NASA/CR—2001-211294

User's Manual and Final Report for HOT-SMAC GUI Development

Phil Yarrington
Collier Research and Development Corporation, Hampton, Virginia

November 2001

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA Access Help Desk at 301–621–0134

- Telephone the NASA Access Help Desk at 301–621–0390

- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076

NASA/CR—2001-211294

User's Manual and Final Report for HOT-SMAC GUI Development

Phil Yarrington
Collier Research and Development Corporation, Hampton, Virginia

Prepared under Contract NAS3–00159

National Aeronautics and
Space Administration

Glenn Research Center

November 2001

Trade names or manufacturers' names are used in this report for identification only. This usage does not constitute an official endorsement, either expressed or implied, by the National Aeronautics and Space Administration.

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100

Available electronically at http://gltrs.grc.nasa.gov/GLTRS

Table of Contents:

Section 1: Getting Started with the HOT-SMAC Interface 1
- Setup Tab 3
- Geometry Tab 4
- Materials Tab 5
- Thermal Boundary Conditions Tab 6
- Mechanical Boundary Conditions Tab 7
- Analysis Options Tab 8
- Results Tab 10
- Changing the Model 12

Section 2: Detailed Example Tutorial 15
- Problem Description 17
- Problem Setup 18
- Material Properties 18
- Geometry 19
- Material / Window Assignment 21
- Thermal Boundary Conditions 23
- Mechanical Boundary Conditions 25
- Analysis Options 27
- Running The Analysis 28

Section 3: HOT-SMAC Verification Example 31
- 8X24 Coarse Grid 33
- 26X74 Medium Grid 39

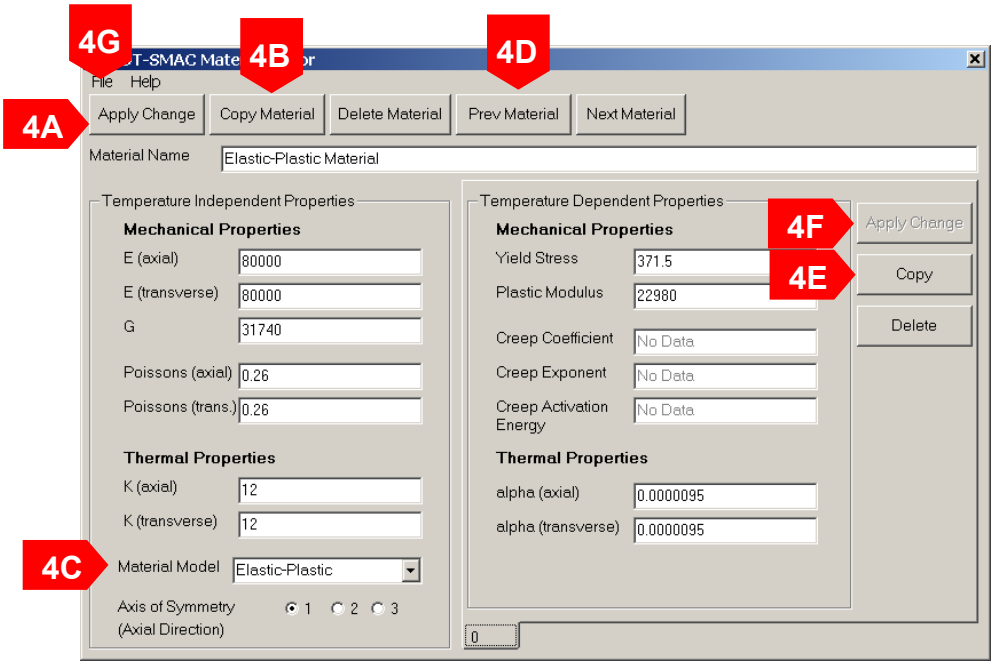
Section 4: HOT-SMAC – HyperSizer Integration Research 43

Section 1: Getting Started with the HOT-SMAC Interface

Getting Started with the HOT-SMAC Interface.

Setup Tab

1. Go to **Start|Programs|HOT-SMAC** to start the HOT-SMAC interface.
2. On the **Setup** tab, choose a grading scheme, either graded in one dimension or graded in two dimensions.
3. Choose a constitutive model. For this example, choose Elastic-Plastic (the default) as the constitutive model. Therefore, only elastic or elastic-plastic materials can be chosen for this analysis.
4. Notice in the Materials frame that there are currently zero phases (or materials) available. To set up materials, press the "Select/Define Materials" button.
   a. When this button is pressed, a default material called "New Material" is created and shown in the HOT-SMAC Material Editor form. Change the name of this material to "Elastic-Plastic Material" and press the <Enter> key. Press the "Apply Change" (**A**) button to save this change.
   b. To create a second material, press the "Copy Material" button (**B**) and



name the new material "Elastic". Change this material to an elastic-only material by selecting "Elastic" from the Material Model dropdown (**C**) and then press the "Apply Change" button. Notice that all properties except the elastic properties are disabled for this material.

   c. Switch back to Elastic-Plastic Material by pressing the "Prev Material" button (**D**).
   d. You can make this material temperature dependent by "copying" the current temperature dependent material properties and then changing the copied properties. To copy the material property, press the "Copy" button (**E**) in the Temperature Dependent Properties frame and then type 650 for

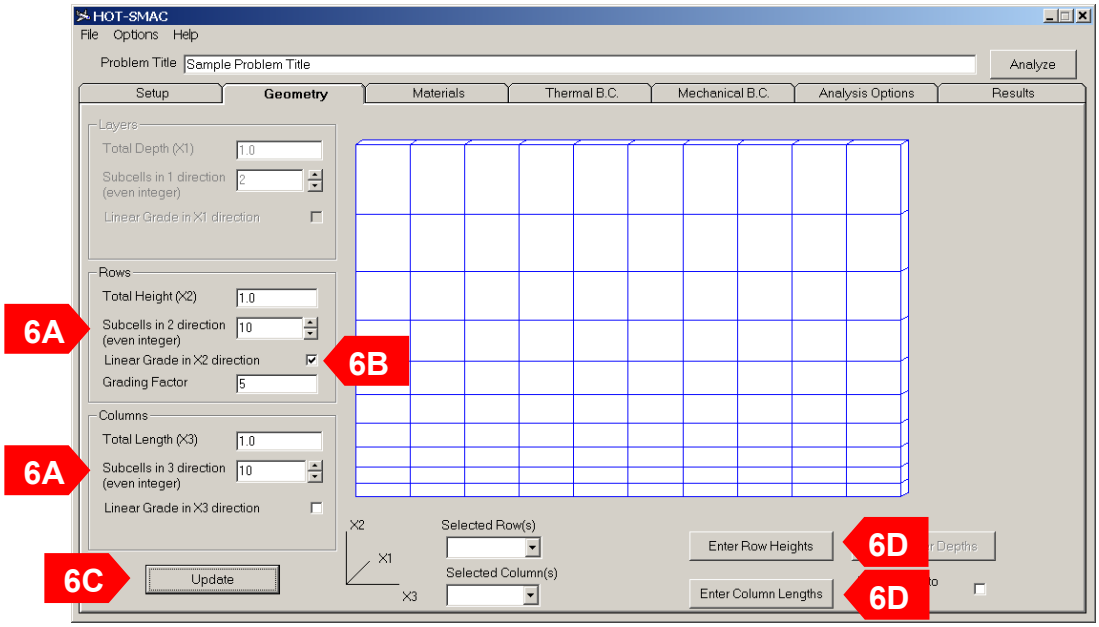
the new temperature. For the yield stress at 650 degrees, type 269.5 and for the Plastic Modulus, type 670. Press the "Apply Change" button (**F**) to apply these temperature dependent changes.

At this point, the materials only exist in the computers memory. You can export these materials to an ASCII file by selecting Export Materials from the File dropdown menu (**G**) and select a file name to export the materials to. To import these materials into a new HOT-SMAC model, select "Import Materials" from this same menu.

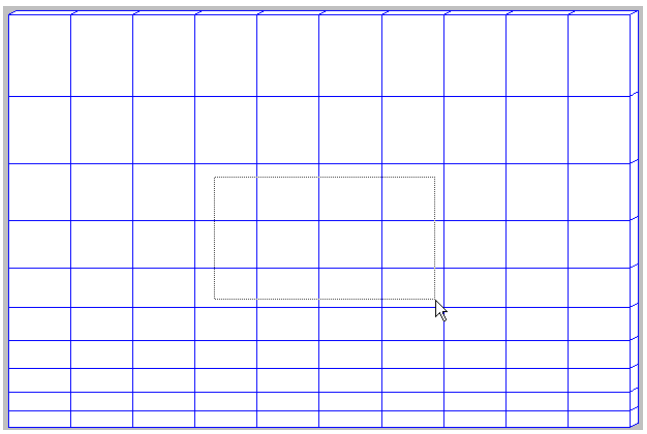
e. Close the material editor form.

Geometry Tab

5. Click on the **Geometry** tab to change the HOT-SMAC problem geometry.



6. You can change the number of subcells in the 2 and 3 direction by adjusting the numbers at **A**. If you want to grade the spacing in a particular direction, check the "Linear Grade" checkbox (**B**) and enter a grading factor. The grading factor is the ratio of the last cell spacing to the first. If you enter a negative number, then the number is the ratio of the first cell spacing to the last. Enter 5 for the Factor as shown above and press the Update button (**C**). The figure should appear as shown here.

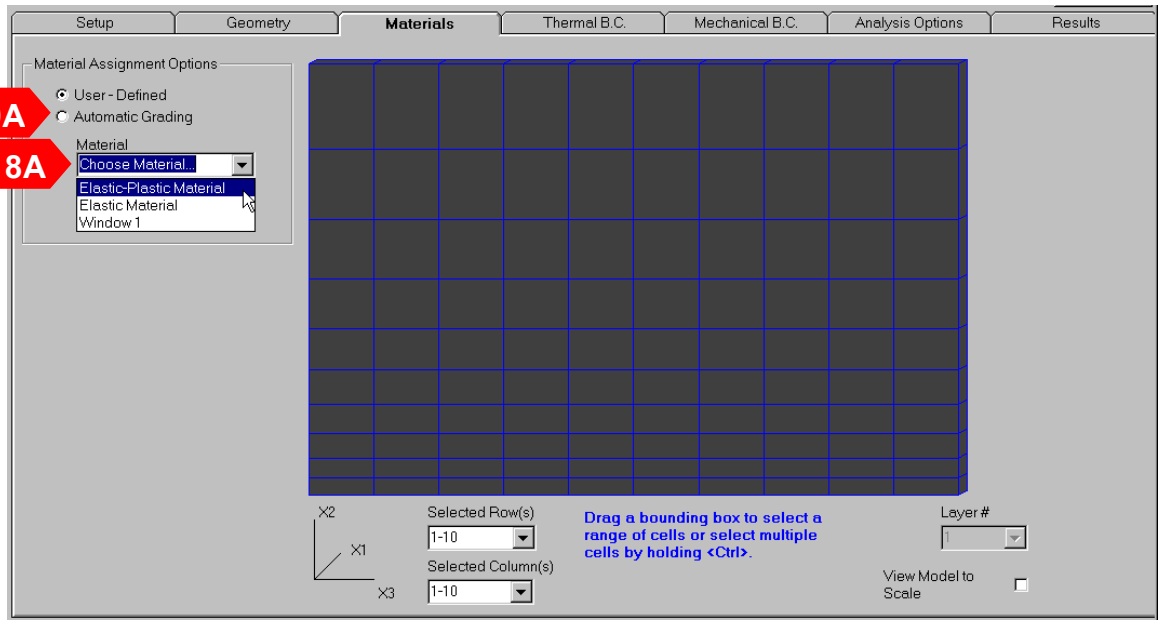


You can also select cells or ranges of cells on the figure to apply changes manually by placing the mouse cursor over the figure and dragging a bounding box. All cells within the bounding box will be selected and turn gray. At this point any changes made to the height, width or linear grading will be applied only to the selected cells. You can

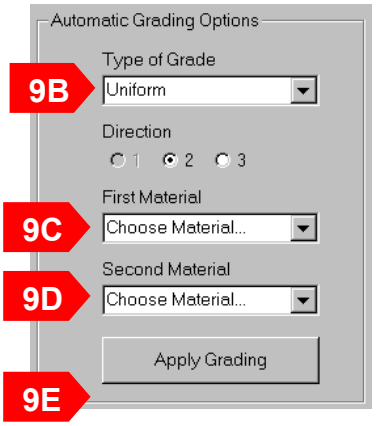
also make changes to individual row heights and column widths by pressing the "Enter Row Heights" and "Enter Column Widths" buttons (**D**).

Materials Tab

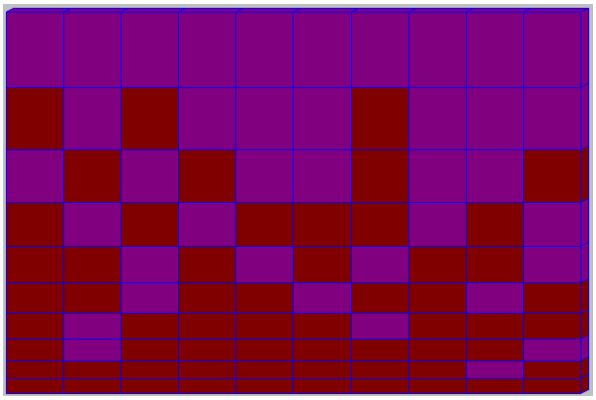
7. The next step is to assign materials to the cells of the HOT-SMAC model. Click the **Materials** tab.



You can assign materials for a cell, a range of cells, or you can let the software automatically grade the material over a range from one material to another.

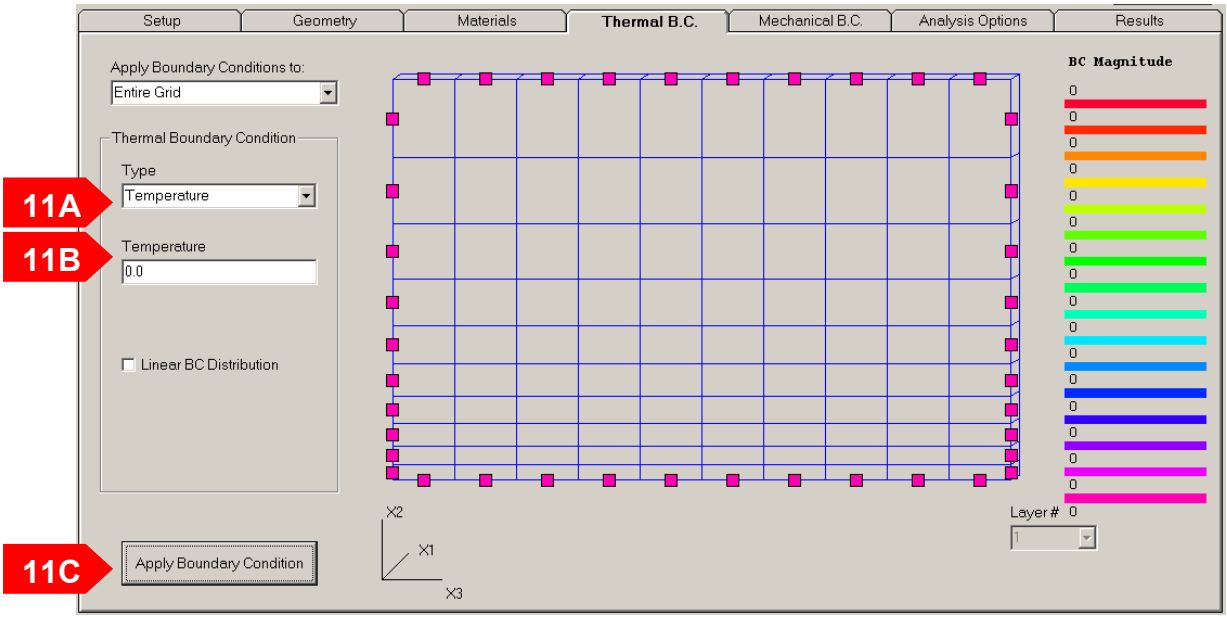


8. Select all of the cells in the model by dragging a bounding box through all cells and releasing the mouse button. From the "Material" dropdown list (**A**), select the Elastic-Plastic Material. Press the "Apply Material" button. Now all of the cells in the model have been assigned the Elastic-Plastic Material.
9. To automatically grade the cells from top to the bottom, check the "Automatic Grading" radio button (**A**). A second frame called "Automatic Grading Options" will appear. Change the type of grading (**B**) to "Linear". For the First Material (**C**), choose "Elastic-Plastic Material" and for the Second Material (**D**), choose "Elastic Material." Now select all of the cells in the model again (by drawing a bounding box) and press the "Apply Grading" button (**E**). The grading has a certain amount of randomness to it, but it should look similar to the figure shown here.



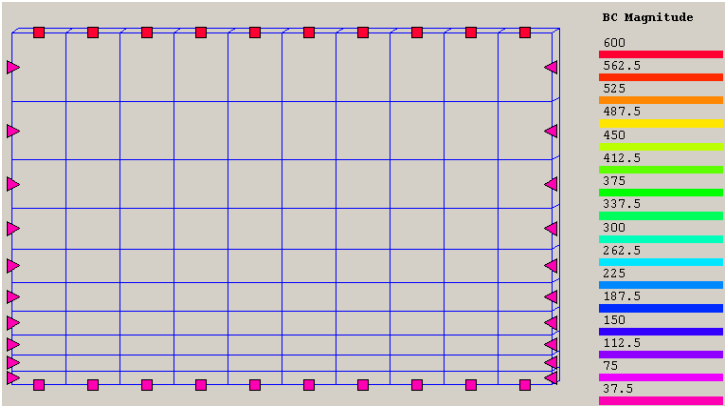
Thermal Boundary Conditions Tab

10. The next step is to apply the thermal boundary conditions to the model. Click the **Thermal BC** tab.



This tab shows the boundary cells of the model with each thermal boundary condition shown as a symbol on the edge of the boundary cell. The default boundary condition for each cell is a uniform temperature (shown with the square symbol) of 0 degrees. You can apply a flux boundary or a convective boundary by selecting cells and changing the "Type" and "amplitude" of the boundary conditions at **A** and **B**. Keep in mind that at least one cell must be a non-heat flux boundary in order to prevent singularities. For this example, we will change the side boundary cells to zero heat flux, the temperature along the top to 600 and the temperature along the bottom to 0.

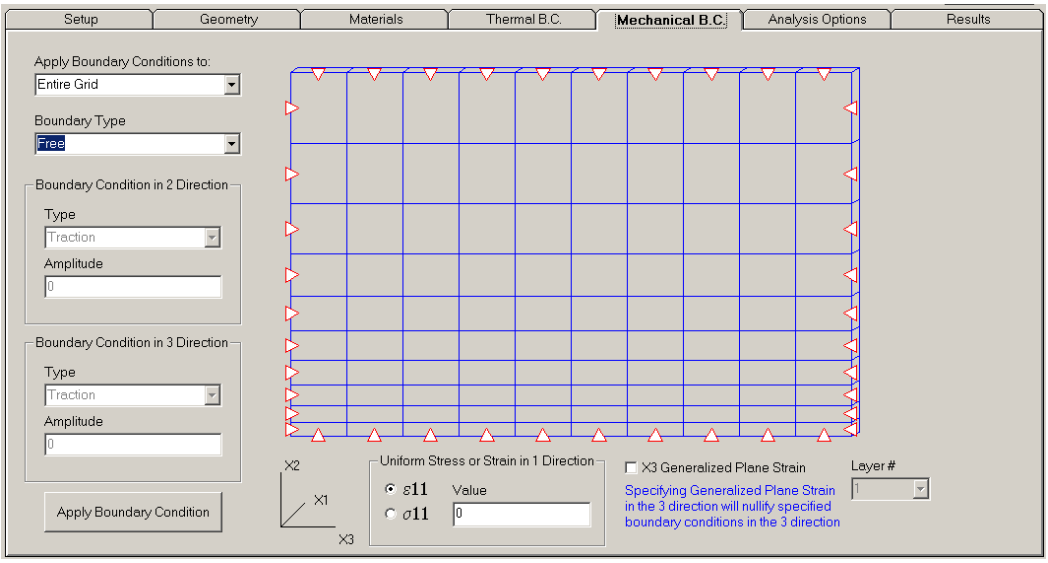
11. Select the cells along the top boundary by dragging a bounding box around those cells. Change the Thermal Boundary Condition type (**A**) to "Temperature" and enter 600 for the Amplitude (**B**). Press the Apply Boundary Condition button (**C**).
12. Select the cells along the right boundary, change the type to "Flux", and set the Amplitude to 0. Press the Apply Boundary Condition button. Repeat for the left boundary cells.



You can also apply a boundary condition with a linear grade from one end to the other by checking the "Linear BC Distribution" checkbox.

Mechanical Boundary Conditions Tab

13. The next step is to apply the mechanical boundary condition. Click the **Mechanical BC** tab.

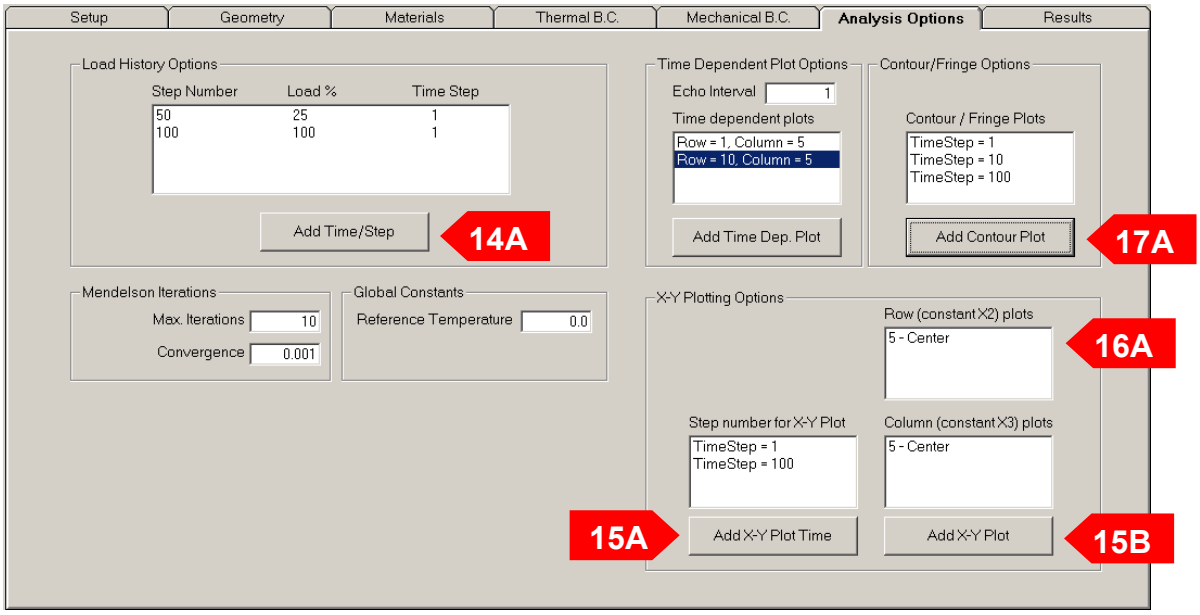


The default mechanical boundary condition is a fixed (zero displacement) boundary on all cells. You can modify the boundary conditions for each cell or a range of cells just as with the thermal boundary conditions. There are three preset boundary condition types available for mechanical boundaries. Free (zero traction in each direction), roller (zero normal displacement), and pinned (zero displacement in both directions). You can also choose a "Custom" boundary condition with any combination of surface traction or displacement in the two directions.

If the mechanical boundary conditions are changed, they must be set to ensure that there is no rigid body motion to prevent singularities. For this example, leave the boundary conditions as their default value of fixed on all boundaries.

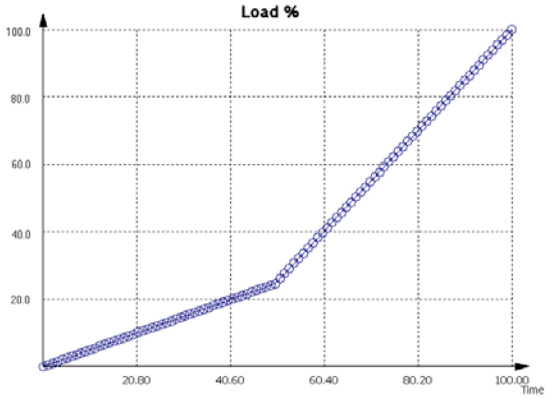
Analysis Options Tab

14. At this point, the physical problem is fully defined. Before submitting the HOT-SMAC analysis, we must set up the parameters of the run. To do this, click the **Analysis Options** tab.



The Analysis Options tab is used to specify the number of analysis integration steps, the Mendelson iterations for elastic-plastic analysis, the options for plotting results, and some global parameters such as global reference temperature.

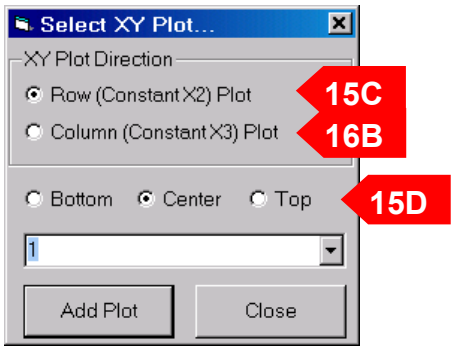
For elastic-plastic problems, the load step is varied linearly from 0 to a percentage of the full load as shown in the Load History frame. The number of load steps and % of load at each step are dictated by pressing the Add Time/Step button at **A**. For example, in the case shown here, the load will be applied from 0 to 25% of full load from load step 0 to 50, and from 25-100% of full load from load step 50 to 100. For time-dependent problems (e.g. creep problems), the time step is also dictated in this frame.



X-Y column and row plots are generated at the Time/Load steps entered in the "Step Number for X-Y plot" window.

15. Click the"Add X-Y Plot Time" button at **A** and enter 100 so that the software will generate X-Y plots at load step 100. Press the "Add X-Y Plot" button (**B**) on the Analysis Options tab. A new form will appear labeled "Select XY Plot…"

On this form, you can specify to plot either row plots or column plots on the HOT-SMAC model. The dropdown list at **C** is used to select the row or column on which to plot and the radio button at **D** is used to dictate whether to plot the top, bottom or center for row plots, or the right, left or center for column plots.



16. Select Row 5 from the dropdown combo and then press the "Add Plot" button. The created plot will appear in the "Row (constant X2) Plots" list box (**A**) on the Analysis Options tab. Press the "Add X-Y Plot" button again; click the "Column" radio button (**B**), and select column 5 from the dropdown list to add a plot for the center of column 5.

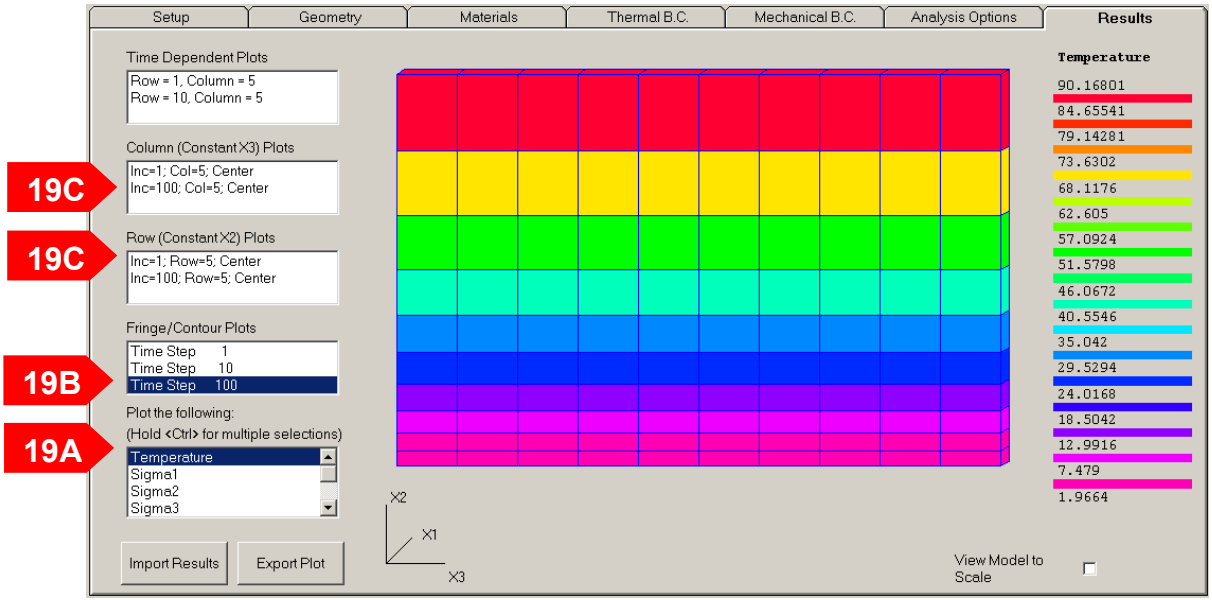
17. The final step is to specify the integration steps at which to plot contours. Press the "Add Contour Plot" button at **A**. In the "Select Time Step…" dialog that appears, enter 10 and press OK. Repeat this procedure to add step 100.

We are now ready to submit the HOT-SMAC analysis. Press the "Analyze" button in the upper right hand corner of the Main HOT-SMAC form. The HOT-SMAC analysis will begin and a "Cancel" button will appear over the main HOT-SMAC window. To stop the analysis, press the Cancel button. The HOT-SMAC interface will be locked out until the analysis is complete.

After the analysis is complete, if there was a problem running the HOT-SMAC analysis, a message box will give you the choice of reviewing the ASCII output file. If the analysis was successful, there will be no message. If there were no problems with this analysis, we can now view the analysis results graphically on the "Results" tab.

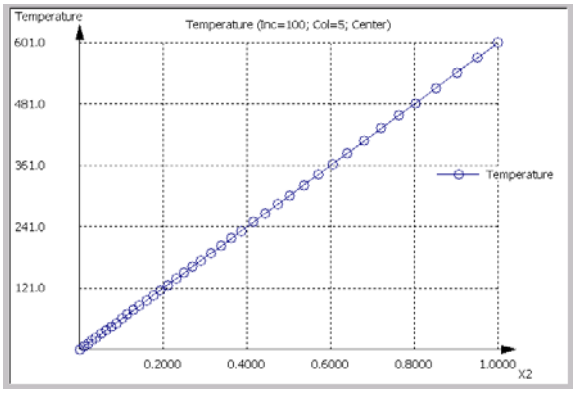
Results Tab

18. Click the **Results** tab. When this tab is first clicked, no results are selected therefore it is blank.
19. Select a quantity to plot by selecting the quantity "Temperature" in the list box at **A**. Next, select the step at which you want to plot results (**B**). You can plot contours at Step 10 or Step 100 according to the choices that were selected on the Analysis Options tab.



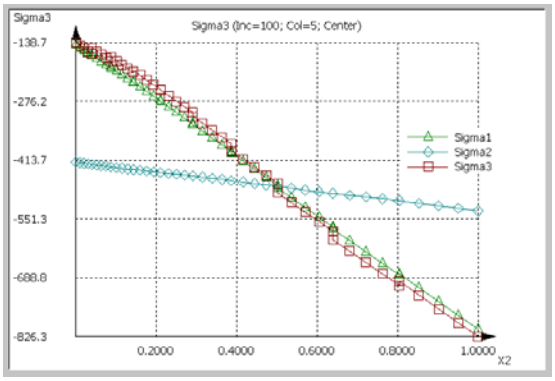
You can also display row and column plots by selecting a quantity at **A** and selecting a row or column plot at **C**.

20. Plot the temperature along the centerline column by selecting "Inc=100; Col=5, Center"



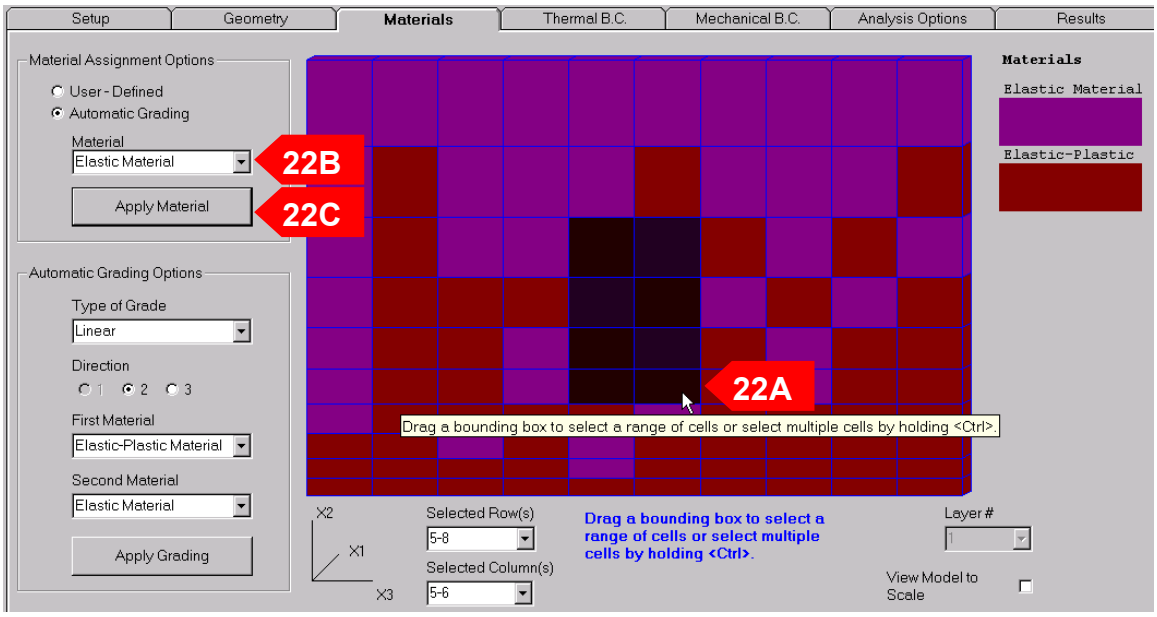
You can display multiple quantities on a single plot by pressing the <Ctrl> key while selecting quantities to plot.

21. Plot the centerline stresses in the 1, 2 and 3 directions simultaneously by first selecting Sigma1 and then while holding the <Ctrl> key, select Sigma2 and Sigma3. The resulting plot should look something like this:

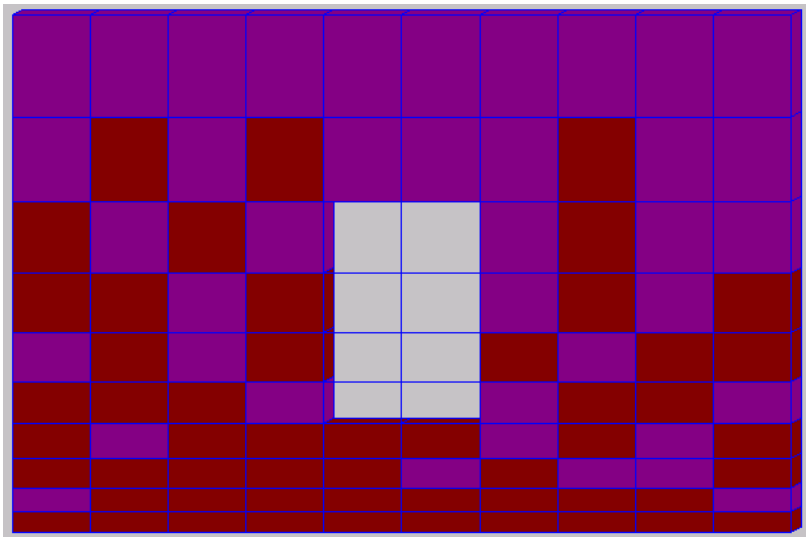


Changing the Model

After running the analysis, you can change the model and re-submit the analysis. We are now going to add a "window" or "hole" into the model and re-run the analysis.

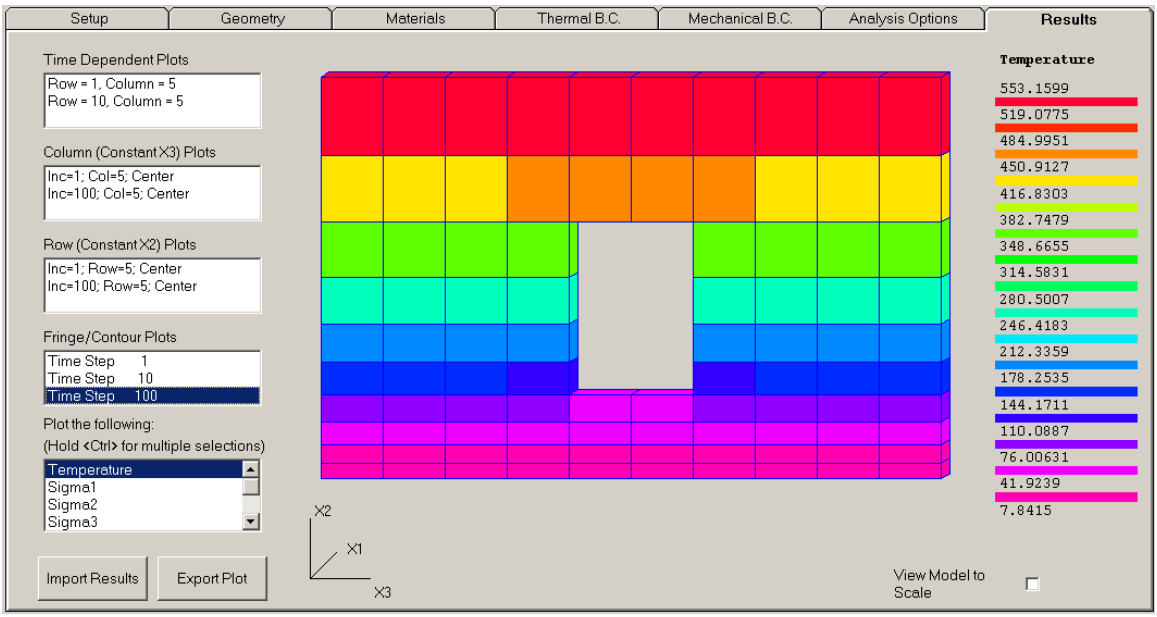


22. Go back to the **Materials** tab and select a range of cells by dragging a bounding box (**A**). From the Material dropdown (**B**), select "Window 1". Press the Apply Material button (**C**). A Window in the model should appear.



Each window that you define in the model has its own boundary condition assignments. These are specified on the **Thermal BC** and the **Mechanical BC** tabs.

23. Go to the **Thermal BC** tab. From the dropdown labeled, "Apply Boundary Conditions to:", select Window 1. The boundary cells of the Window defined in step 22 should be displayed. You can assign boundary conditions to these cells just as in steps 11 and 12. Change the boundary conditions on these cells to 0 flux boundaries.
24. Press the "Analyze" button and when the analysis completes, go back to the **Results** tab to view the results.


Setup
Geometry
Materials
Thermal B.C.
Mechanical B.C.
Analysis Options
Results
Time Dependent Plots
Row = 1, Column = 5
Row = 10, Column = 5
Column (Constant X3) Plots
Inc=1; Col=5; Center
Inc=100; Col=5; Center
Row (Constant X2) Plots
Inc=1; Row=5; Center
Inc=100; Row=5; Center
Fringe/Contour Plots
Time Step 1
Time Step 10
Time Step 100
Plot the following:
(Hold <Ctrl> for multiple selections)
Temperature
Sigma1
Sigma2
Sigma3
Import Results
Export Plot
X2
X1
X3
Temperature
553.1599
519.0775
484.9951
450.9127
416.8303
382.7479
348.6655
314.5831
280.5007
246.4183
212.3359
178.2535
144.1711
110.0887
76.00631
41.9239
7.8415
View Model to Scale

Section 2: Detailed Example Tutorial

Thermo-Elastic Analysis of an Internally Cooled Structure

Detailed Example Tutorial

The following example duplicates an example presented in NASA/TM 2001-210702, "Thermo-Elastic Analysis of Internally Cooled Structures Using a Higher Order Theory", in which a thermal barrier coated actively cooled panel is subjected to an intense flame heating boundary condition. First, the example will be worked step by step, and finally, the results of this analysis compared with the 2001 NASA/TM will be presented.

Problem Description

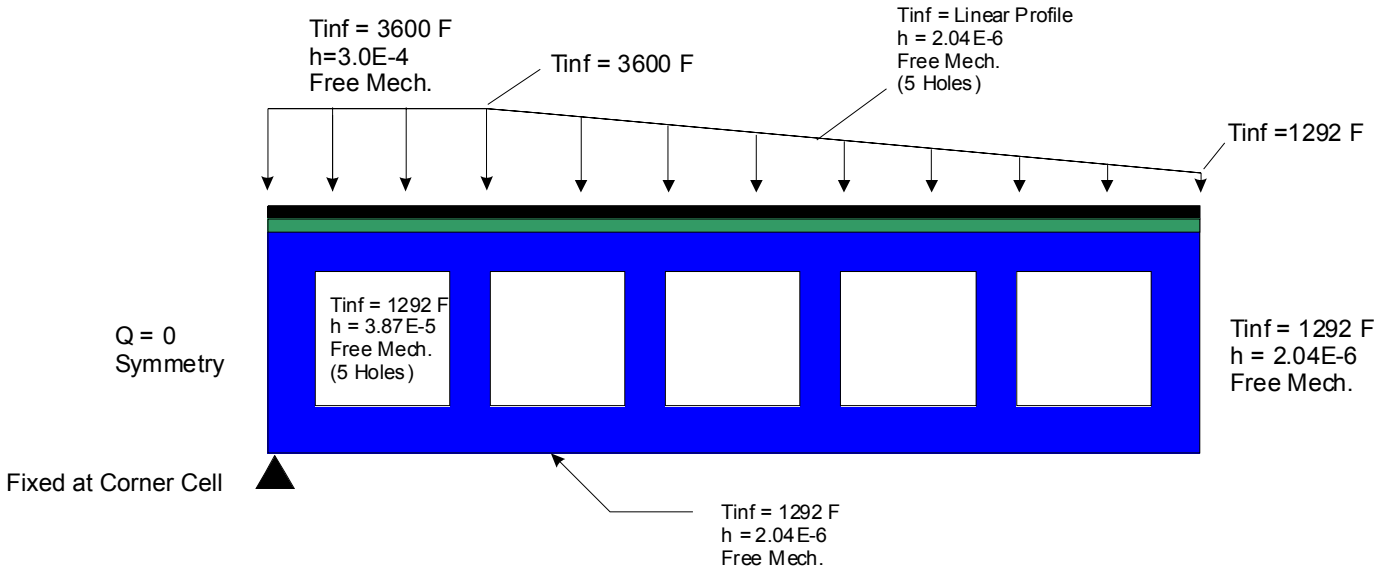


Table 1: Material Properties

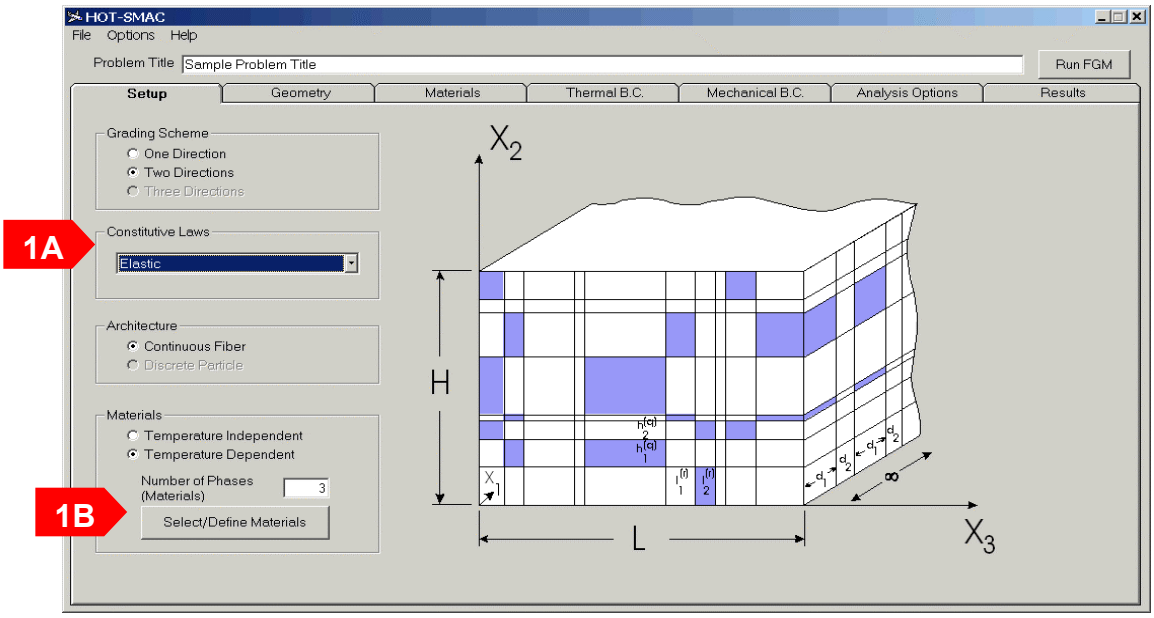
Material	E(psi)	ν	α (10^{-6}/°F)	K (Btu/in·s·°F)
Substrate: Silicon Nitride	4.35×10^{7}	0.22	1.83	4.01×10^{-4}
Bondcoat: Mullite	2.1×10^{7}	0.20	2.94	7.84×10^{-5}
Topcoat: Porous Zirconia	3.63×10^{2}	0.25	6.25	2.68×10^{-6}

Table 2: Dimensions

Dimension	Value (in.)
Total length	0.5
Total height	0.135
Hole length	0.07
Hole height	0.07
Distance between holes	0.025
Bondcoat thickness	0.005
Topcoat thickness	0.005

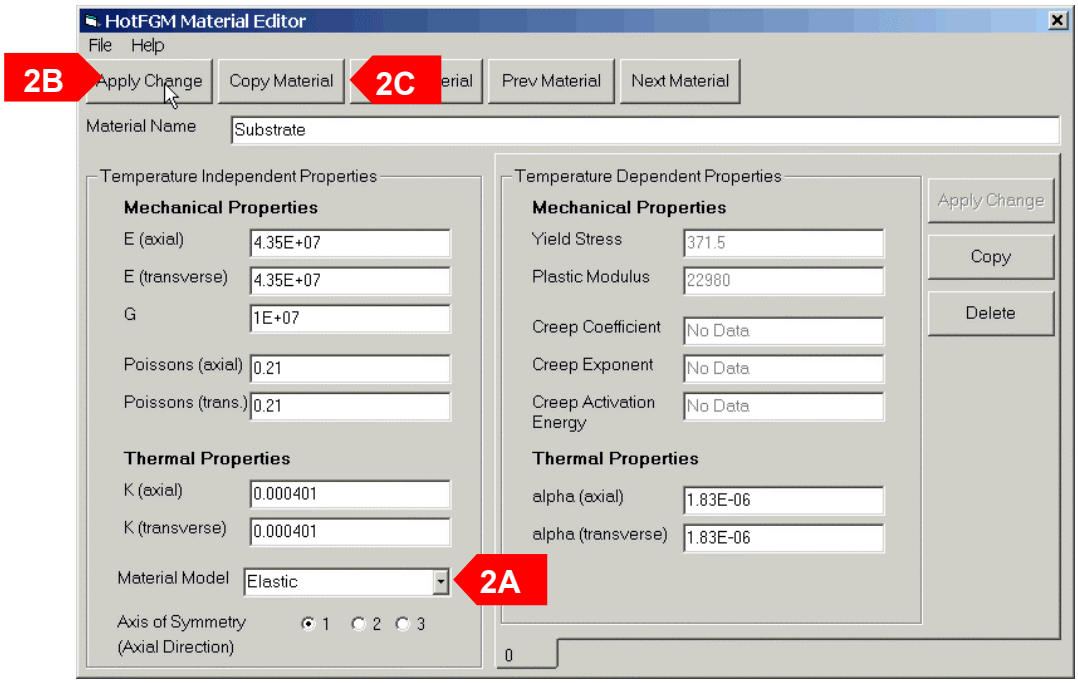
Problem Setup

1. Start HOT-SMAC. The first tab that appears on the main HOT-SMAC window is the **Setup** tab. Because the materials in this problem will be treated as linear elastic, select "Elastic" from the Constitutive Laws drop-down (**A**). Next, press the "Select/Define Materials" button (**B**) to bring up the Material Editor form.



Material Properties

2. On the Material Editor Form, change the Material Model (**A**) to "Elastic" and enter the material properties as outlined in Table 1. The Silicon Nitride substrate material properties are shown here. After entering material properties, be sure to press the "Apply Change" button (**B**) to save the current material. To enter the next material,

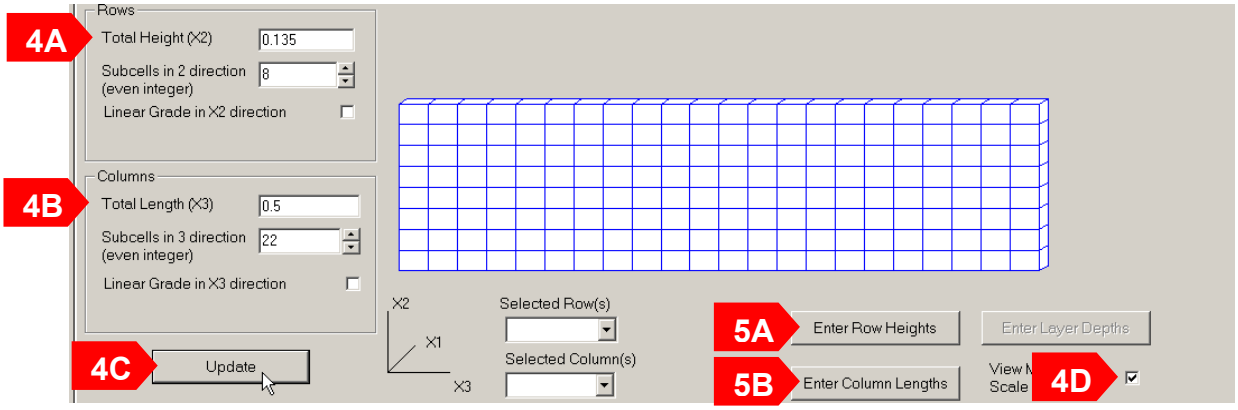


press the “Copy Material” button (**C**) and enter the name of the new material. Then enter the new material properties and press the “Apply Change” button. To navigate through the materials, use the “Prev Material” and “Next Material” buttons.

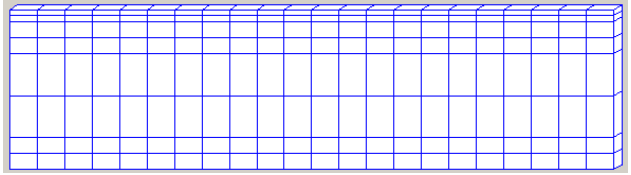
3. To save the materials for future use, you can export them to an ASCII file. Select “Export Materials” from the File menu, and type a filename for the exported materials. All materials in the current HOT-SMAC model will be exported. The exported materials file will have the extension “fmat”. In another HOT-SMAC problem, you can import this ASCII file by selecting “Import Materials” from the same menu. Close the Material Editor Form

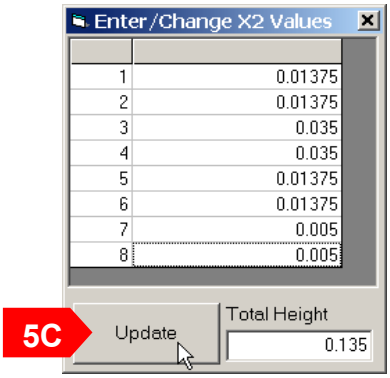
Geometry

4. On the HOT-SMAC main window, select the **Geometry** tab. Begin by setting the overall dimensions and the number of cells. For the Height (**A**), enter 0.135 inches and 8 subcells and for the Length (**B**), enter 0.5 inches and 22 subcells. When complete, press the “Update” button (**C**). By default, the model is drawn to maximize the size of the graphics window. To view the model to scale, check the “View Model to Scale” checkbox (**D**).



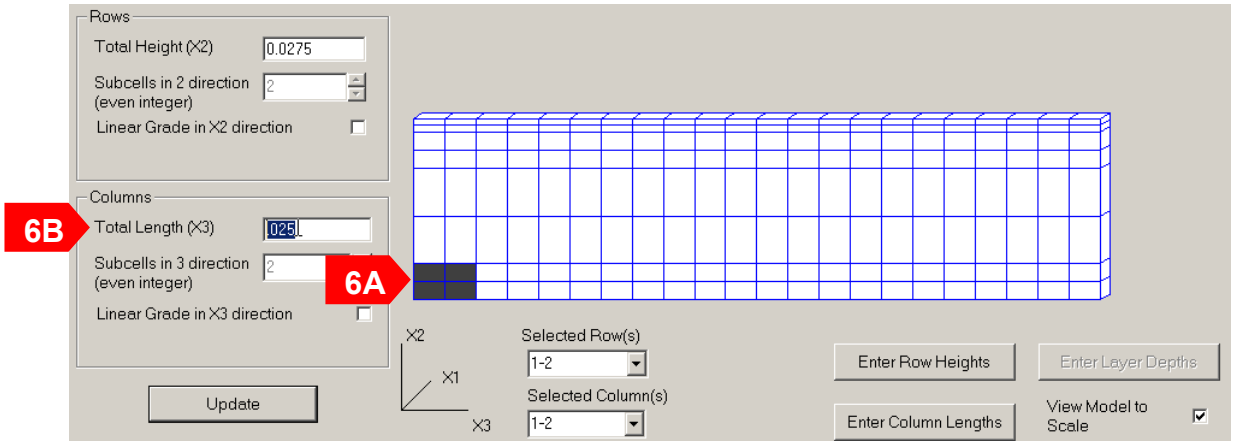
5. There are several ways of manipulating the cell sizes. You can enter the individual row heights and column widths by pressing the “Enter Row Heights” (**A**) and “Enter Column Widths” (**V**) buttons, respectively. Enter the heights for each row, by pressing the “Enter Row Heights” button (**A**). As you enter each height value, press the <Tab> key to enter this value and move to the next value, or press the <Enter> key. As you enter each value, the total height of the grid will be displayed in the “Total Height” textbox. After entering all





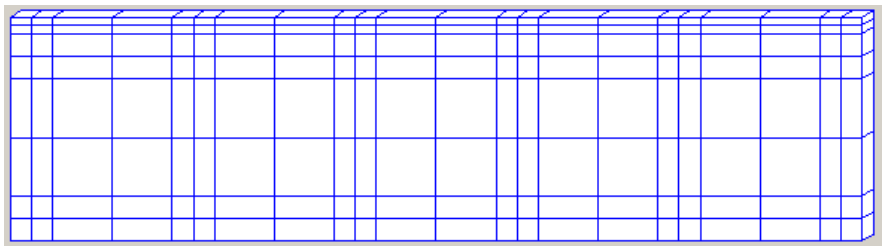
of the cell heights, press the “Update” button (**C**). The grid should now appear as shown.

6. You can also manipulate the cell sizes by selecting cells and then changing the dimension of the selected cells as a group. Try this by selecting the first two columns of cells (**A**) by left clicking on the grid and dragging a bounding box around the cells. Any cells that are included in the bounding box will be selected. To de-select cells, press the right mouse button. Alternatively, cells can be selected by holding the <Ctrl> key and left clicking on individual cells. Notice that when these cells are selected, the total height and length of the selected cells (i.e. the sum of the individual row heights and column lengths) are displayed in the Height and Length textboxes. You can change this height and/or length by entering a new value for the dimension in the corresponding textbox. Change the total length (**B**) of the first two columns to 0.025 inches. The mesh will be updated and the first two columns will each have



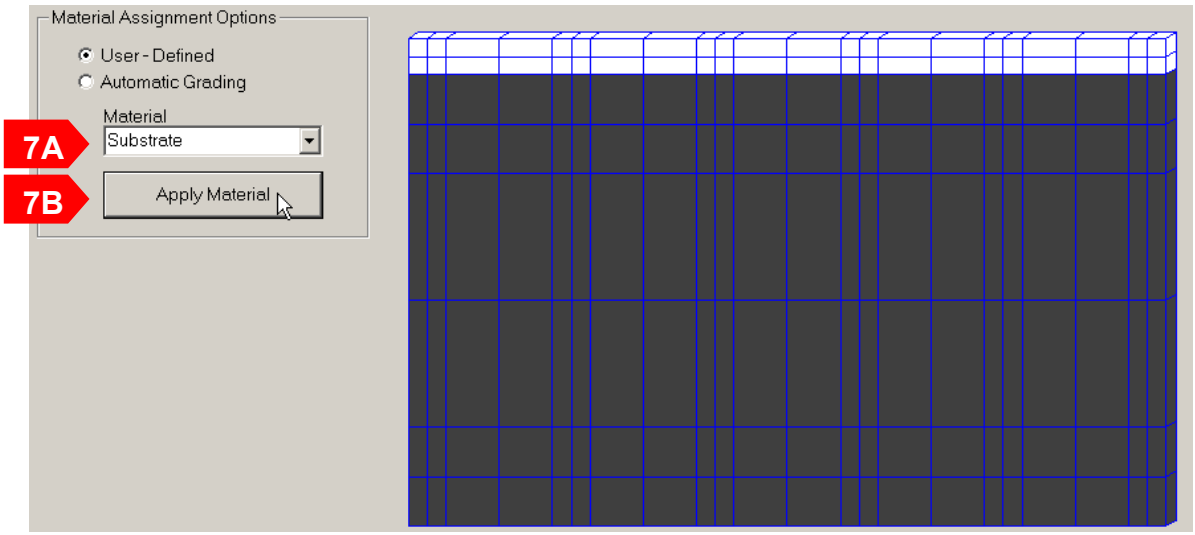
lengths of 0.0125 inches so that the total length of the first two columns is 0.025 inches.

Repeat this procedure for the second two columns for a length of 0.07 inches and alternate 0.025 inches and 0.07 inches down the length of the grid.

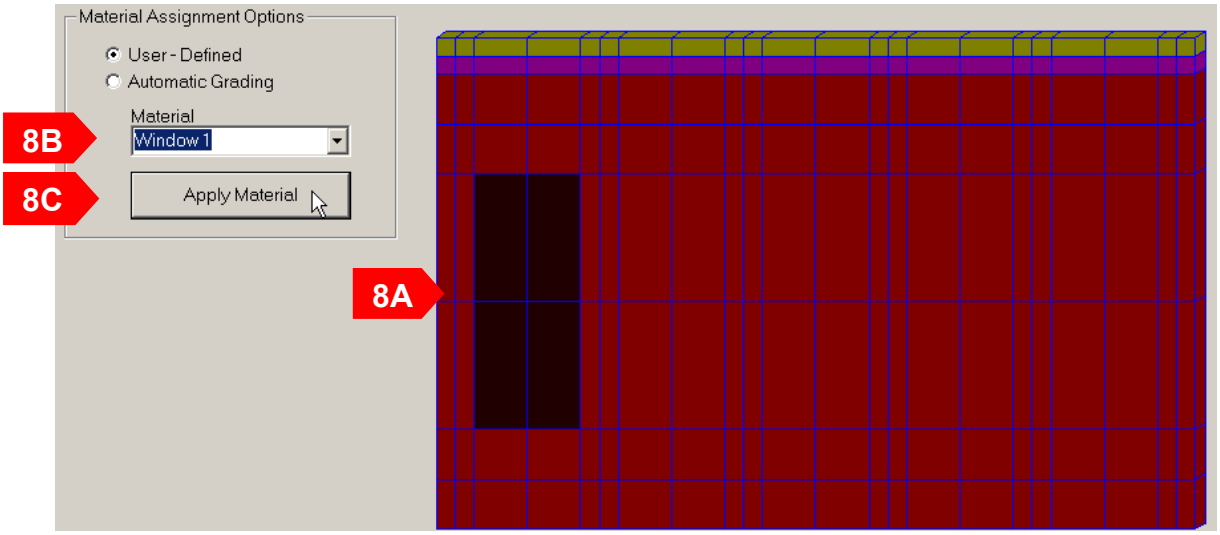


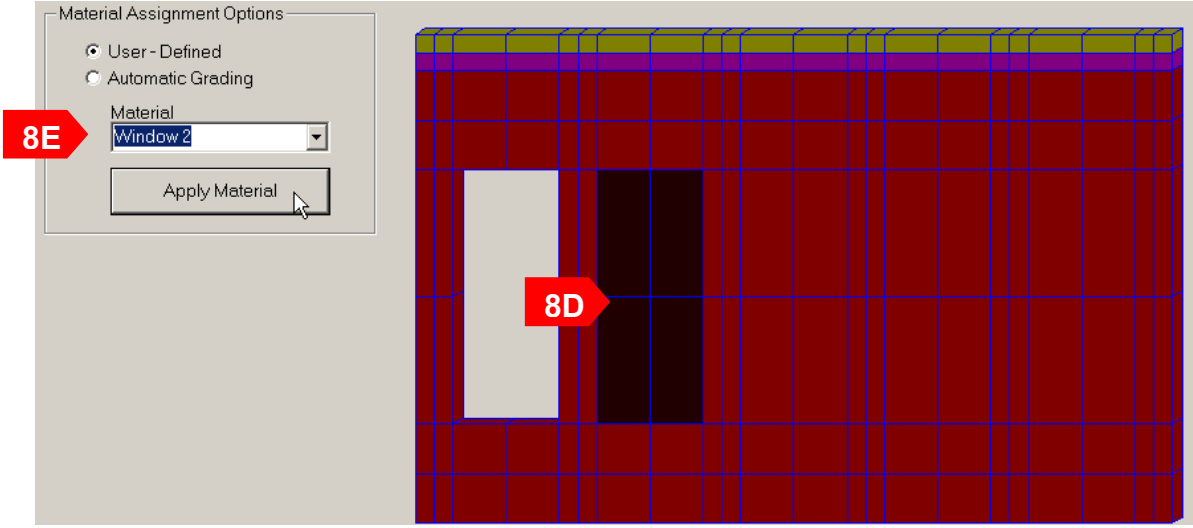
Material / Window Assignment

7. On the HOT-SMAC main window, select the **Materials** tab. Drag a bounding box around all but the top two rows of the grid, select "Substrate" from the Materials dropdown (**A**) and then press the "Apply Material" button (**B**). Select the next row, and apply the "BondCoat" material, and select the top row and apply the "Top Coat" material. It might help when entering the materials to turn off the "View Model to Scale" checkbox.



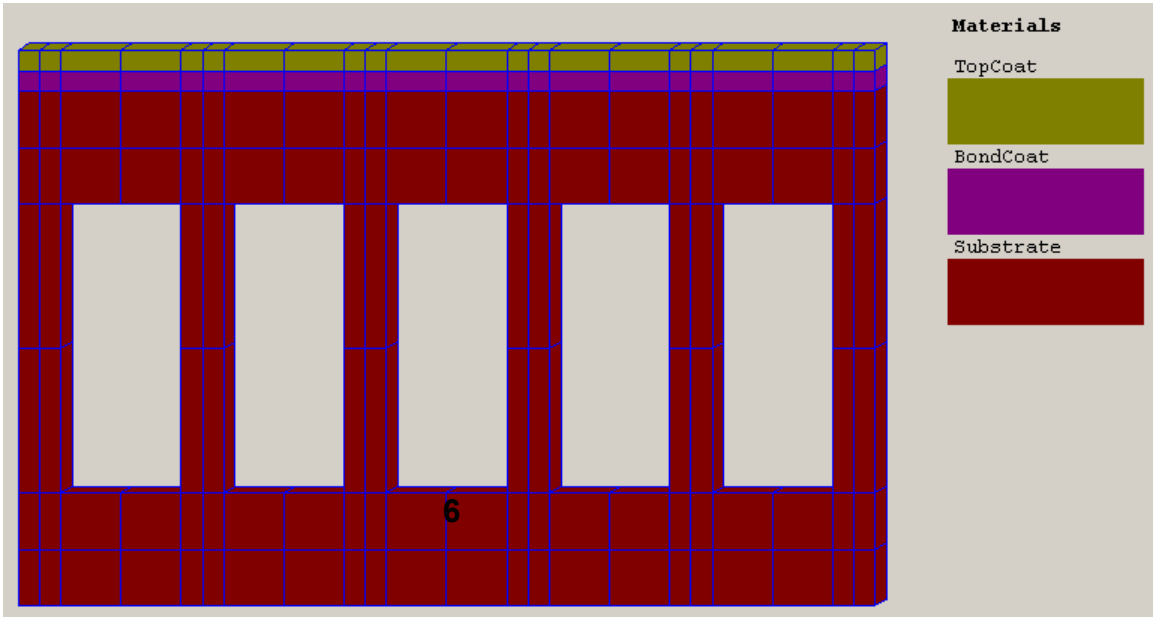
8. Windows (or holes) in the grid are also entered on the Materials tab. To enter the first window, select the cells shown (**A**) below [(3,3) to (4,4)], select "Window 1" from theMaterial dropdown (**B**), and press "Apply Material" (**C**).





Each window is treated as a separate entity and must begin on an odd row/column and begin on an even row/column. If cells are selected that do not meet this criteria, the GUI will notify you and attempt to adjust the window size to meet this criteria. Next, enter the second window by selecting cells (7,3) to (8,4) (**D**). Select "Window 2" from the Material dropdown (**E**) and press "Apply Material".

Repeat this procedure for windows 3, 4 and 5.

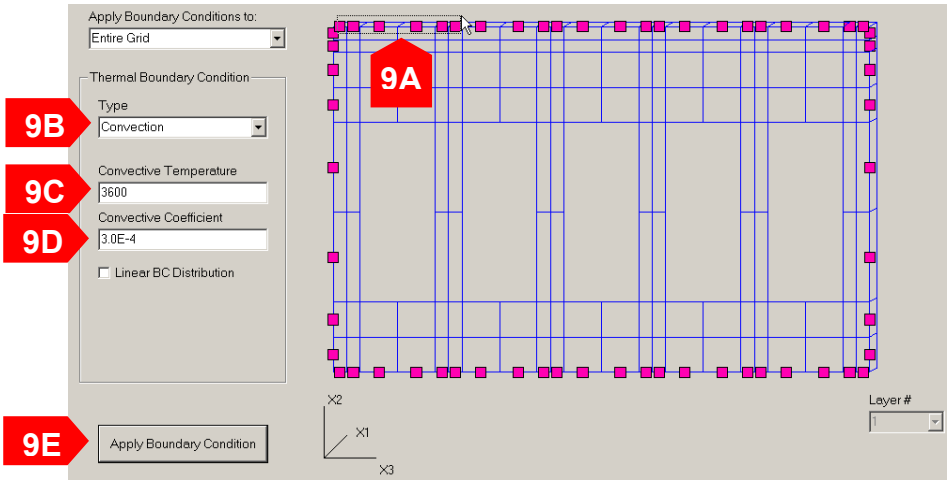


Windows can be removed from the grid by selecting "Remove Windows in Grid" from the Options menu of the HOT-SMAC main window. Note that if windows are removed from the grid, then materials must be re-assigned to the cells that were a part of that window.

Thermal Boundary Conditions

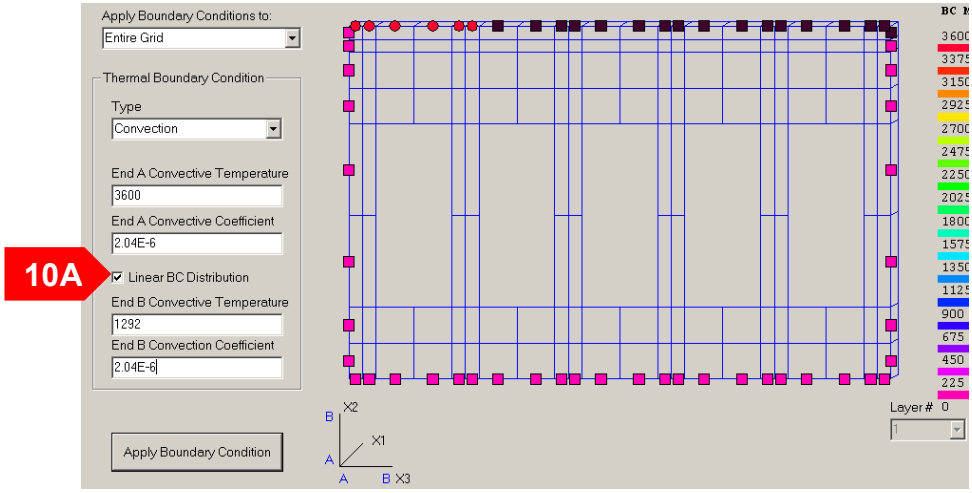
By default, all boundaries (for the entire grid and for each window) are assigned a uniform temperature of 0 degrees. In the following steps, we will assign the convective heating and cooling boundary conditions to the model.

9. On the HOT-SMAC main window, select the "Thermal B.C." tab. When this tab is selected, the boundary conditions for the entire grid (as opposed to the windows) are displayed. Draw a bounding box to select the boundary cells corresponding to the flame (cells 1-6 along the top) (**A**). The boundaries of these cells will become



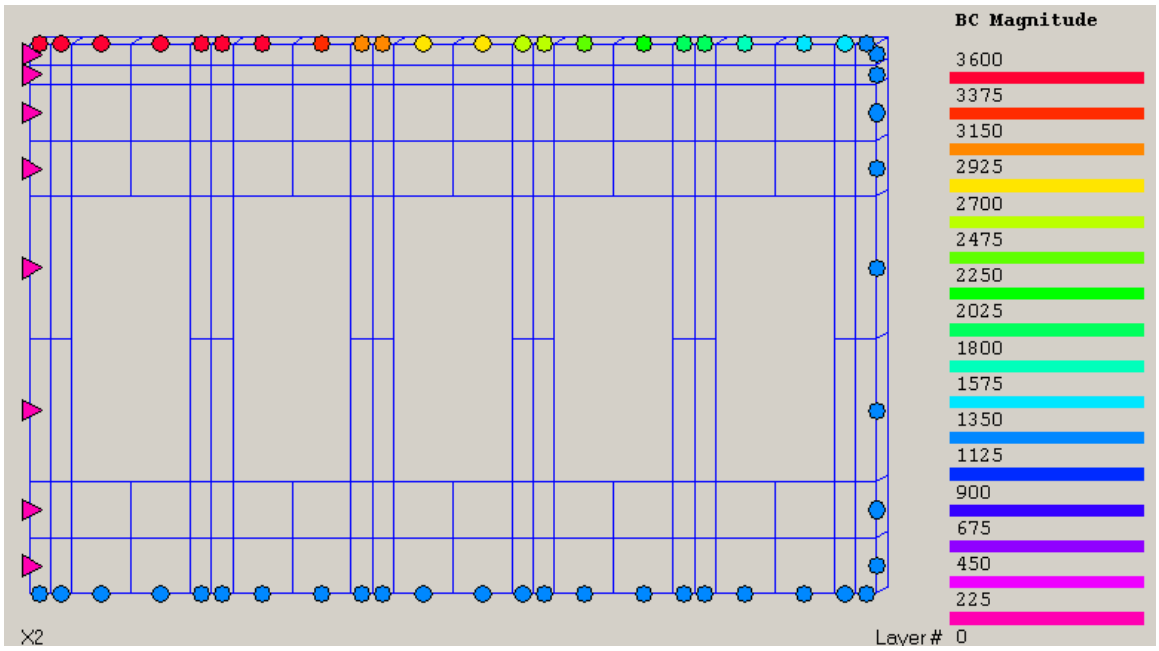
highlighted. Selected "Convection" from the "Type" dropdown menu (**B**) and enter 3600 F and 3.0E-4 as the convection temperature and coefficient respectively (**C, D**). Finally, press the Apply Boundary Condition button.

10. Next, enter the convective boundary away from the flame. In this case, the boundary condition linearly varies from the flame to the right end of the model. Select the remaining boundary cells and then check the "Linear BC Distribution"

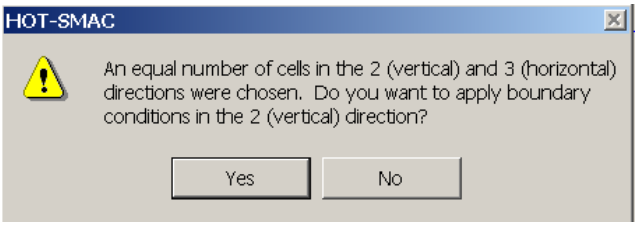


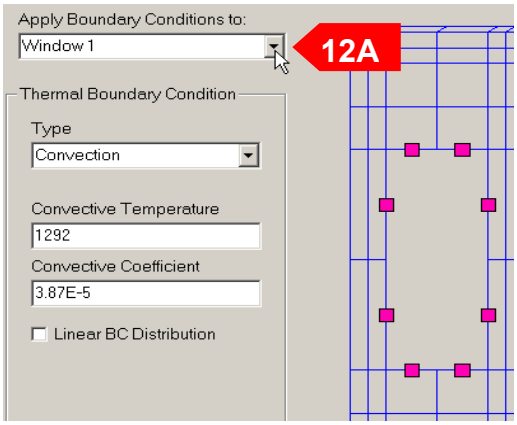
checkbox (**A**). A second set of convective temperature and coefficient textboxes will appear. Enter the values shown and press the "Apply Boundary Condition" button.

11. The entire right side and bottom boundary conditions will be assigned a convective temperature of 1292 F and coefficient of 2.04E-6. The left side boundary condition is a symmetry boundary and should be assigned a flux boundary condition of 0 (insulated). Assign these boundary conditions now by selecting the appropriate boundary cells, selecting the Type, typing the numerical values and pressing the "Apply Boundary Condition" button.



12. Next, apply thermal boundary conditions to the first window in the grid. Select "Window 1" from the "Apply Boundary Conditions To:" dropdown list (**A**). The boundary cells for Window 1 are highlighted. Select all of the boundary cells for Window 1. When you select the cells, the following message box will pop up:





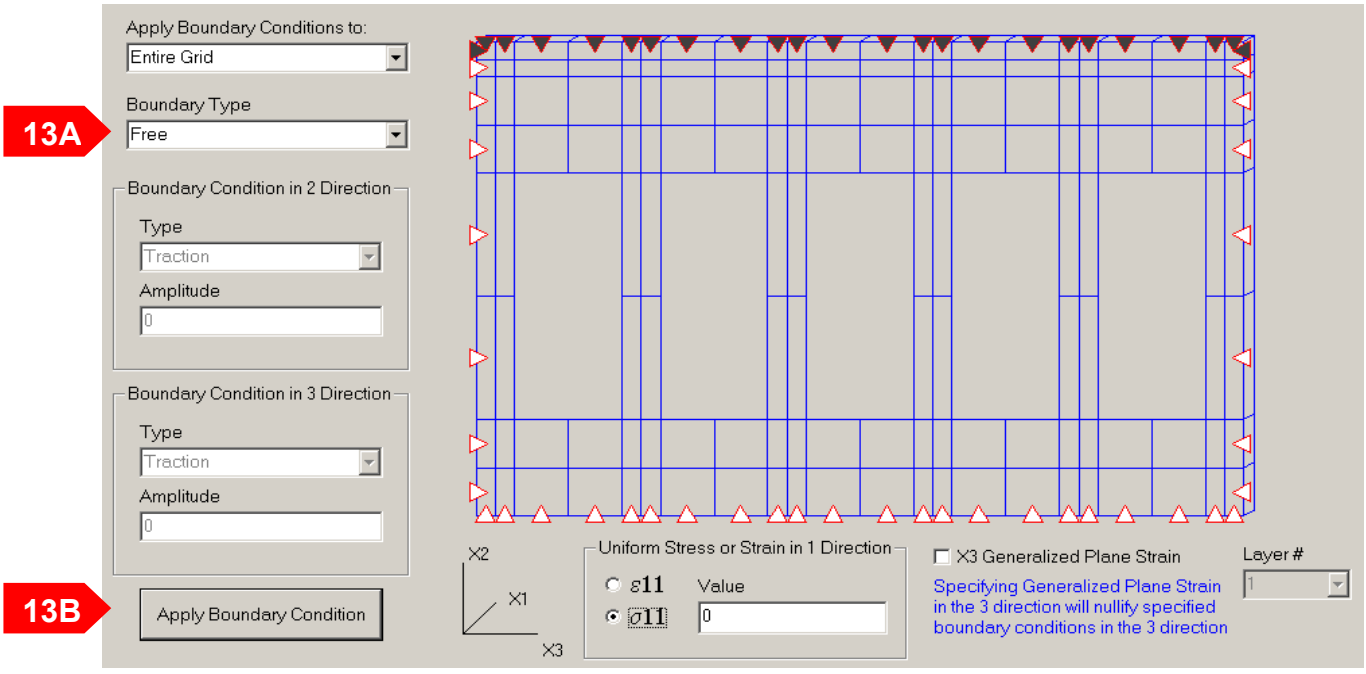
Because boundary conditions for some cells (such as the corner cells) are entered for both the top/bottom and left/right surfaces, when boundary cells are selected, the HOT-SMAC GUI attempts to determine whether you intend to apply boundary conditions in the vertical or horizontal directions. If it cannot determine which to apply, it will ask you. Press the Yes button to apply the vertical boundary conditions. Type in the appropriate boundary condition values (T=1292 F,

h=3.87E-5) and press the “Apply Boundary Condition” button. Select the boundary cells again, and this time, press No when the message box pops up to indicate that you are applying boundary conditions to the horizontal boundaries. Repeat this procedure for the remaining four windows.

Mechanical Boundary Conditions

Mechanical boundary conditions are entered in a very similar way to thermal boundary conditions. In the following steps we will assign mechanical boundary conditions to the model. Just as with the thermal boundary conditions, we start by applying boundary conditions to the entire grid and then apply them to each window.

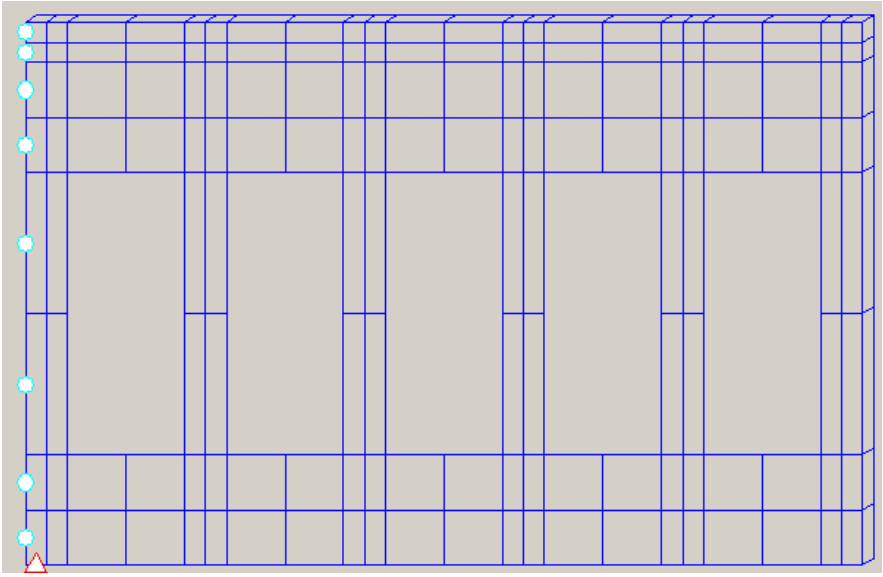
13. From the HOT-SMAC main window, select the Mechanical B.C. tab. The default boundary condition for all boundaries is “pinned” or zero displacement in both the 2 and 3 directions. Select the boundary cells along the top, select “Free” from the “Boundary Type” dropdown, and press the “Apply Boundary Condition” button. Repeat this procedure to apply free boundary conditions along the right side boundary and then for the bottom boundary.



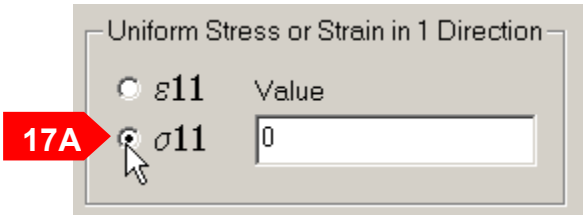
14. For the left boundary, select all of the boundary cells, select “Roller” from the “Boundary Type” dropdown, and press “Apply Boundary Condition”.

15. In order to prevent singularities, the boundary conditions must prevent any rigid body motion. This is accomplished by “pinning” the lower-left corner cell. Select this cell, answering “Yes” when the dialog pops up to indicate that this boundary condition will be applied in the 2 (vertical direction). Select “Pinned” from the Boundary Type dropdown, and press “Apply Boundary Condition”.

The boundary conditions for the entire grid should appear as shown below.



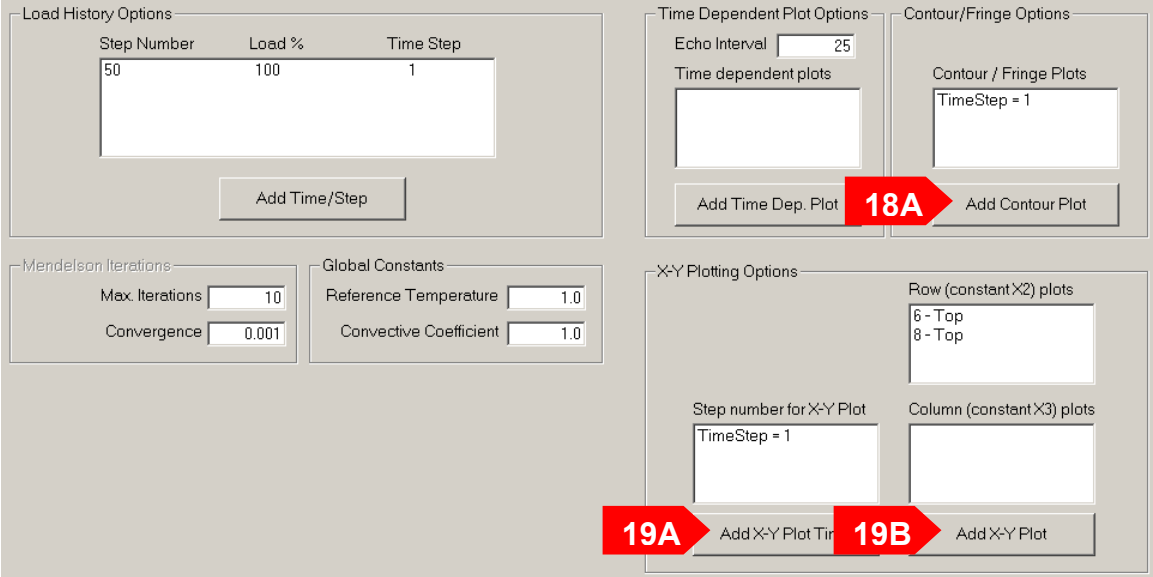
16. Select the boundary cells for each window (using the "Apply Boundary Conditions To:" dropdown) and make each boundary for each window "free".



17. Finally, specify that the out-of-plane (1 direction) boundary condition is that of zero stress by clicking the radio button at **A**. Note that this indicates that the average stress for the cross section is zero, not the stress for each cell.

Analysis Options

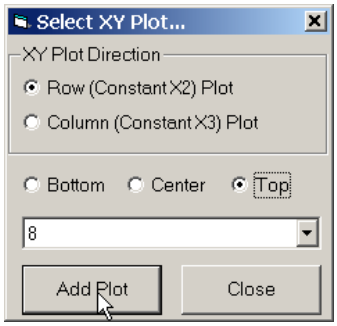
Because this is a linear, elastic, non-time dependent problem, the entries on the left side of the **Analysis Options** tab are not applicable to this problem. For linear elastic problems, only one load step is performed, and the Mendelson iterations are not applicable. Therefore we need only specify plotting options on the right hand side.

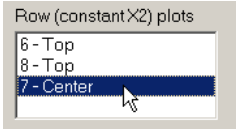


Because this is not a time dependent problem, there is no need to add a time dependent plot under the Time Dependent Plot Options.

18. Add a contour plot by pressing the "Add Contour Plot" button (**A**). In the resulting dialog, type "1" to indicate that a contour plot should be generated on the first (and in this case, only) load increment. Press the OK button.

19. Add an XY Plot Time point by pressing the "Add X-Y Plot Time" button (**A**). In the resulting dialog, type "1" to indicate that row and column plots should be generated on the first (and in this case, only) load increment. Press the "Add X-Y Plot" button (**B**) to add rows and/or columns to plot. In the resulting dialog, add a row plot for the top surface of the grid, which is row 8 along the top of the cell. Press the "Add Plot" button to add this plot. Press the Add X-Y Plot button again to add another row plot along the interface between the substrate and the bondcoat, which is row 6 along the top. You can delete any row or column plot (or X-Y Plot Time or Contour Plot Time). Simply highlight the plot on the Analysis Options tab, and press the <Delete> key.

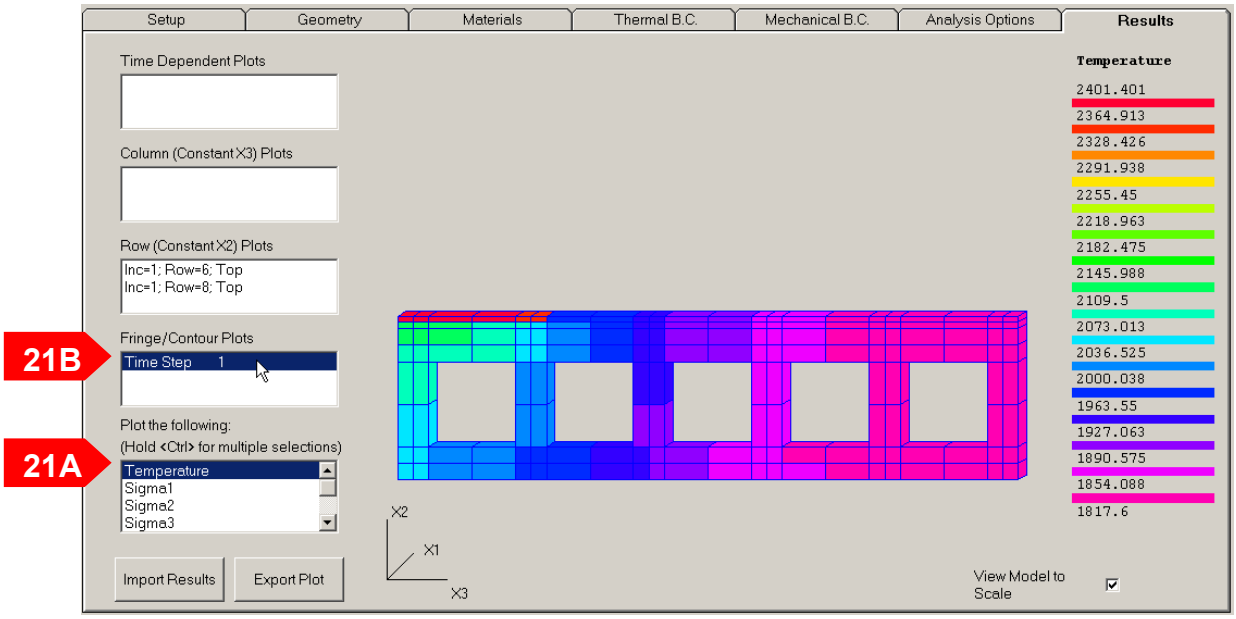




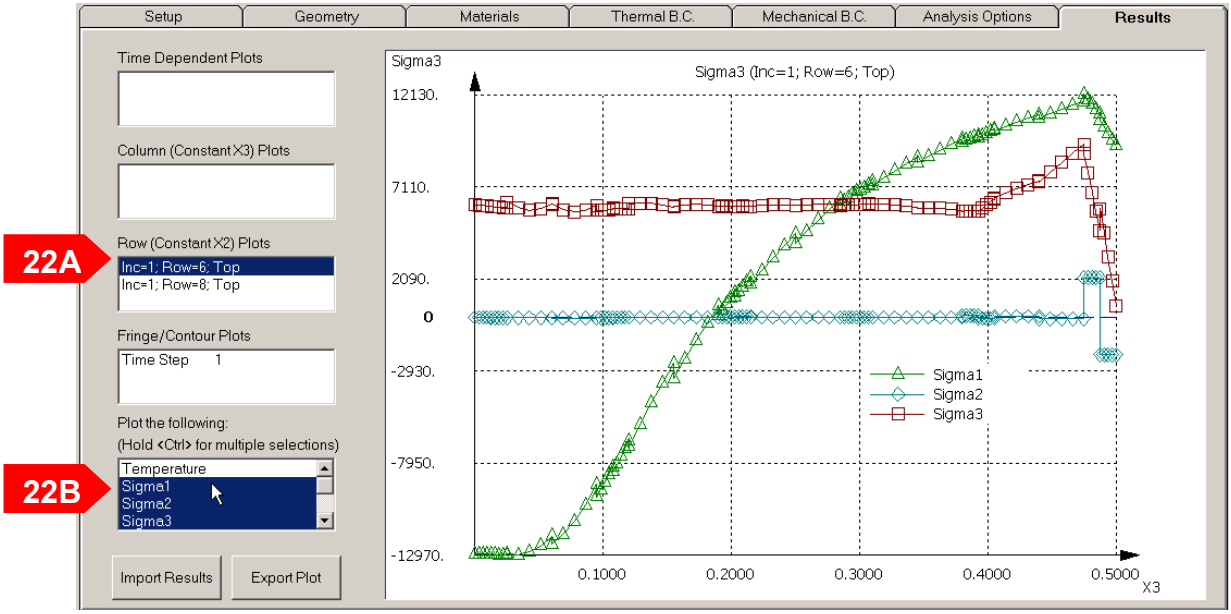
Running The Analysis

20. Now that the problem is set up, press the "Analyze" button to run the HOT-SMAC analysis and import the results into the GUI.

21. On the HOT-SMAC main window, press the **Results** tab to view results from the latest HOT-SMAC analysis. The first time that this tab is shown, there are no results selected for display. First select a quantity to display. Highlight "Temperature" in the list box (**A**) labeled "Plot the following". Next highlight "Time Step 1" in the "Fringe/Contour Plots" listbox (**B**) to display contours of temperature at the first load increment. Click on other quantities in listbox **A** to view stress results on the model.



22. Next, plot stresses along the substrate-bondcoat interface. Highlight the item, "Inc=1, Row=6, Top" in the "Row (Constant X2) Plots" listbox (**A**). When this is highlighted, whatever quantity is highlighted at **B** is plotted in an X-Y graph as a



function of X3. You can also plot multiple quantities in the X-Y plot. To plot multiple quantities, hold the <Ctrl> key and select quantities in the bottom listbox () or just click an hold the mouse button on any quantity and “swipe” multiple quantities.

Section 3: HOT-SMAC Verification Example

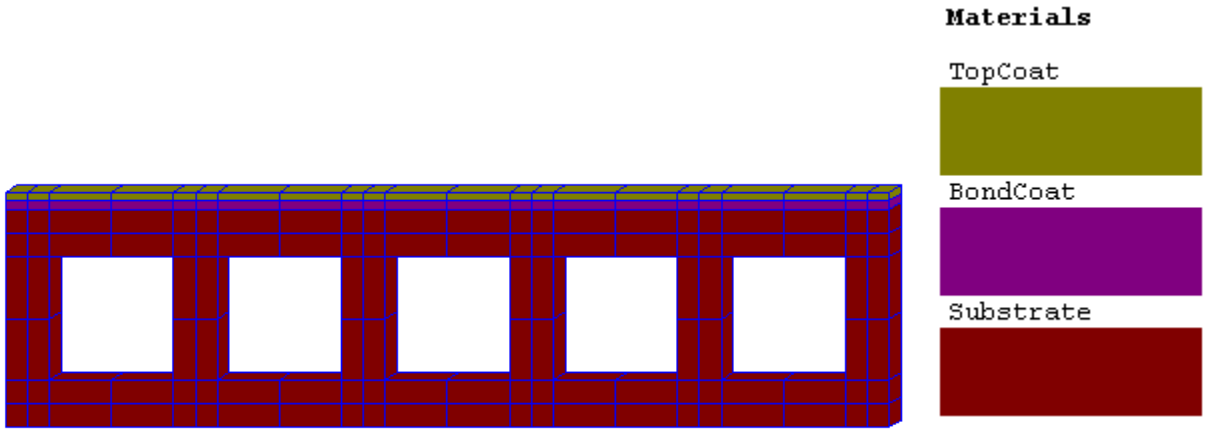
Thermo-Elastic Analysis of an Internally Cooled Structures

Analysis Results

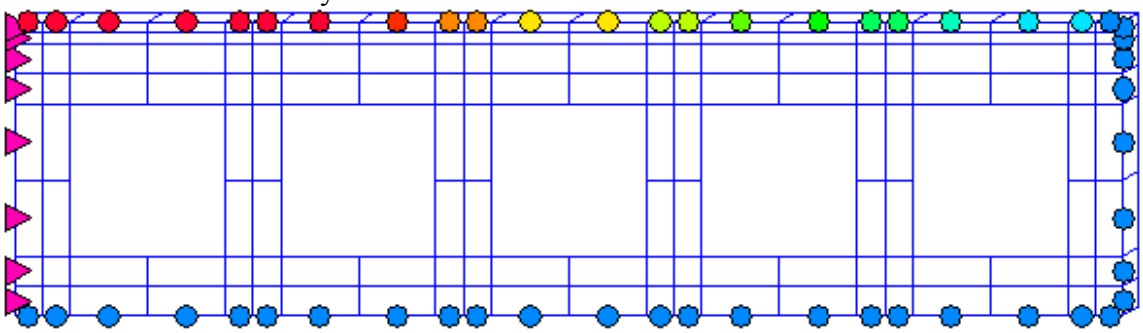
Analysis results are presented here for the NASA/TM example problem that was used as an example tutorial in the last section. Comparisons to the NASA/TM results are presented as appropriate. The solution is first presented on the 8x22 mesh from the previous section. Next, the analysis is presented on a mesh with 26 cells in the X2 direction and 74 cells in the X3 direction.

8X24 Coarse Grid

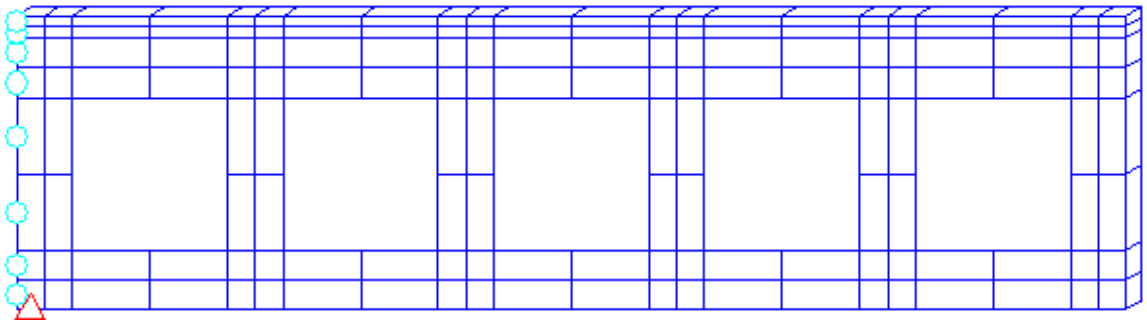
- Material Layout



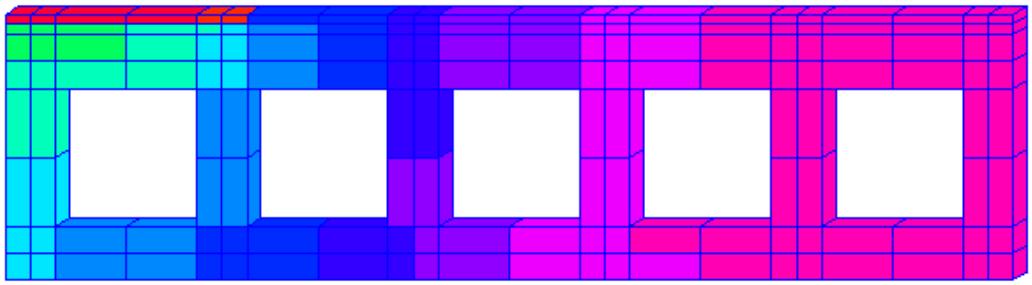
- Thermal Boundary Conditions

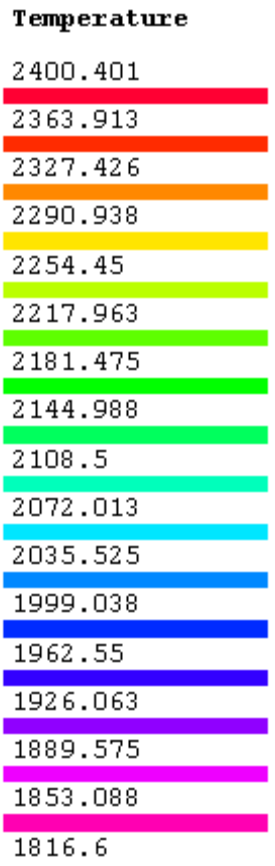


- Mechanical Boundary Conditions

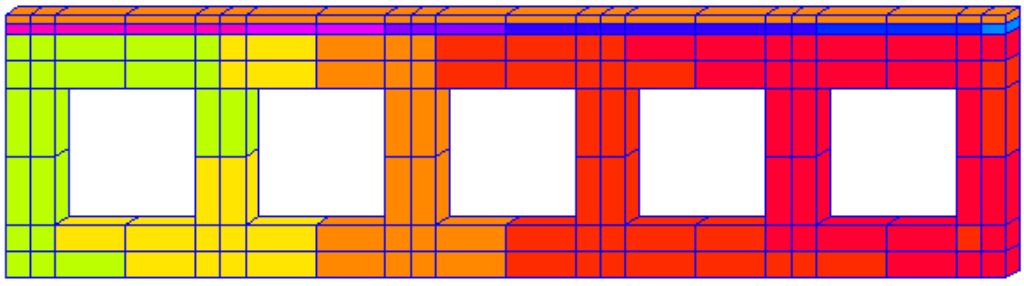


- Temperature Contours

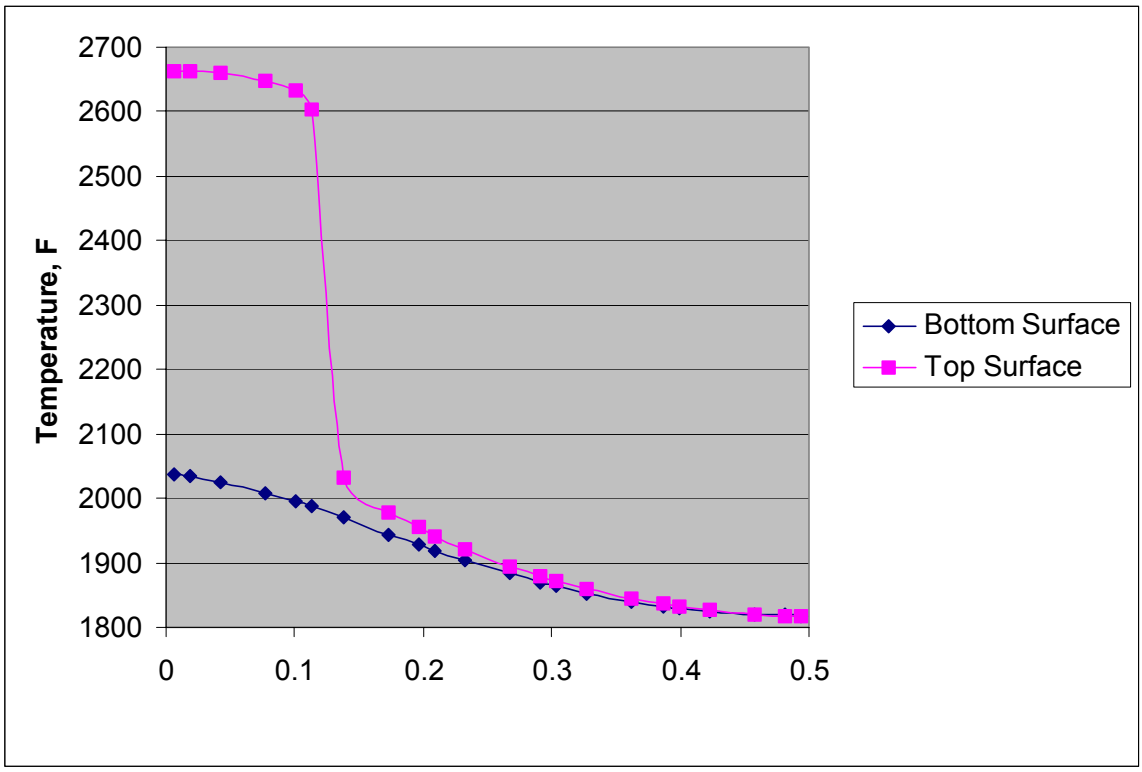




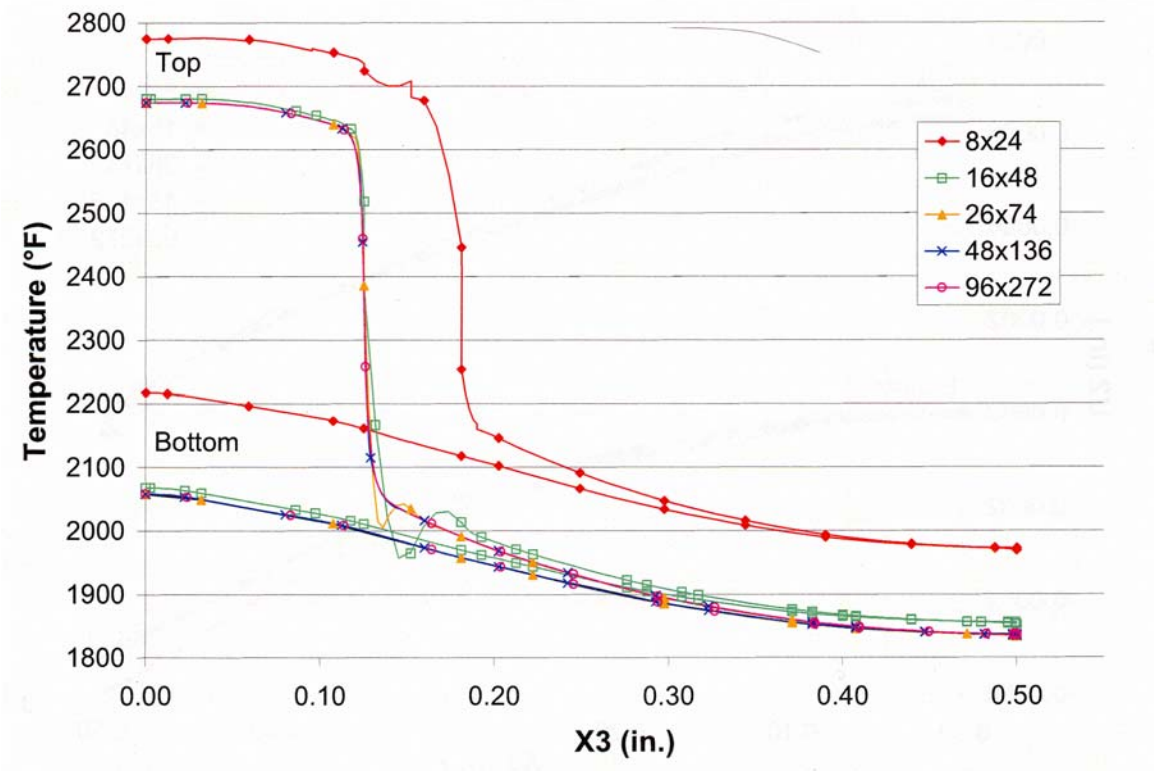
- Contours of σ11



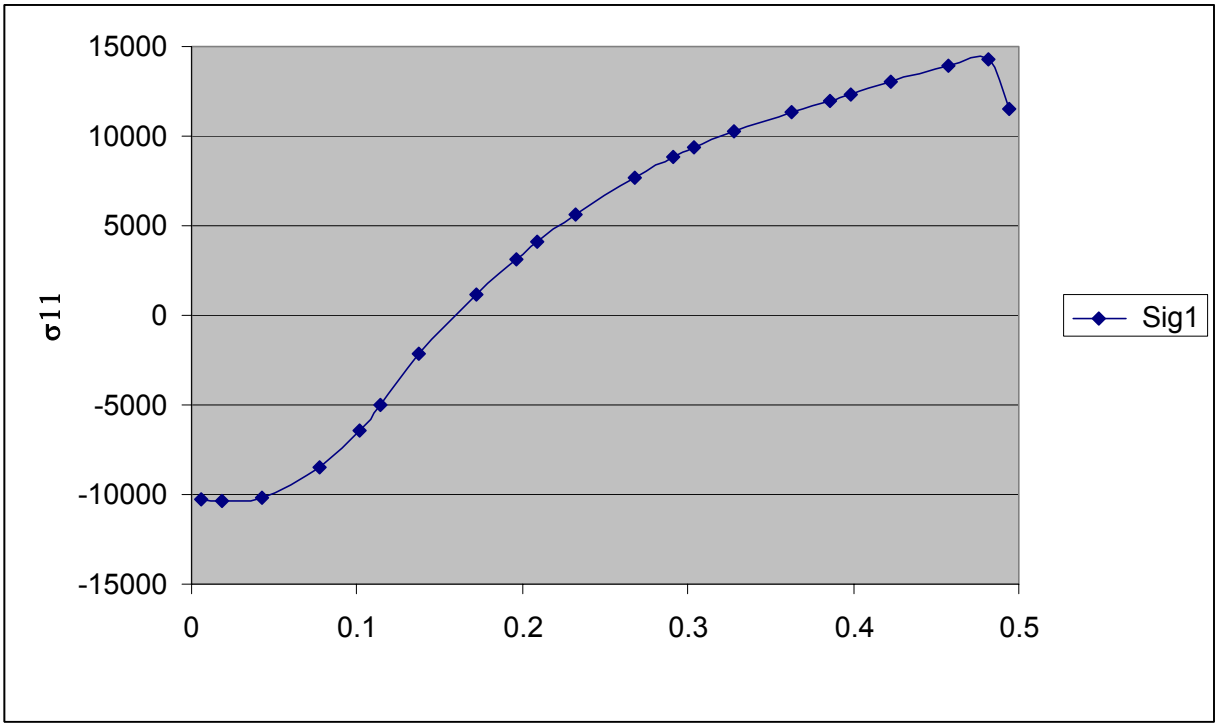
- Temperature along top and bottom



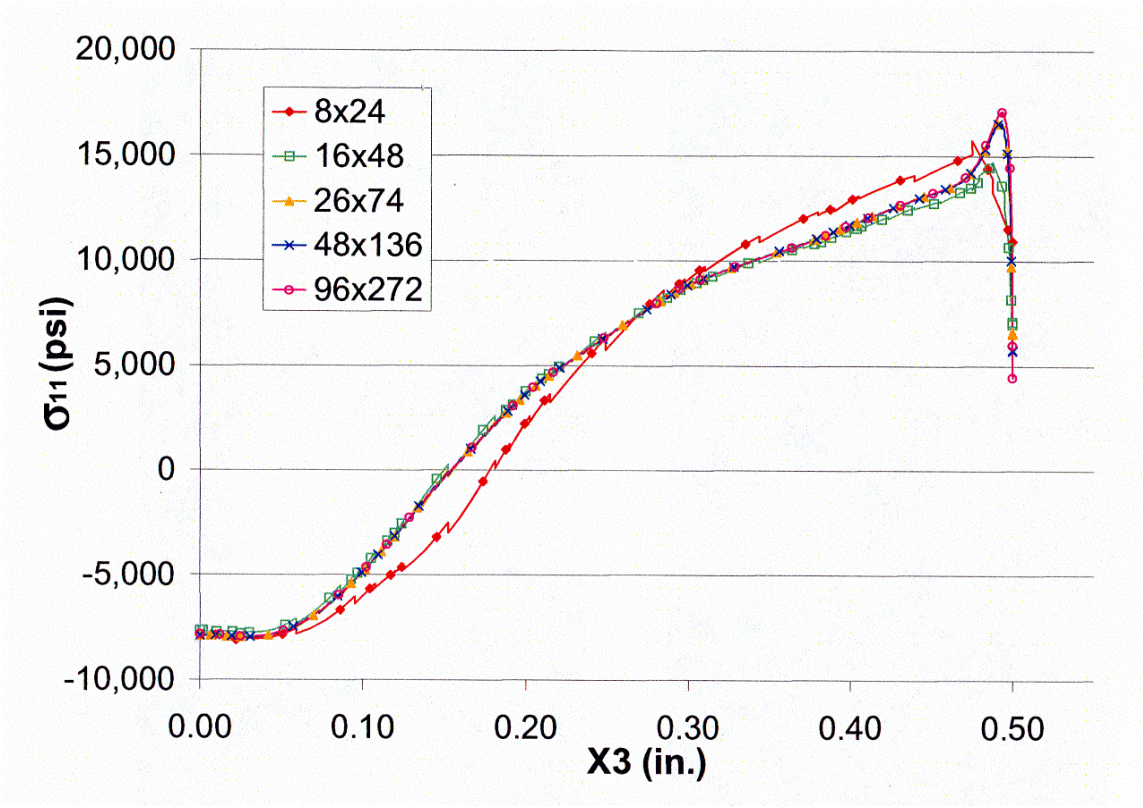
- Temperature along top and bottom from Ref. [1]



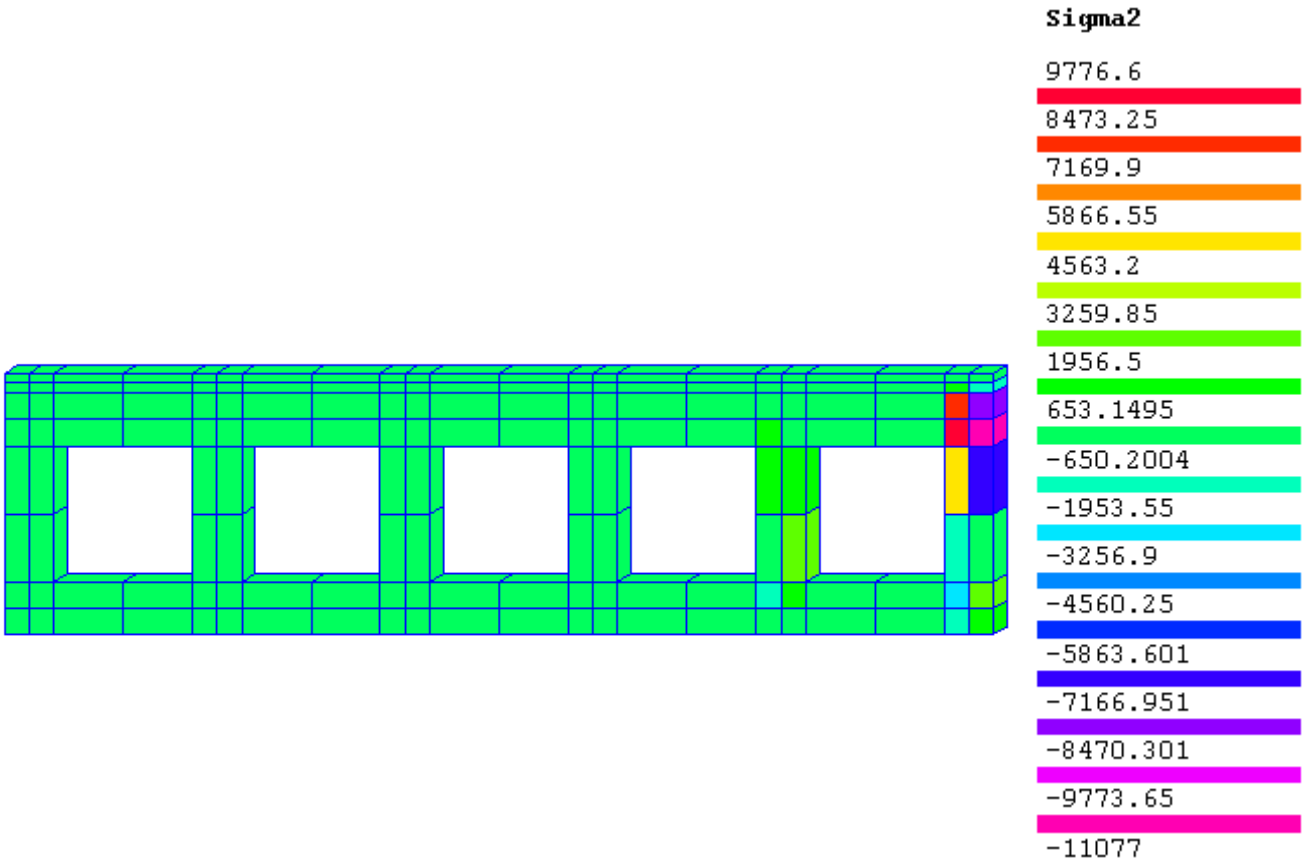
σ11 in substrate along substrate-bondcoat interface (Present)



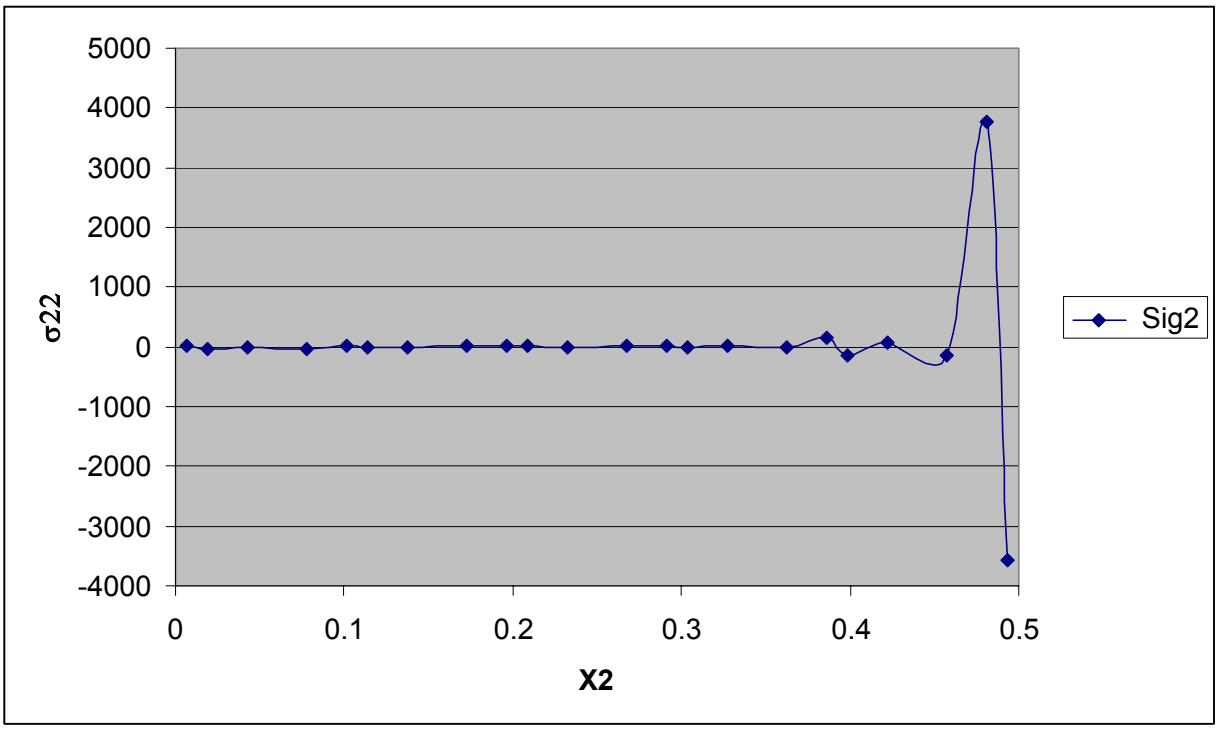
- σ11 in substrate along substrate-bondcoat interface from Ref. [1]



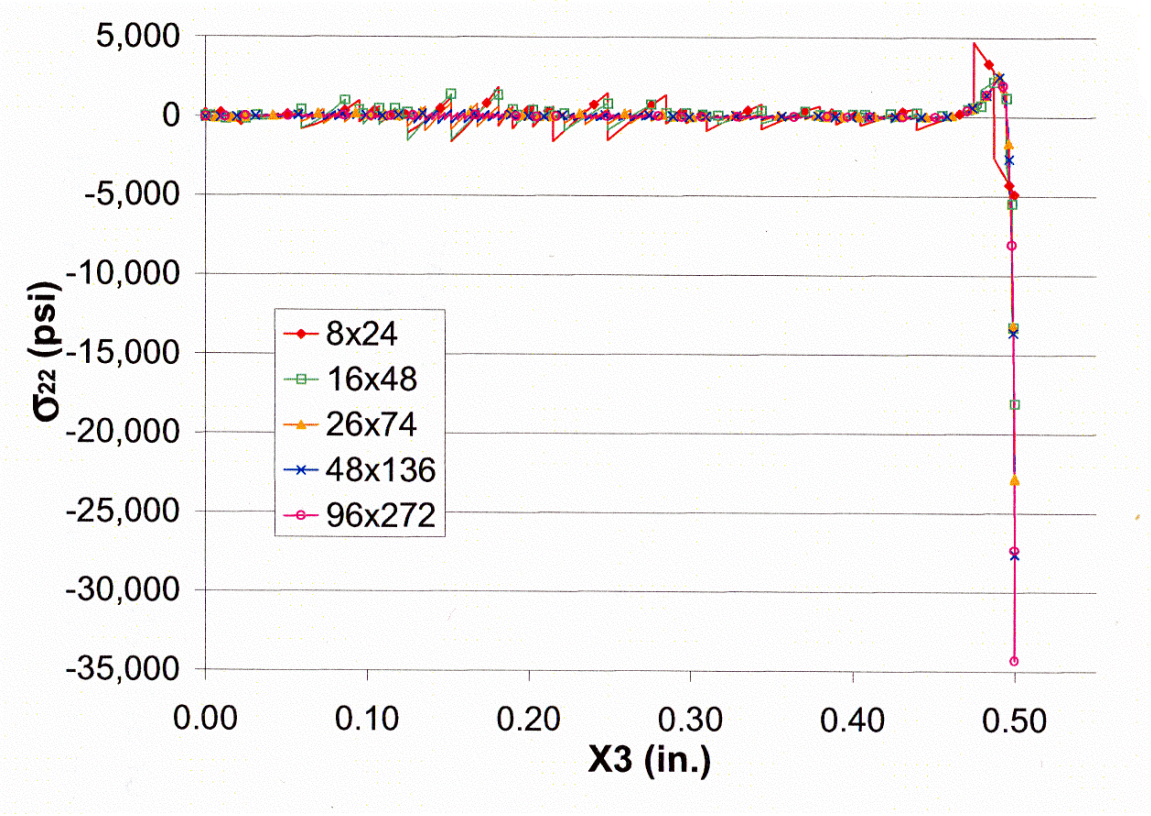
σ22 Contours



- σ22 in substrate along substrate-bondcoat interface (Present)

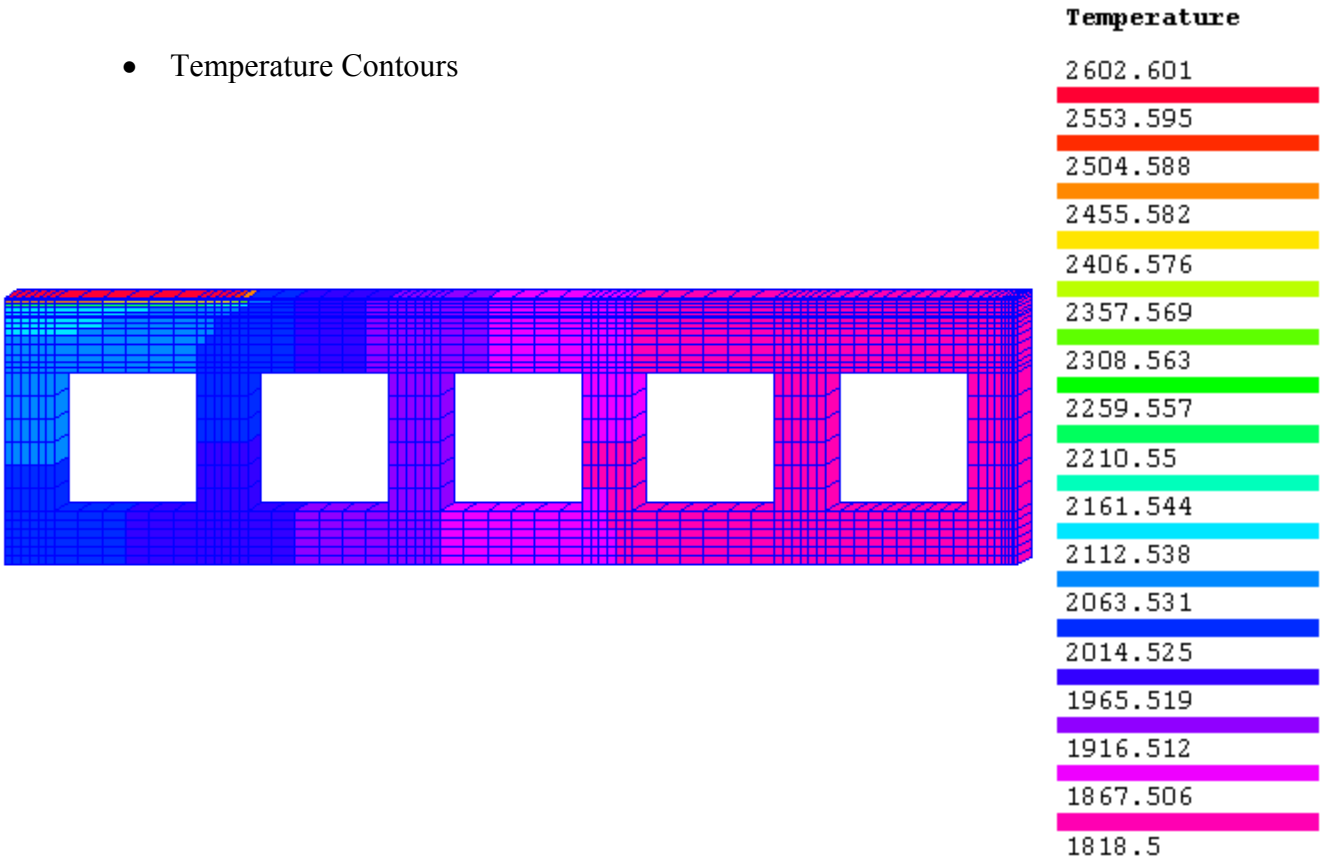


- σ22 in substrate along substrate-bondcoat interface from Ref. [1]

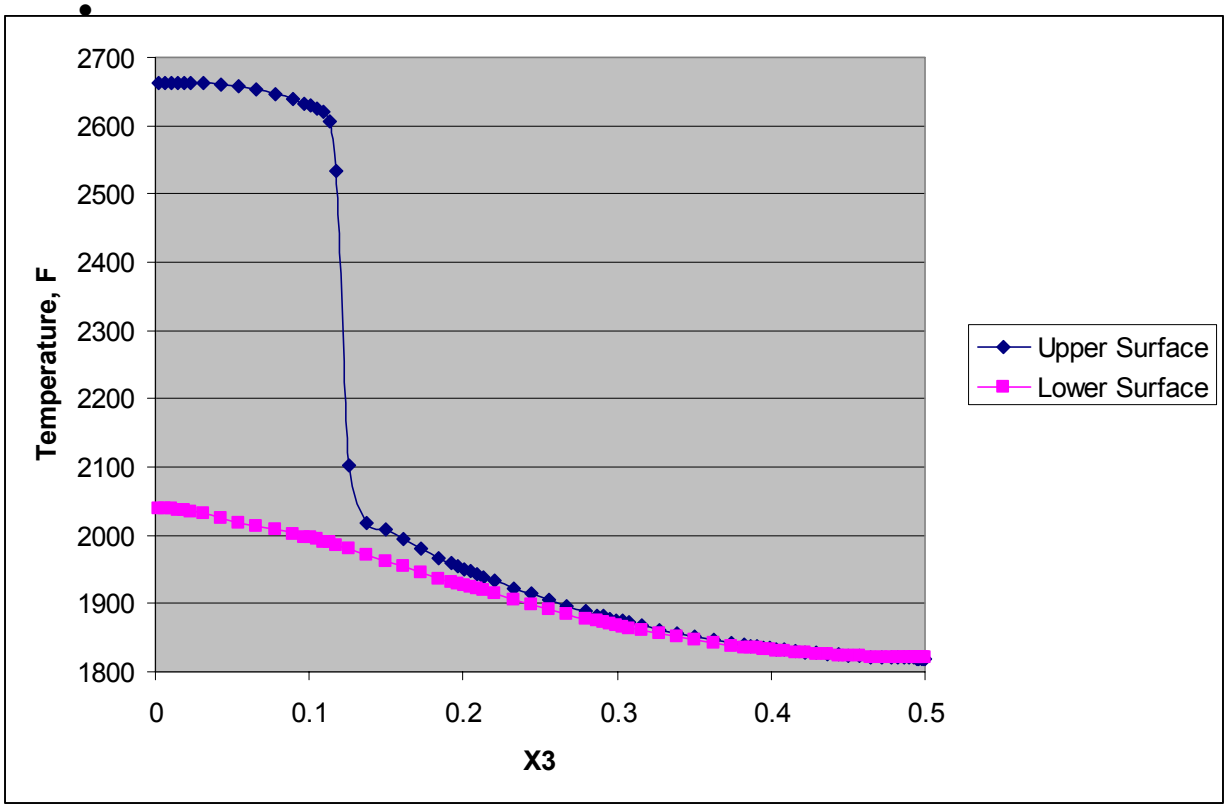


26X74 Medium Grid

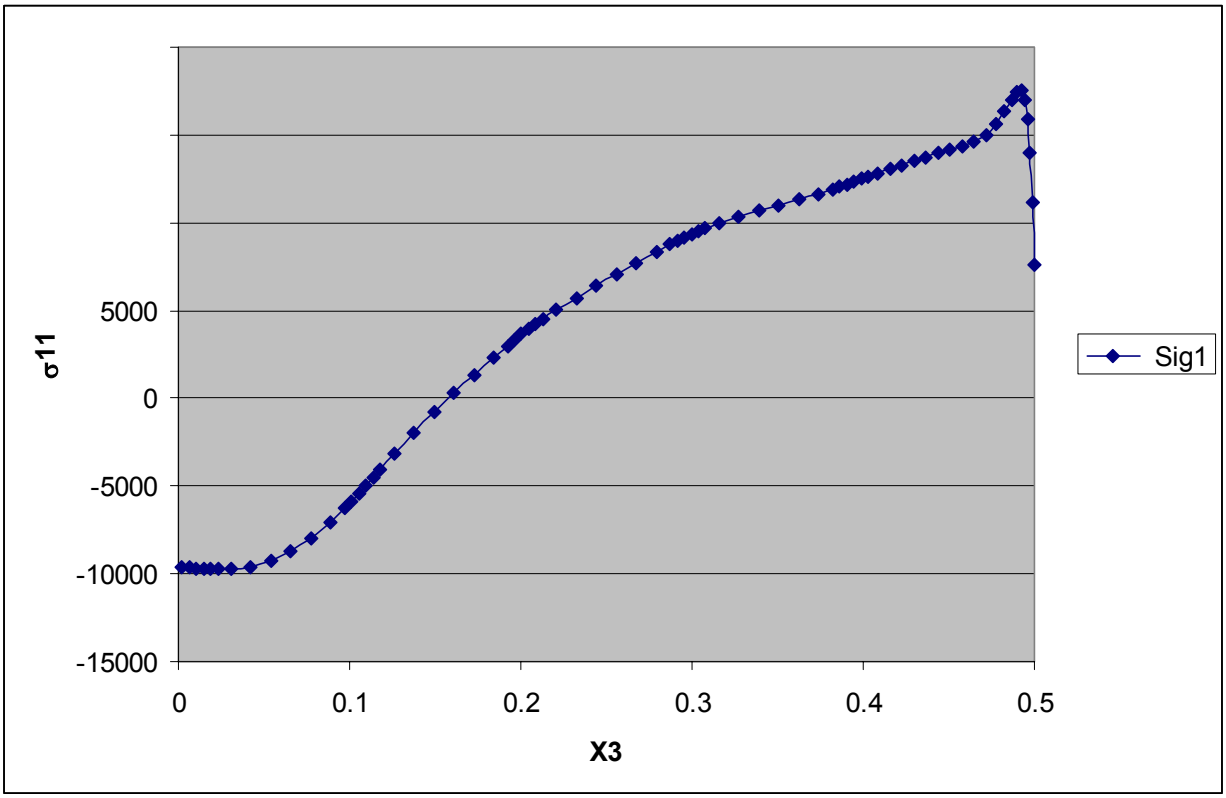
- Temperature Contours



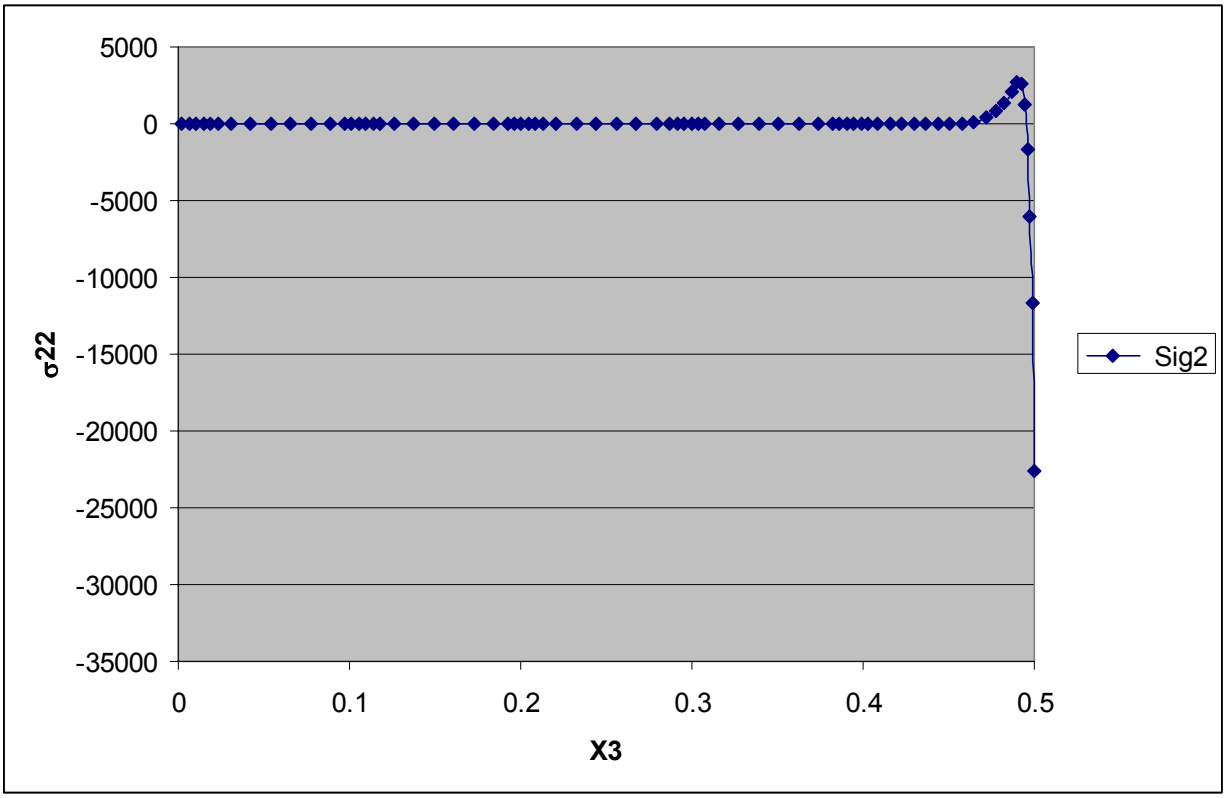
- Temperature along upper and lower surfaces
-



σ11 along substrate-bondcoat interface



- σ22 along substrate-bondcoat interface



Discussion

The results presented here are not so much to validate the physics of the HOT-SMAC method as to validate the process of building HOT-SMAC grids and analyzing the results using the new HOT-SMAC GUI against a body of established HOT-SMAC results.

In general, the results predicted by the current version of HOT-SMAC compare well to the results presented in Reference [1]. The first thing to notice is that in the temperature profile plots, the current version of the software appears to generate a very similar answer to the finer mesh solutions of the software used in [1].

The s11 and s22 stress result trends seem to be the same, however, the magnitudes seem to be off slightly for a given mesh density. For example, in the s11 plots at the interface between the substrate and the bondcoat, the lower bound on the stress is approximately –10000 psi, where in [1], the lower bound appears to be closer to –8000 psi. Three possible causes for this discrepancy are: 1) The meshes are not identically the same; 2) it appears that the elastic material properties (E, ν, K) are temperature dependent in the work of Ref. [1], where in the current software, only the thermal expansion coefficient is temperature dependent, 3) an averaging of field quantities within each subcell is used in the current software package; whereas in Ref. [1] actual point wise results where shown. It appears after further investigation that the bulk of the discrepancy is due to this averaging process; consequently the use of this averaging procedure will now become an option rather than mandatory.

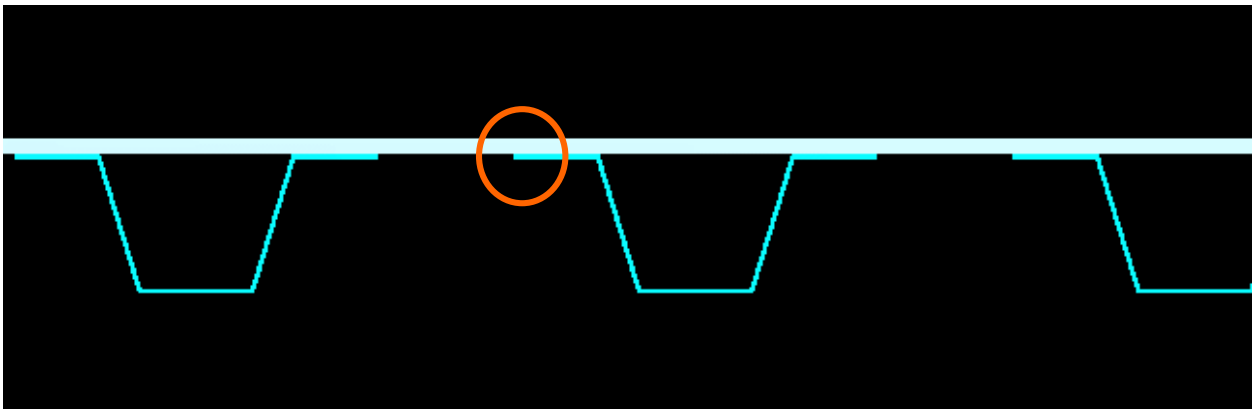
Section 4: HOT-SMAC – HyperSizer Integration Research

HOT-SMAC - HyperSizer integration research

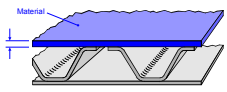
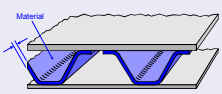
HyperSizer passes panel level solutions to HOT-SMAC to obtain local analyses of particular design features. The HyperSizer solution obtains accurate thermoelastic internal loads and resulting stress and strain fields for all of the panel span segments based upon a consistent state of Free Body Diagram load balance and strain compatibility. However, HyperSizer formulations are based on classical lamination theory and Kirchoff's plane sections remain plane deformation theory and as such do not consider Z axis (axis in the HOT-SMAC **2** direction) effects.

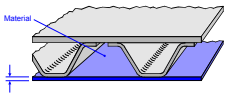
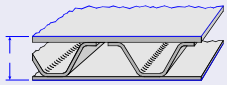
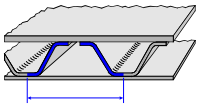
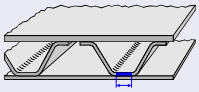
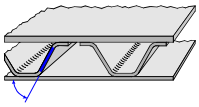
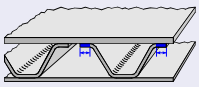
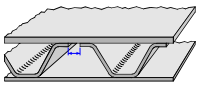
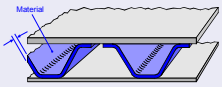
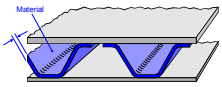
In this example, the HOT-SMAC software is used to supplement HyperSizer's solutions by providing computation of shear stress (τ_{23}) and normal peel stress (σ_2) for bond regions and areas of free edges. The problem is a hat-shaped, metallic stiffened panel with a machined/extruded built-up plate region to represent the facesheet to flange design feature.

Shown in the figure is the panel cross section to relative scale. The orange circle represents the area of analysis refinement.

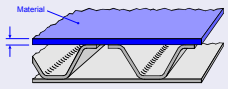
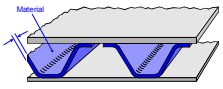


The actual design dimensions and material follow.

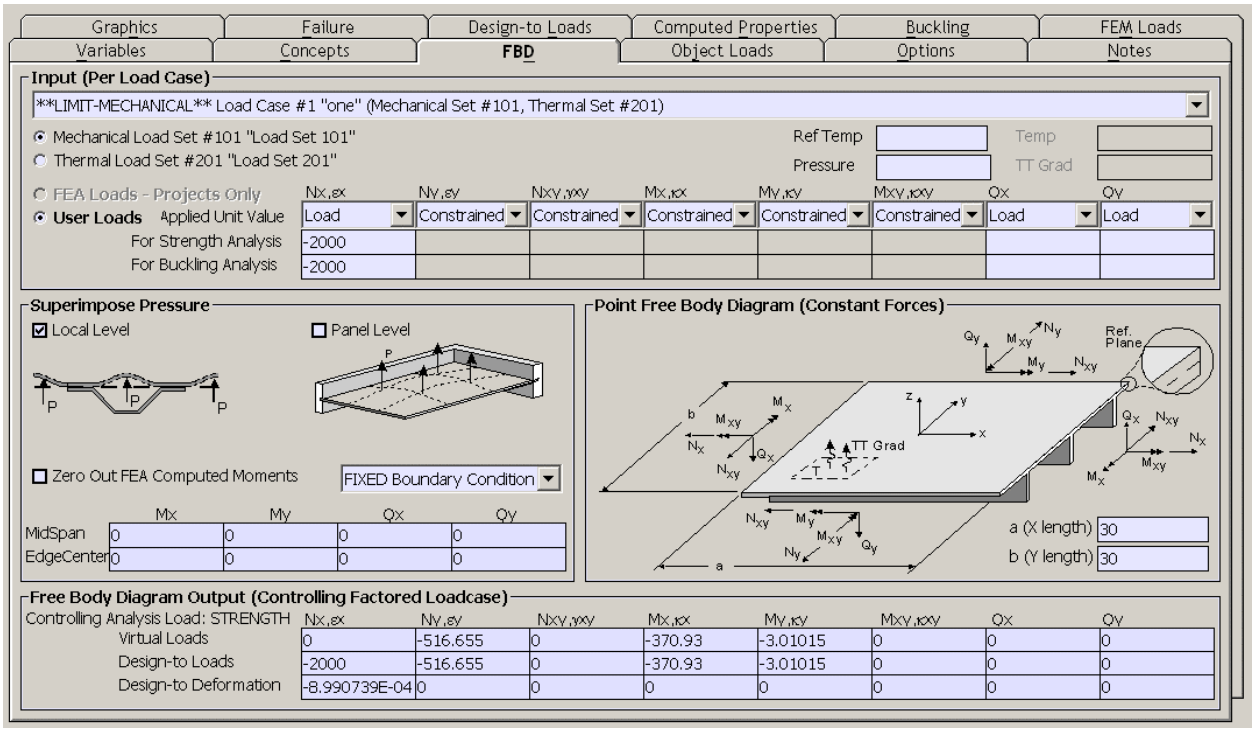
	Design	1
Unit Weight		2.91
Top Face - thickness		0.0936
Core Web - thickness		0.029

Bottom Face - thickness		0
Panel - height		1.713
Corrugation - spacing		5.9
Bottom Crown - width		1.328
Core Web - angle		72.91
Top Flange - width (hat only)		2
Top Clear Span - free width		1.594
Top Flange - thickness		0.029
Bottom Crown - thickness		0.029

Variable Design Materials

Variable		Materials
Top Face - thickness		1 - Titanium "Ex3002-Ti-6Al-4v, commonly used aerospace Titanium (UNS R56400)"
Core Web - thickness		1 - Titanium "Ex3002-Ti-6Al-4v, commonly used aerospace Titanium (UNS R56400)"

For this specific panel design, the HyperSizer user is able to enter the applied edge loading and edge boundary conditions. In this example a compressive load of –2000 (lb/in) is applied with constraints against rotation and translation of the other remaining edges. To enforce zero curvature and strain in the panel in these directions will cause virtual loads to be developed. In this case the virtual loads of Ny = -516.655, Mx = -370.93, and My = -3.01015 are developed. Note that in the direction of the applied membrane load, a strain of –8.990739E-04 is developed.

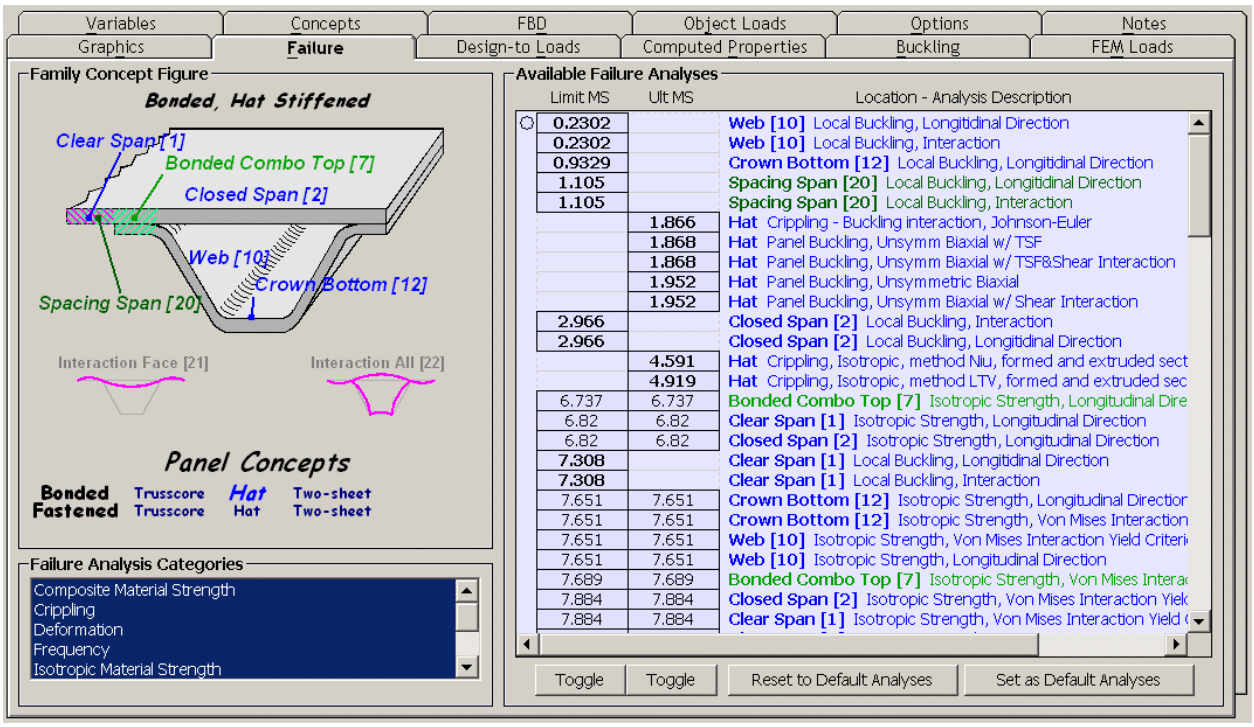


These resolved internal panel loads are then further refined to each of the defined analysis_objects that build up the panel. For this example we are interested in analysis_objects Clear Span [1] and Bonded Combo Top [7]. Their loads are shown in the figure below and their location on the panel is indicated by graphic in the failure tab.

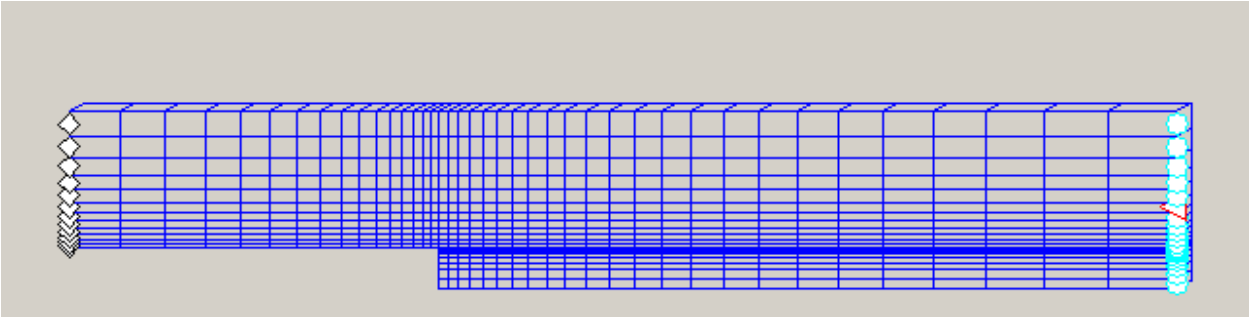
Load Results

Object	Nx	Ny	Nxy	Mx	My	Mxy	Temperatu...	TTG
Clear Span [1]	-1508.22	-516.655	0	0	0	0	72	0
Closed Span [2]	-1508.22	-516.655	0	0	0	0	72	0
Bonded Combo Top [7]	-1975.52	-516.655	0	-29.0754	-8.87994	0	72	0
Web [10]	-422.385	0	0	0	0	0	72	0
Crown Bottom [12]	-422.385	0	0	0	0	0	72	0

The failure tab shows margins-of-safety for the various failure modes that may occur for the stiffened panel. These failure analyses include metallic yield and laminated composite material strengths for the in-plane normal and shear directions. However, for regions such as the analysis_object Bonded Combo Top [7], they do not include interlaminar stresses nor peel stresses (Z axis effects). For this type of analysis the HyperSizer resulting state of stress and strain is passed to HOT-SMAC for more detailed analysis.

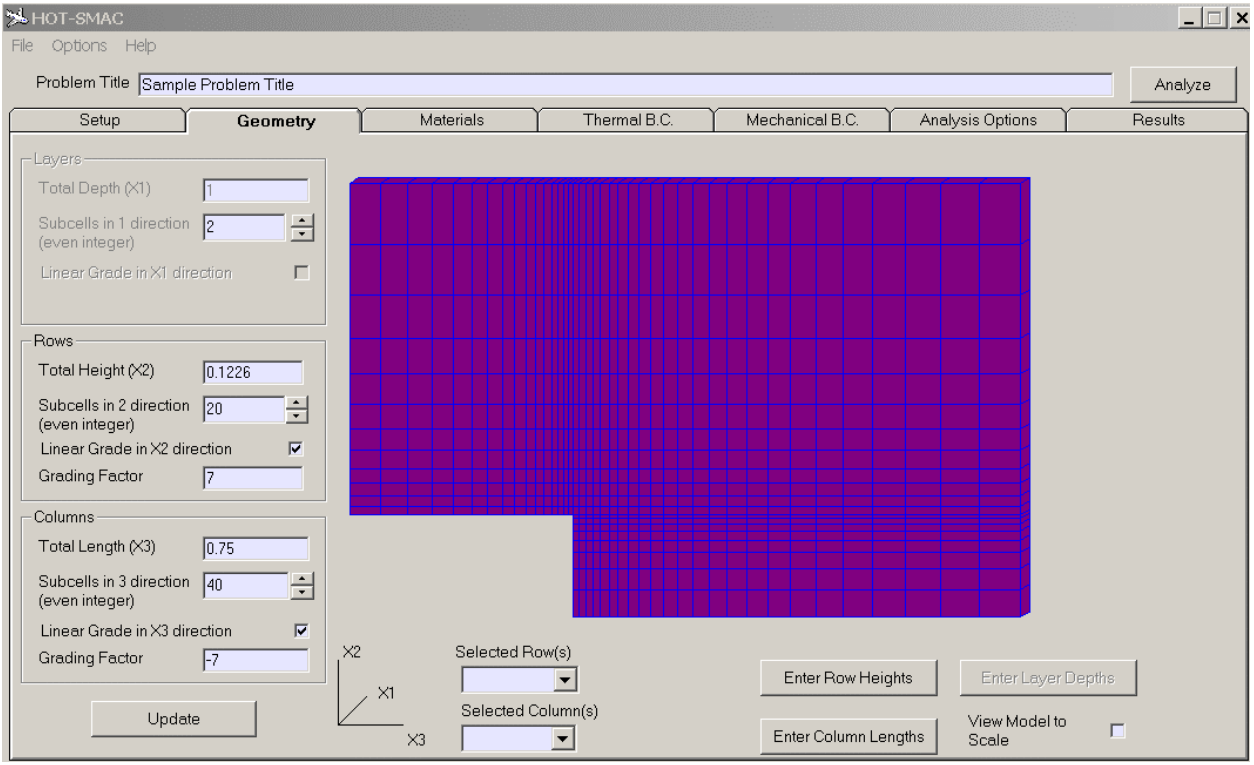


A HOT-SMAC model of cells was generated using the GUI developed under this CTO contract. The model is depicted below in relative scale proportion.



As can be seen in this image, the model is refined in the stressed areas of interest. Because of the difficulty in seeing the mesh refinement, the HOT-SMAC interface allows the user to exaggerate the model scale. This GUI feature is used in the remaining images of the HOT-SMAC model results.

The model has 20 cells in the vertical (X2) direction and 40 cells in the horizontal (X3) direction. A window was placed in the bottom left quadrant to represent the free edge feature of the flange line. The interface's feature of being able to automatically grade the mesh refinement was used to generate more cells in the region of the corner.

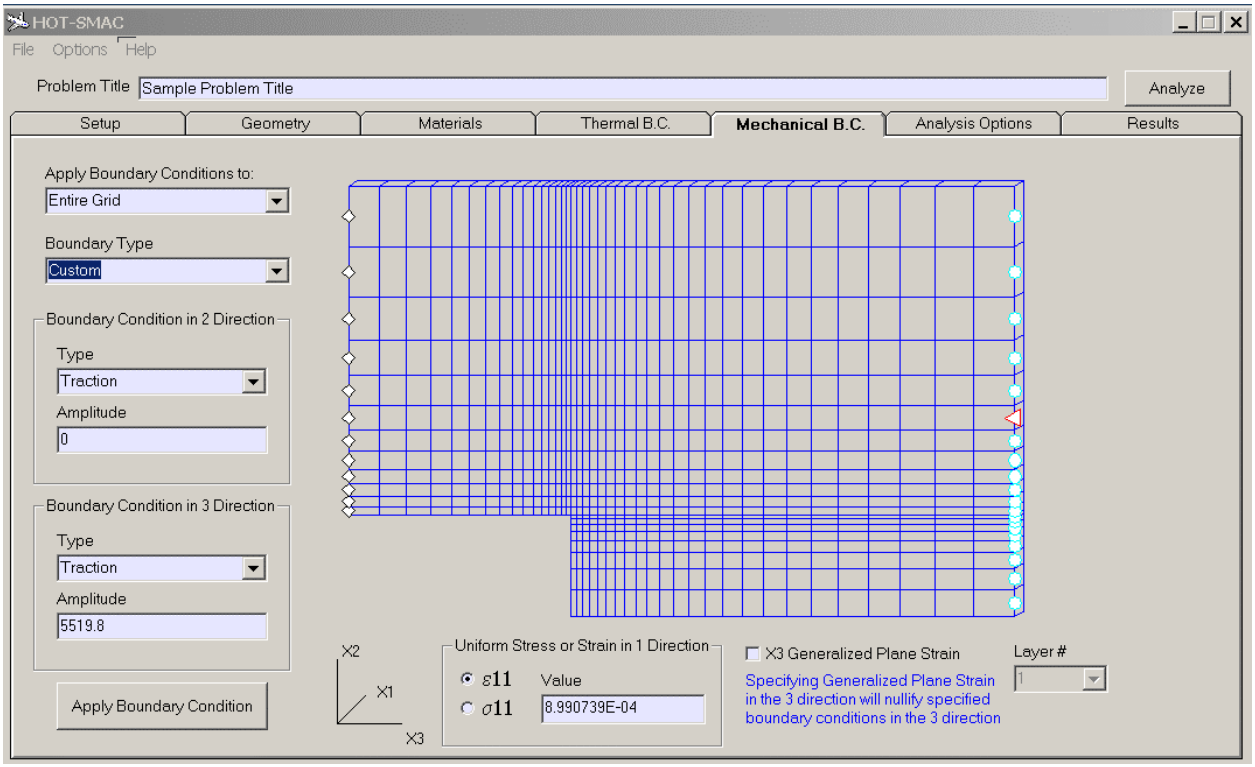


A symmetry boundary condition was idealized by using the roller boundary type on the centerline of the bond region, represented with the circles on the right hand side. The red triangle is a pinned b.c. and is added to prevent solution rigid body motion.

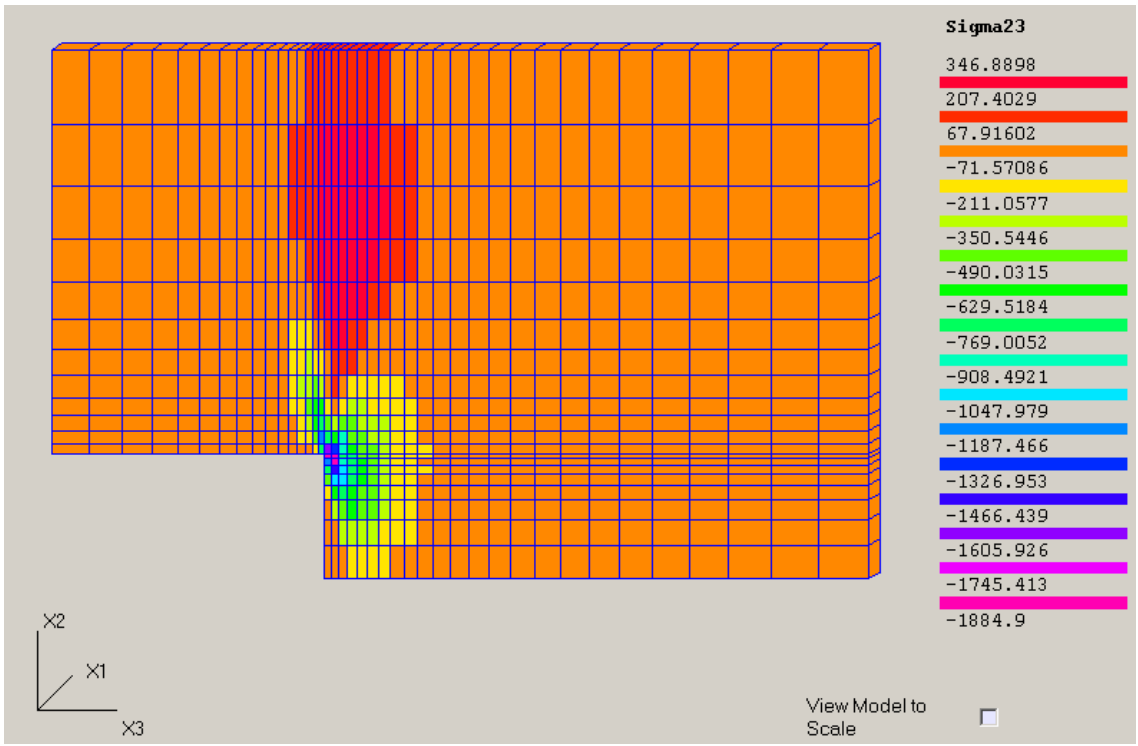
The HyperSizer computed stress was applied uniformly to the left hand side of the model as represented with the black triangles (custom boundary condition). The HOT-SMAC terminology for this applied loading is traction for stress and amplitude for magnitude.

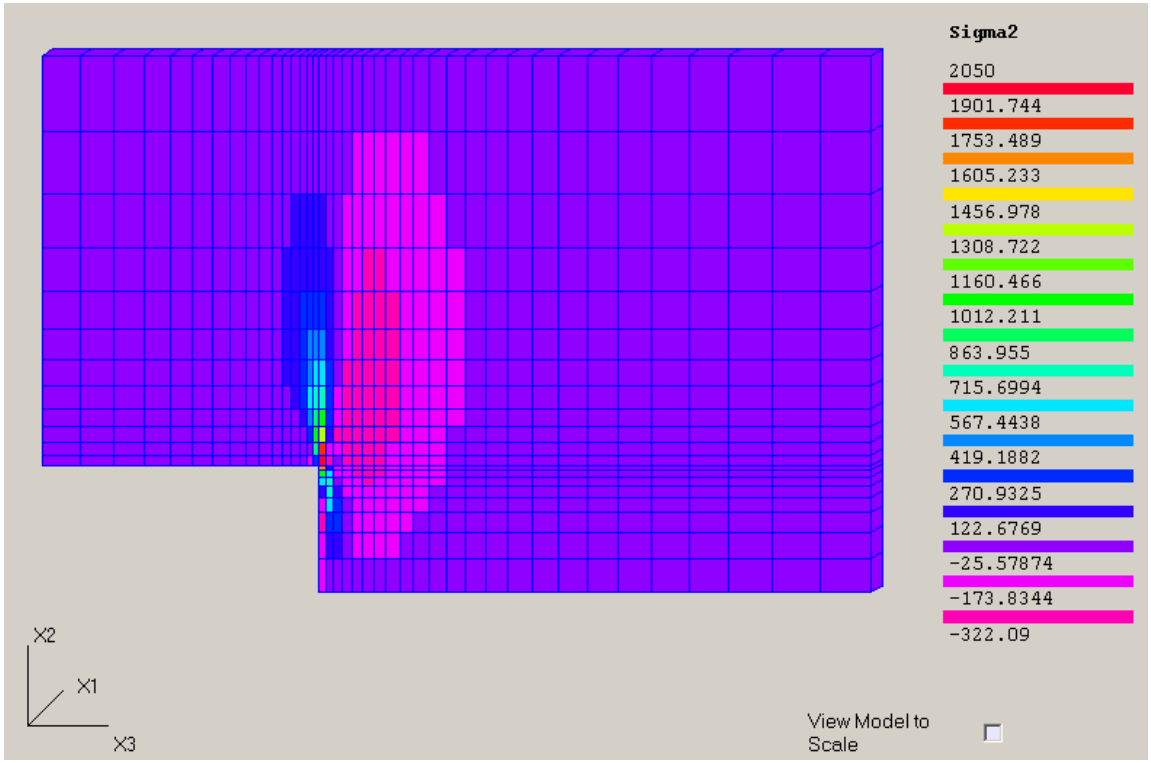
This traction load is calculated by (Ny/skin thickness) = 516.655 / 0.0936 = 5519.8 psi.

The cross sectional face (1 direction) is loaded in uniform strain with a magnitude of 8.990739E-04 that was computed by HyperSizer. Note that the HyperSizer solution was for a compressive load, but the computed results from HyperSizer were reversed when entered into HOT-SMAC so that tensile peel stress would develop.



For this loading and boundary conditions the following values of the stresses are computed with HOT-SMAC.





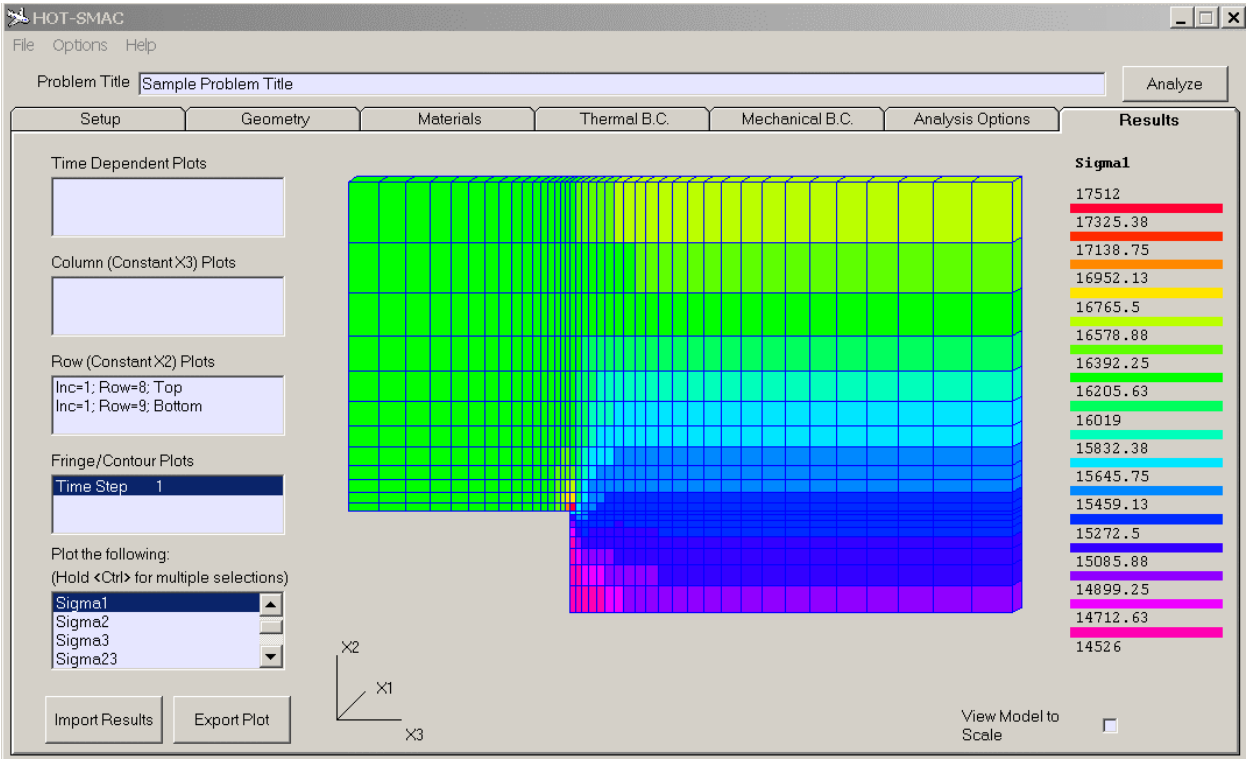
As a check, the resulting value of stress computed by HOT-SMAC (in the 1 direction) is verified with the stress computed by HyperSizer (in the panel X direction).

The left side of the model shows a nearly uniform stress of 16,115psi, as noted in green. The right hand side of this model shows a stress gradient ranging from about 16,700 psi to 15,000 psi with the integrated average that appears to be close to about 16,115psi.

This compares well to HyperSizer's calculation of stress.

The left side has a thickness of 0.0936" and the thicker part a thickness of 0.1226". As can be seen from the image of the HyperSizer Objects Load tab, the Nx value for the thin region is 1508.72 (lb/in) and Nx for the thick region is 1975.52 (lb/in).

1508.72 / 0.0936 = 16,119 psi and 1975.52 / 0.1226 = 16,114 psi.



References

1. Arnold, S. Bednarcyk, B. Aboudi, J., "Thermo-Elastic Analysis of Internally Cooled Structures Using a Higher Order Theory", NASA/TM 2001-210702, January 2001.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (*Leave blank*)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	November 2001	Final Contractor Report

4. TITLE AND SUBTITLE

User's Manual and Final Report for HOT-SMAC GUI Development

5. FUNDING NUMBERS

WU–251–30–07–00
NAS3–00159

6. AUTHOR(S)

Phil Yarrington

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

Collier Research and Development Corporation
45 Diamond Hill Road
Hampton, Virginia 23666

8. PERFORMING ORGANIZATION REPORT NUMBER

E–13088

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Washington, DC 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

NASA CR—2001-211294

11. SUPPLEMENTARY NOTES

Funding provided by Commercial Technology Office. Project Manager, Steven M. Arnold, Structures and Acoustics Division, NASA Glenn Research Center, organization code 5920, 216–433–3334.

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Categories: 39 and 24 Distribution: Nonstandard

Available electronically at http://gltrs.grc.nasa.gov/GLTRS

This publication is available from the NASA Center for AeroSpace Information, 301–621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

A new software package called Higher Order Theory—Structural/Micro Analysis Code (HOT-SMAC) has been developed as an effective alternative to the finite element approach for Functionally Graded Material (FGM) modeling. HOT-SMAC is a self-contained package including pre- and post-processing through an intuitive graphical user interface, along with the well-established Higher Order Theory for Functionally Graded Materials (HOTFGM) thermomechanical analysis engine. This document represents a Getting Started/ User's Manual for HOT-SMAC and a final report for its development. First, the features of the software are presented in a simple step-by-step example where a HOT-SMAC model representing a functionally graded material is created, mechanical and thermal boundary conditions are applied, the model is analyzed and results are reviewed. In a second step-by-step example, a HOT-SMAC model of an actively cooled metallic channel with ceramic thermal barrier coating is built and analyzed. HOT-SMAC results from this model are compared to recently published results (NASA/TM—2001-210702) for two grid densities. Finally, a prototype integration of HOT-SMAC with the commercially available HyperSizer® structural analysis and sizing software is presented. In this integration, local strain results from HyperSizer's structural analysis are fed to a detailed HOT-SMAC model of the flange-to-facesheet bond region of a stiffened panel. HOT-SMAC is then used to determine the peak shear and peel (normal) stresses between the facesheet and bonded flange of the panel and determine the "free edge" effects.

14. SUBJECT TERMS

Thermal analysis; Deformation analysis; Elastic; Plastic; Creep; Computer science; Software

15. NUMBER OF PAGES

61

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	