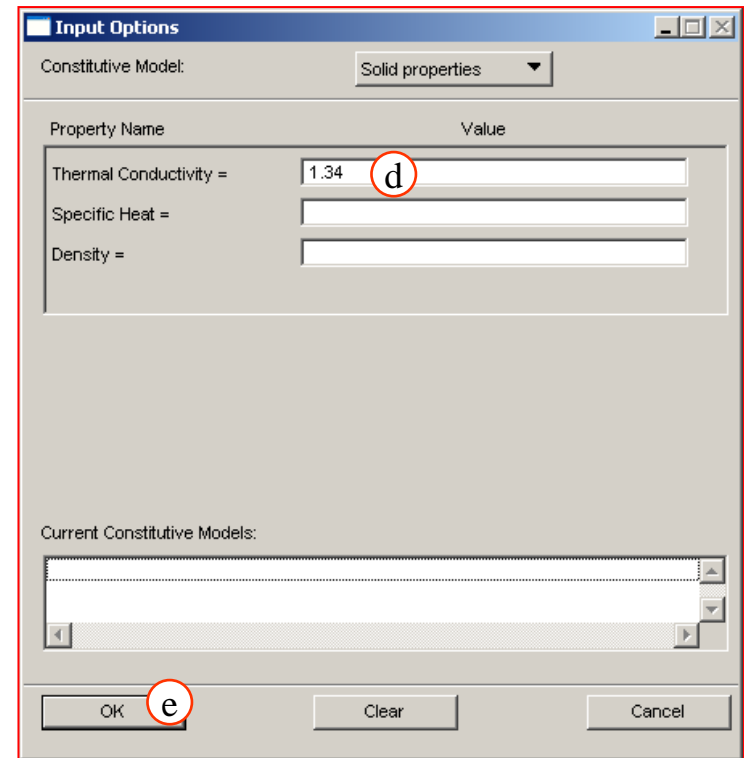
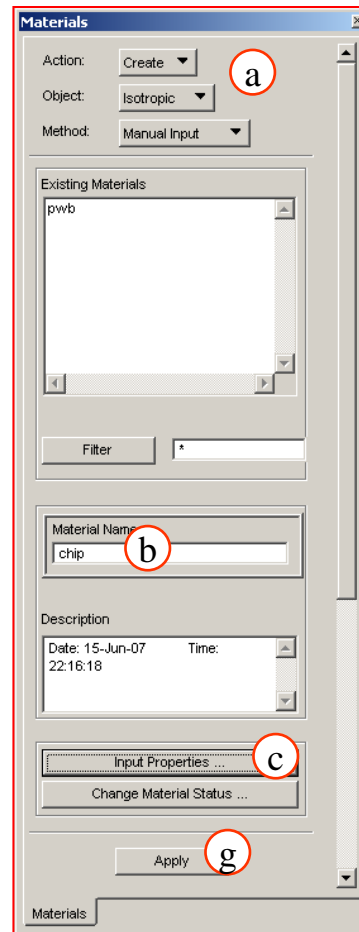


Step 5: Specify Materials (Cont.)

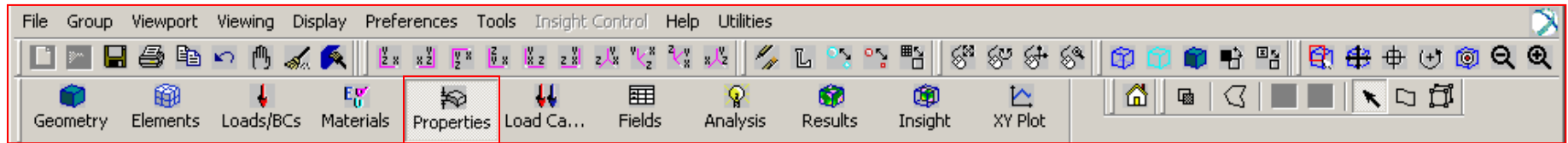


Create an isotropic material property for the chip:

- a. **Materials : Create / Isotropic / Manual Input.**
- b. Enter **chip** for Material Name.
- c. Click on **Input Properties**.
- d. Enter **1.34** for Thermal Conductivity.
- e. Click **OK**.
- f. Click **Apply**.

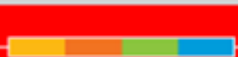
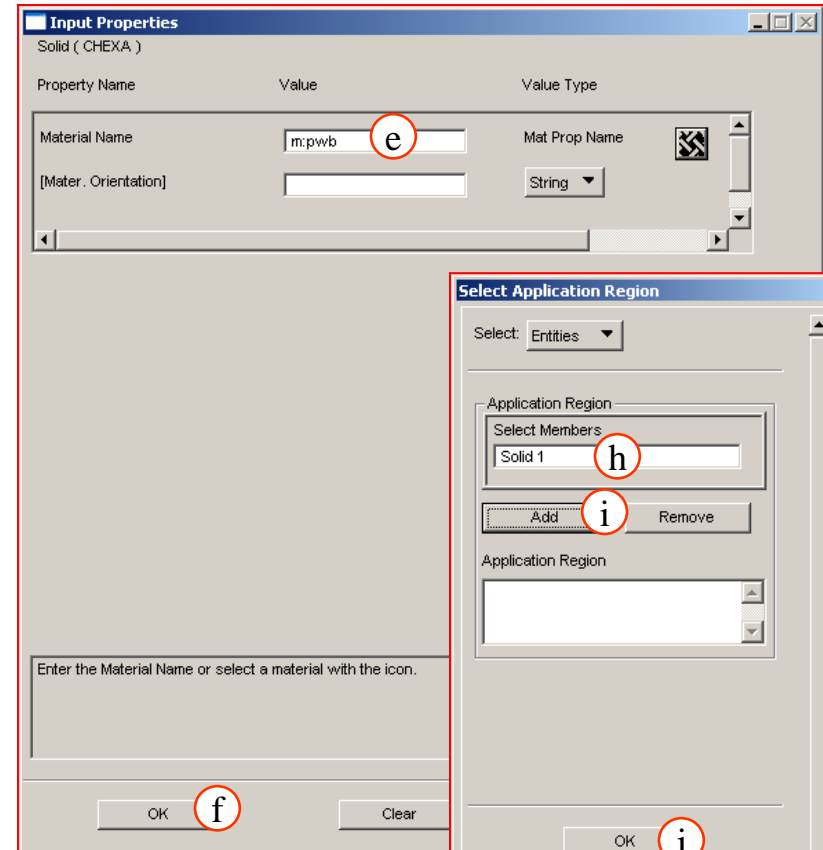
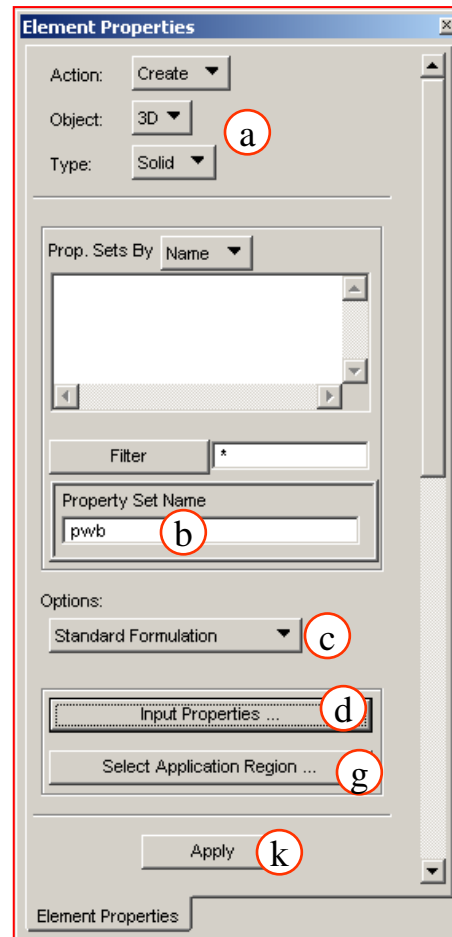


Step 6: Define Element Properties

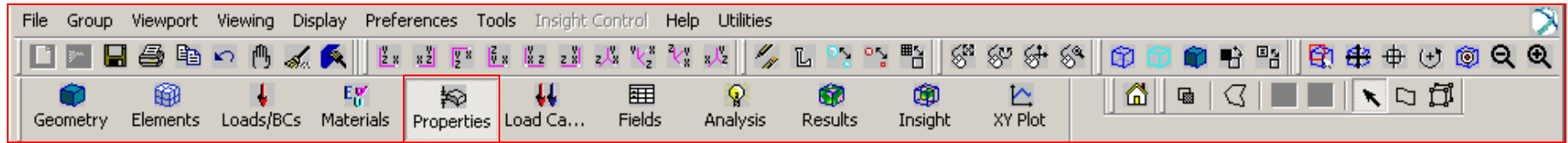


Create 3D solid element properties for the pcb:

- a. **Properties : Create / 3D / Solid.**
- b. Enter **pwb** for Property Set Name.
- c. Select **Standard Formulation** for Option(s).
- d. Click on **Input Properties**.
- e. Click in Material Name box and select **pwb** under Material Property Sets.
- f. Click **OK**.
- g. Click on **Select Application Region**.
- h. Enter **Solid 1** for Select Members.
- i. Click **Add**.
- j. Click **OK**.
- k. Click **Apply**.

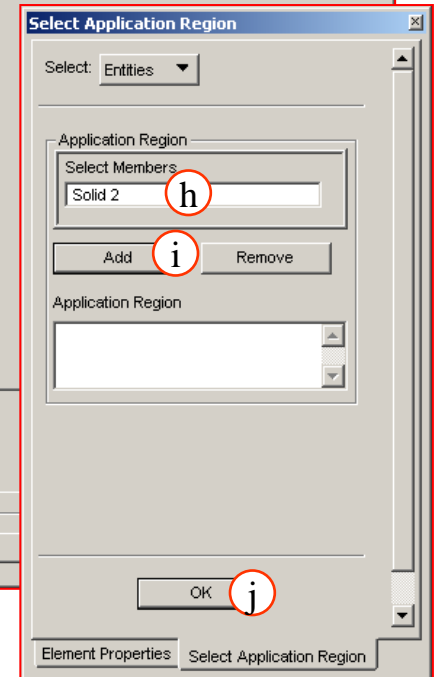
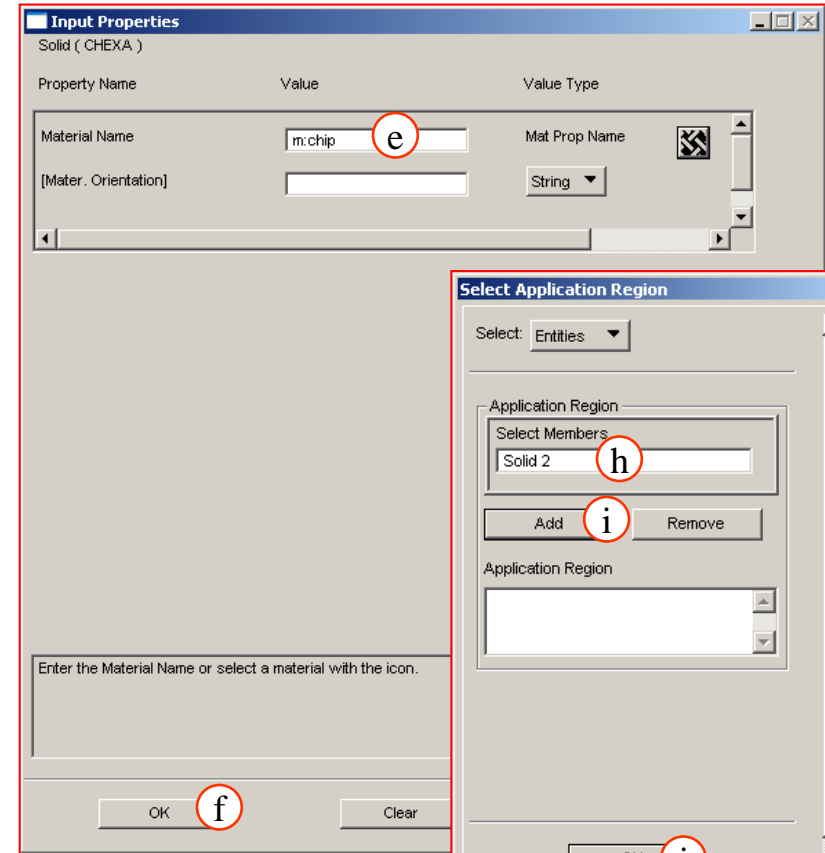
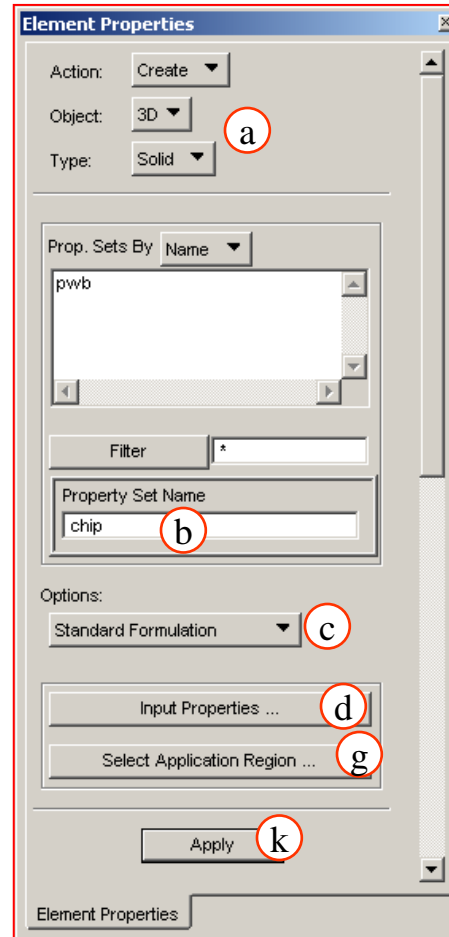


Step 6: Define Element Properties (Cont.)

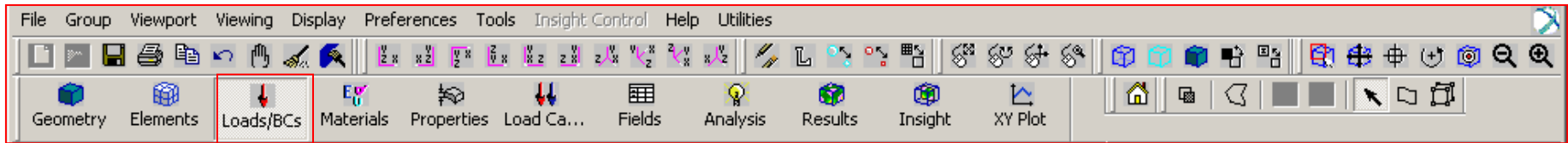


Create 3D solid element properties for the chip:

- a. **Properties: Create / 3D / Solid.**
- b. Enter **chip** for Property Set Name.
- c. Select **Standard Formulation** for Options.
- d. Click on **Input Properties**.
- e. Click in the Material Name box and select **chip** under Material Property Sets.
- f. Click **OK**.
- g. Click on **Select Application Region**.
- h. Enter **Solid 2** for Select Members.
- i. Click **Add**.
- j. Click **OK**.
- k. Click **Apply**.



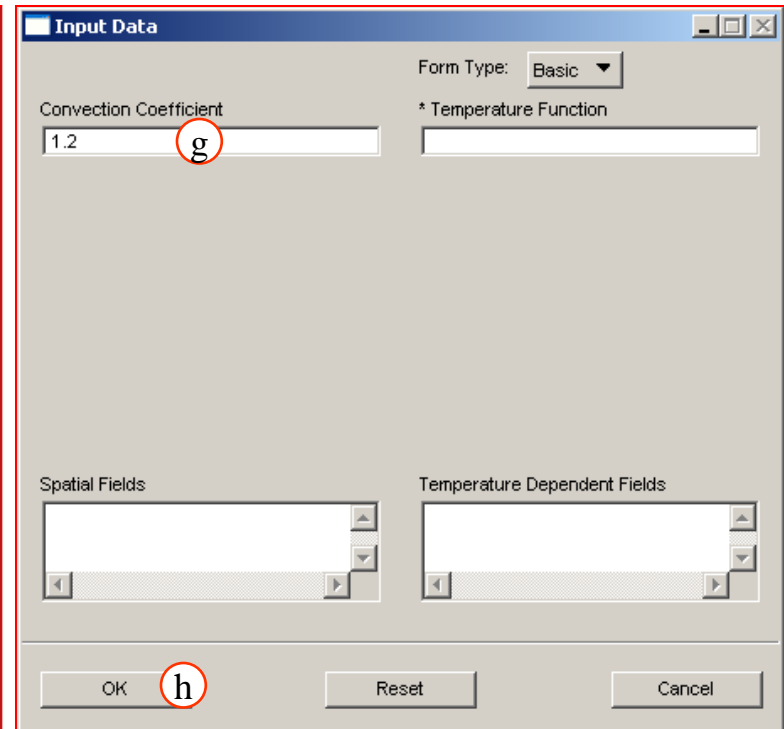
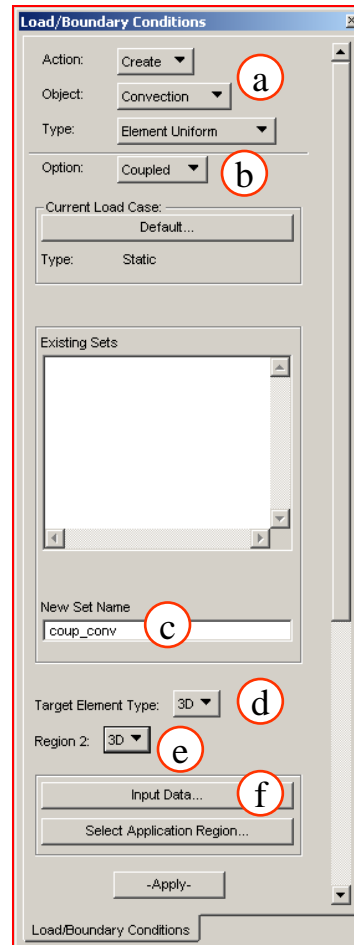
Step 7: Convection Between the two Solid Meshes



Set up a coupled convection loading between the two solids.

Loads/BCs : Create / Convection / Element Uniform.

- a. Select **Coupled** for Option.
- b. Enter **coup_conv** for New Set Name.
- c. Select **3D** for Target Element Type.
- d. Select **3D** for Region2.
- e. Click on **Input Data**.
- f. Enter **1.2** for Convection Coefficient.
- g. Click **OK**.

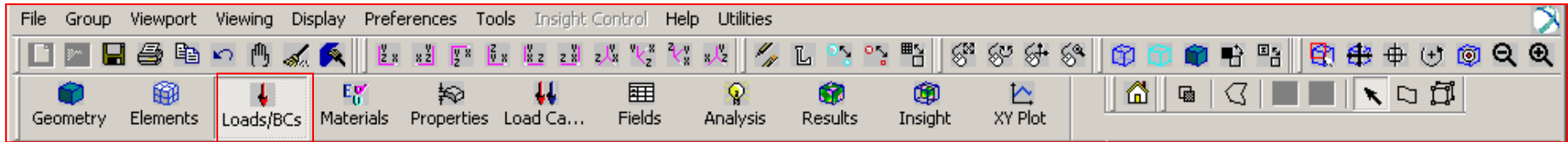


NAS104, Workshop 7, September 2008
Copyright © 2008 MSC Software Corporation

WS7-15

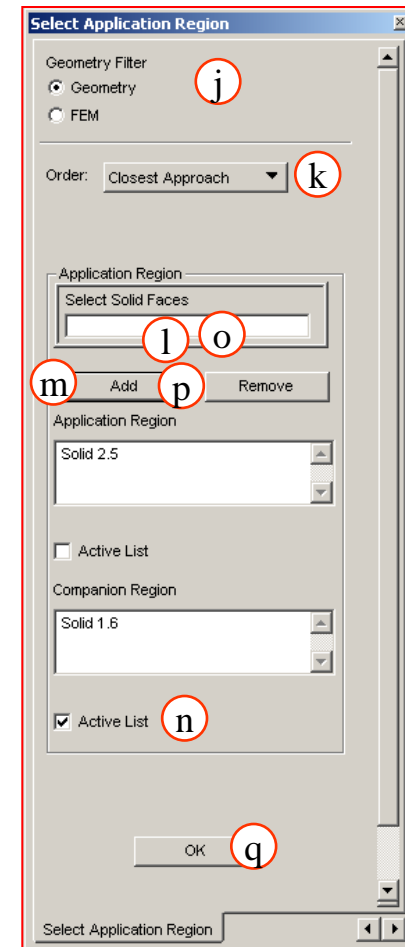
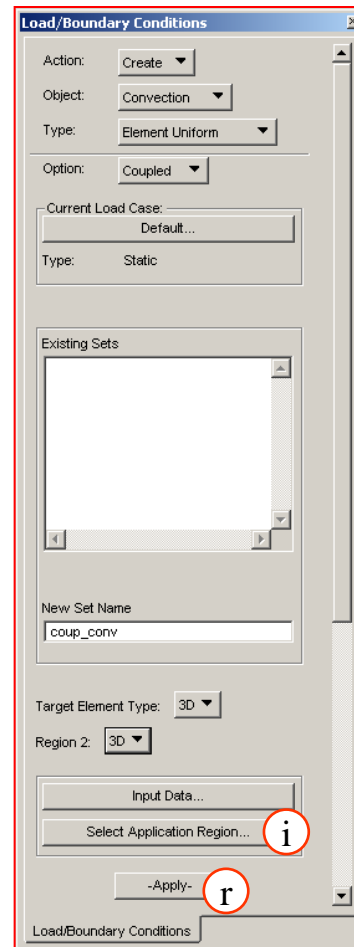


Step 7: Convection Between the two Solid Meshes (Cont.)

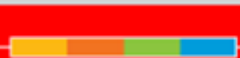
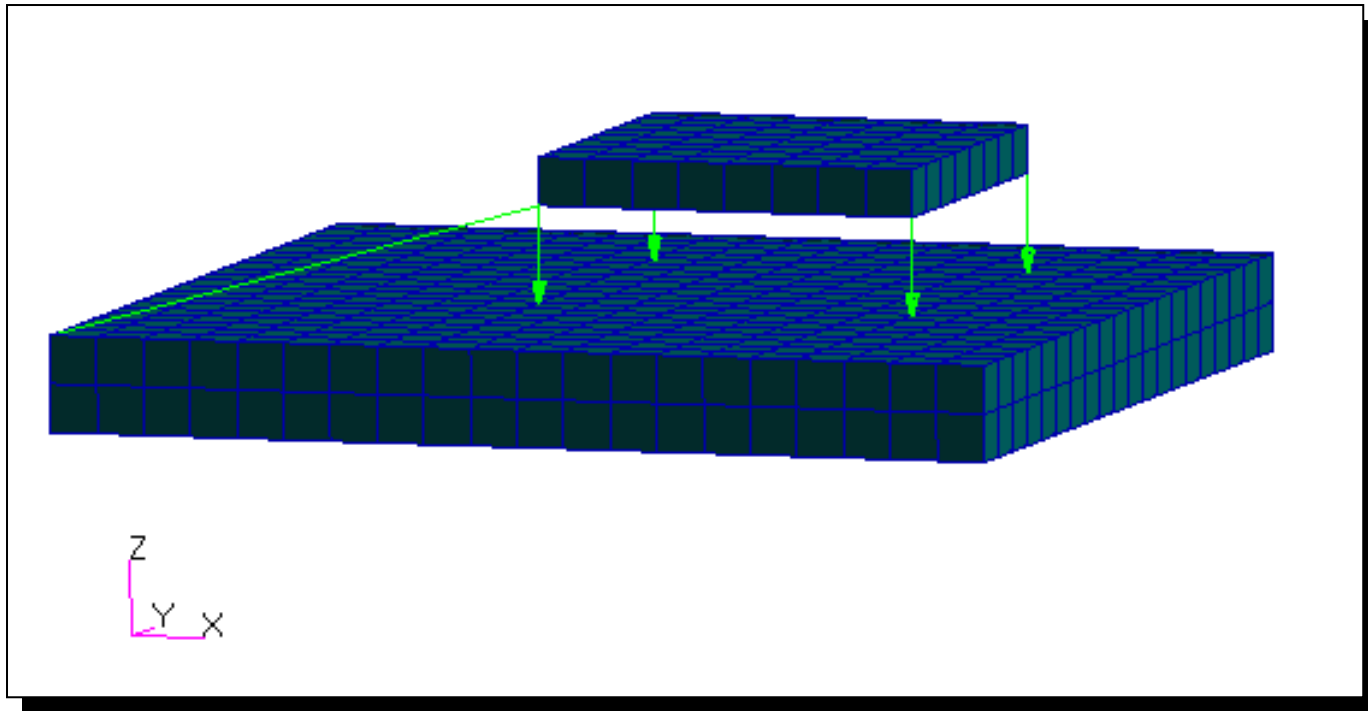


Set up a coupled convection loading between the two solids.
(Cont.)

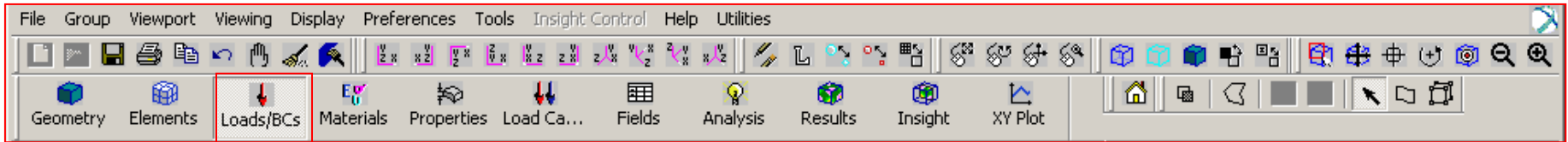
- i. Click on **Select Application Region**.
- j. Click **Geometry** for Geometry Filter.
- k. Select **Closest Approach** for Order.
- l. Enter **Solid 2.5** for Select Solid Faces.
- m. Click **Add**.
- n. Select **Active List** check box under Companion Region.
- o. Enter **Solid 1.6** for Select Solid Faces.
- p. Click **Add**.
- q. Click **OK**.
- r. Click **Apply**.



Step 7: Convection Between the two Solid Meshes (Cont.)

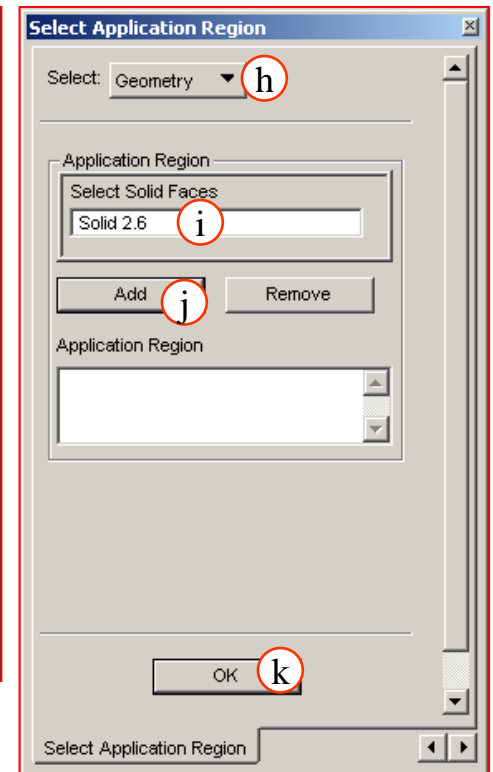
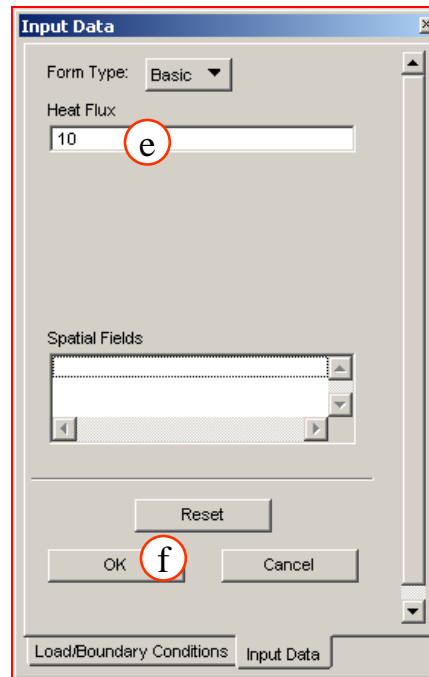
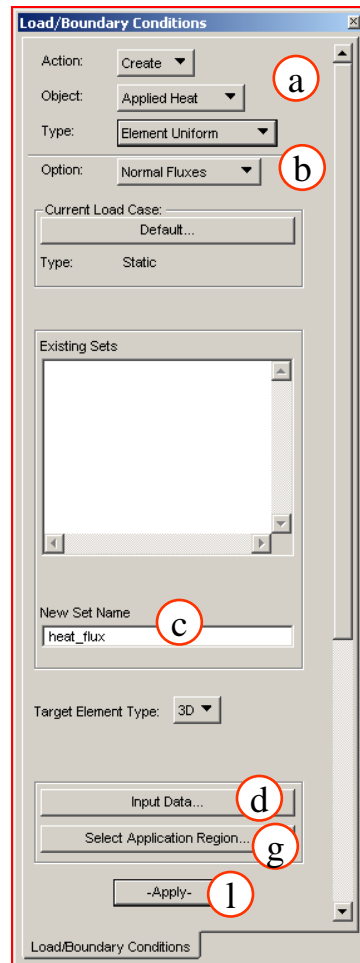


Step 8: Apply a Heat Flux

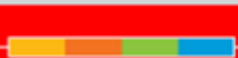
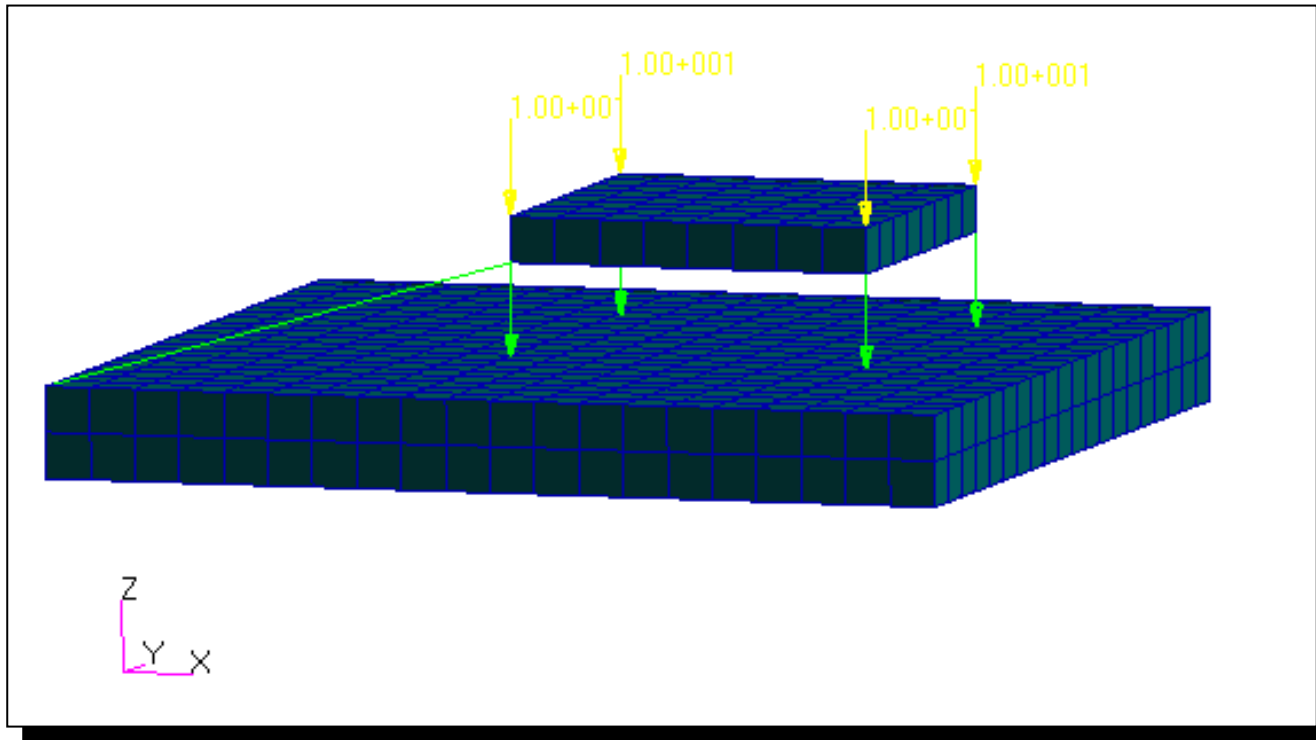


Apply a Heat Flux on the top surface of the chip:

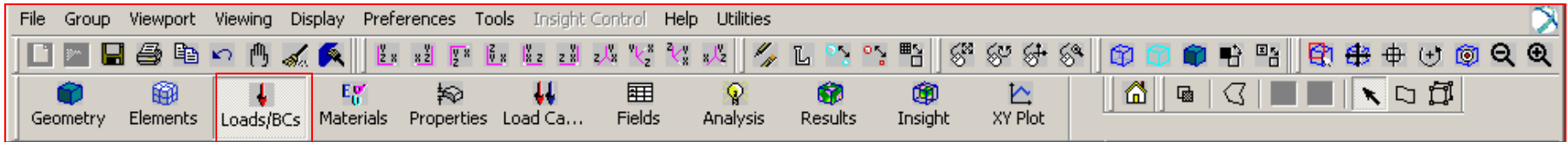
- a. **Loads/BCs : Create / Applied Heat / Element Uniform.**
- b. Select **Normal Fluxes** for Option.
- c. Enter **heat_flux** for New Set Name.
- d. Click on **Input Data**.
- e. Enter **10** for Heat Flux.
- f. Click **OK**.
- g. Click on **Select Application Region**.
- h. Select **Geometry** for Geometry Filter.
- i. Enter **Solid 2.6** for Select Solid Faces.
- j. Click **Add**.
- k. Click **OK**.
- l. Click **Apply**.



Step 8: Apply a Heat Flux (Cont.)

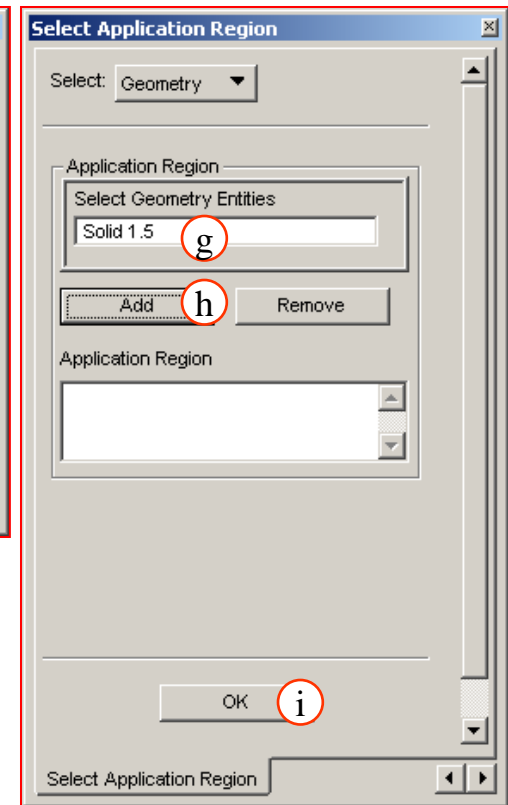
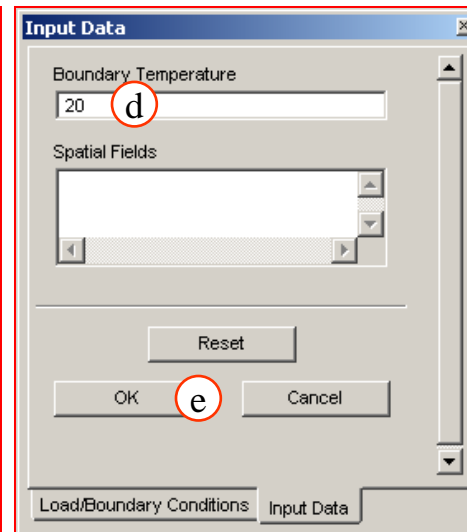
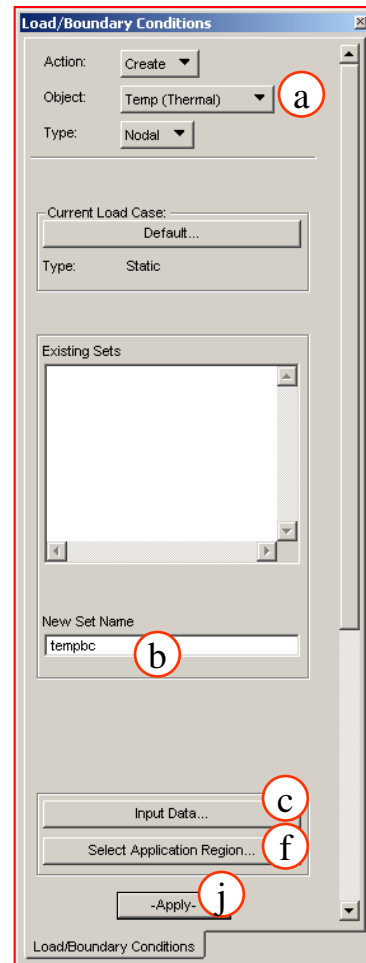


Step 9: Apply a Temperature Boundary Condition

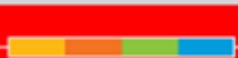
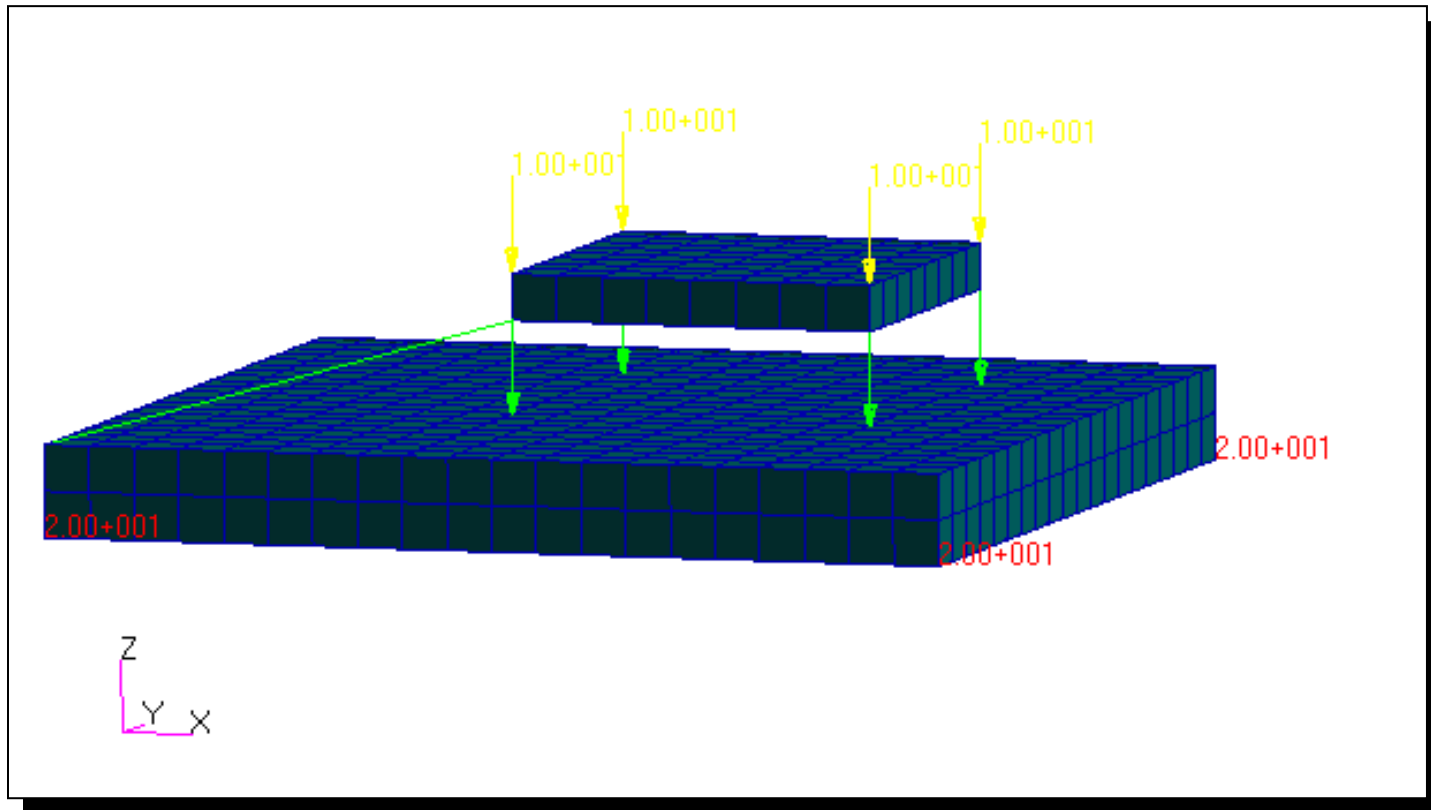


Apply a temperature boundary condition on the bottom of the PCB:

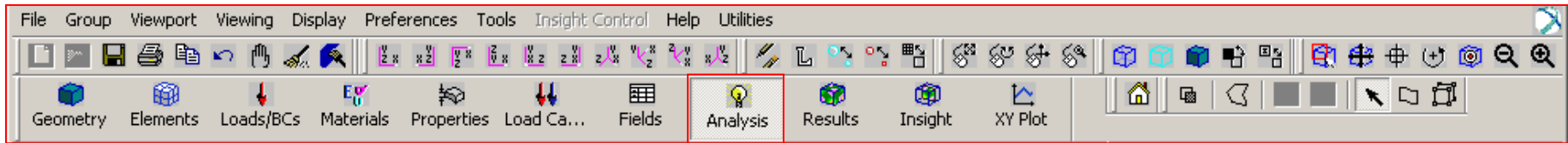
- a. **Loads/BCs : Create / Temp(Thermal) / Nodal.**
- b. Enter **tempbc** for New Set Name.
- c. Click on **Input Data.**
- d. Enter **20** for Boundary Temperature.
- e. Click **OK.**
- f. Click on **Select Application Region.**
- g. Enter **Solid 1.5** for Select Geometry Entities.
- h. Click **Add.**
- i. Click **OK.**
- j. Click **Apply.**



Step 9: Apply a Temperature Boundary Condition (Cont.)

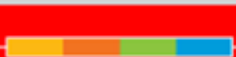
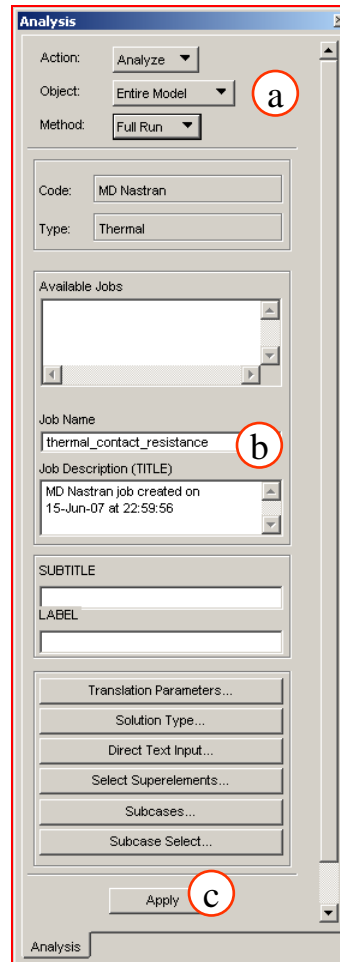


Step 10: Perform the Steady-state Thermal Analysis

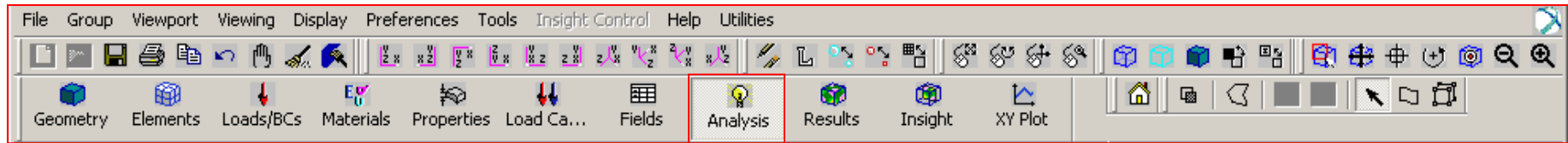


Perform the analysis:

- a. **Analysis : Analyze / Entire Model / Full Run.**
- b. Enter **thermal_contact_resistance** for the Job Name.
- c. Click **Apply**.

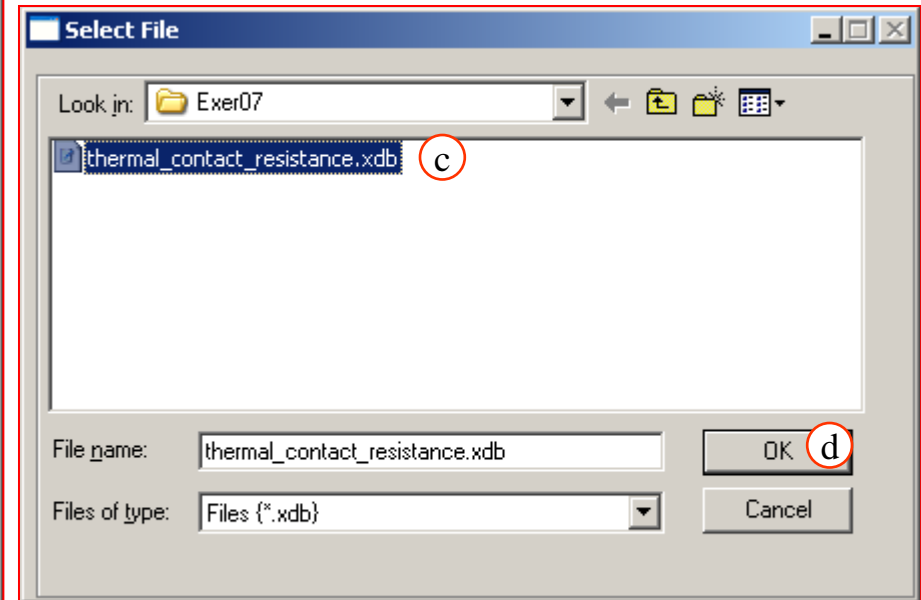
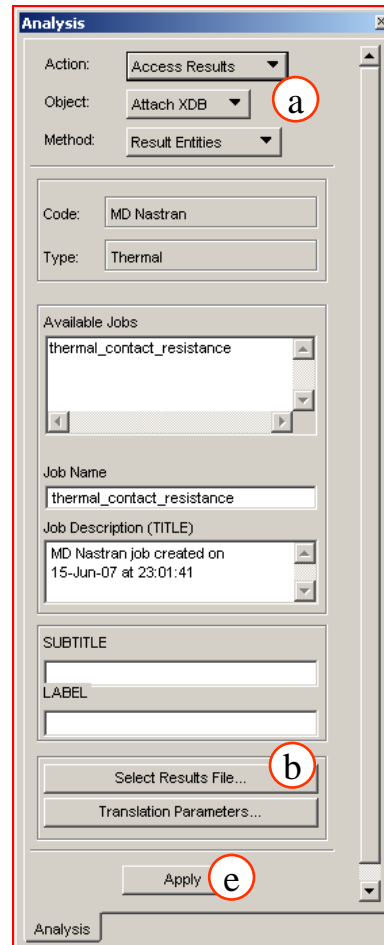


Step 11: Attach the Results File

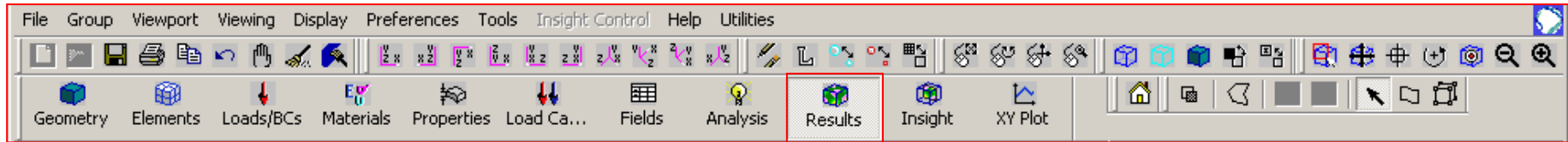


Attach the XDB file:

- a. **Analysis : Access Results / Attach XDB / Result Entities.**
- b. Click on **Select Results File.**
- c. Select **thermal_contact_resistance.xdb** for the File name.
- d. Click **OK.**
- e. Click **Apply.**



Step 12: Display the Temperature Results

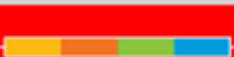
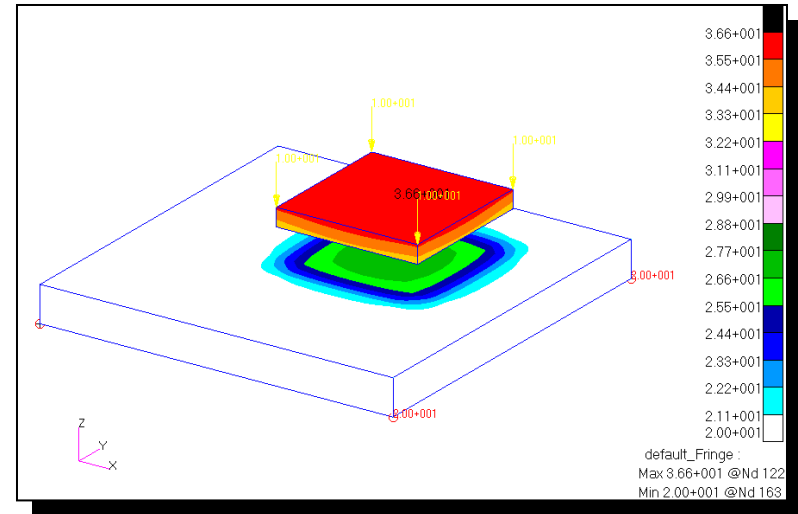
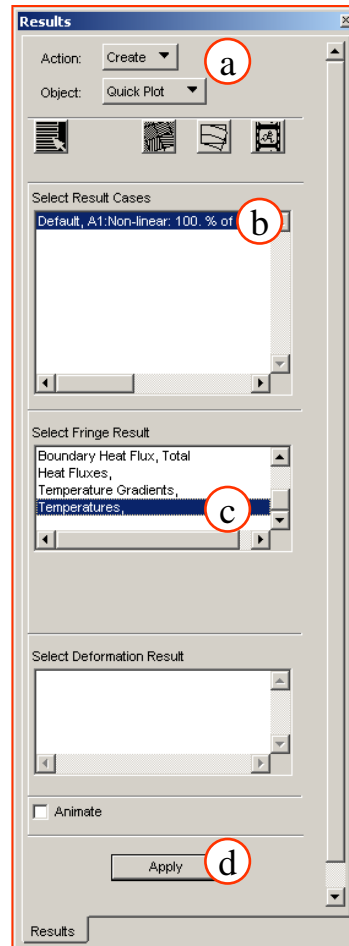


Display the results:

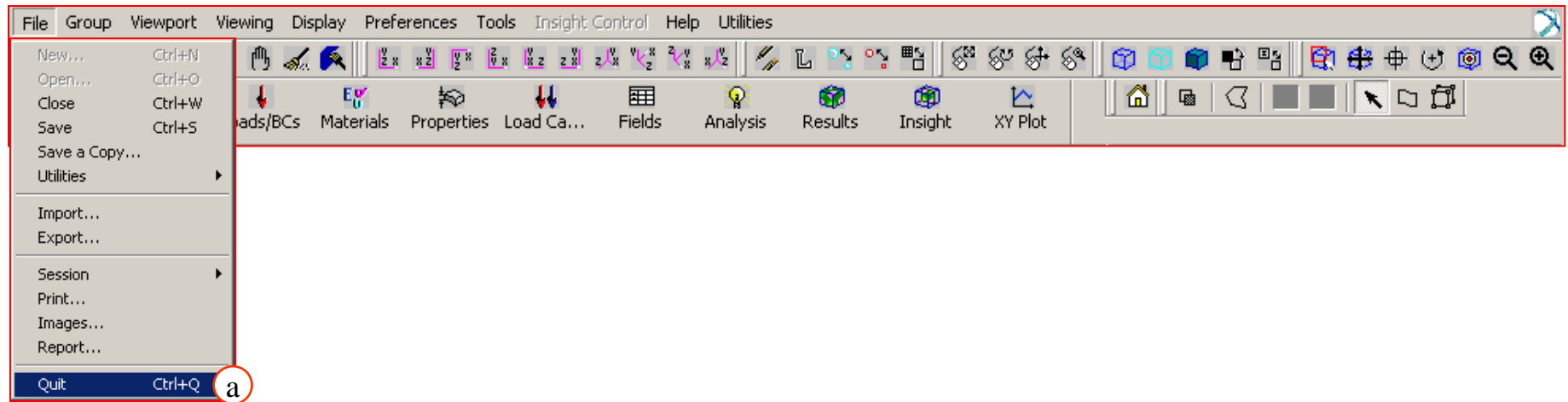
- Results : Create / Quick Plot.**
- Select **Default, A1...** for Select Result Cases.
- Select **Temperatures** for Select Fringe Result.
- Click **Apply**.

Post Workshop questions:

- The meshes of the two solids did match in this case. Would it still work on dissimilar meshes?
- What was the application region of the coupled convection? Did the convection transfer heat across the entire face of this application region?
- How would a user use this loading to interpolate a temperature distribution from a CFD analysis to a heat transfer analysis? In a future workshop, this heat distribution is made into an applied structural load to calculate thermal expansion stresses.



Step 13: Quit MD Patran



Quit MD Patran:

- a. File / Quit.

