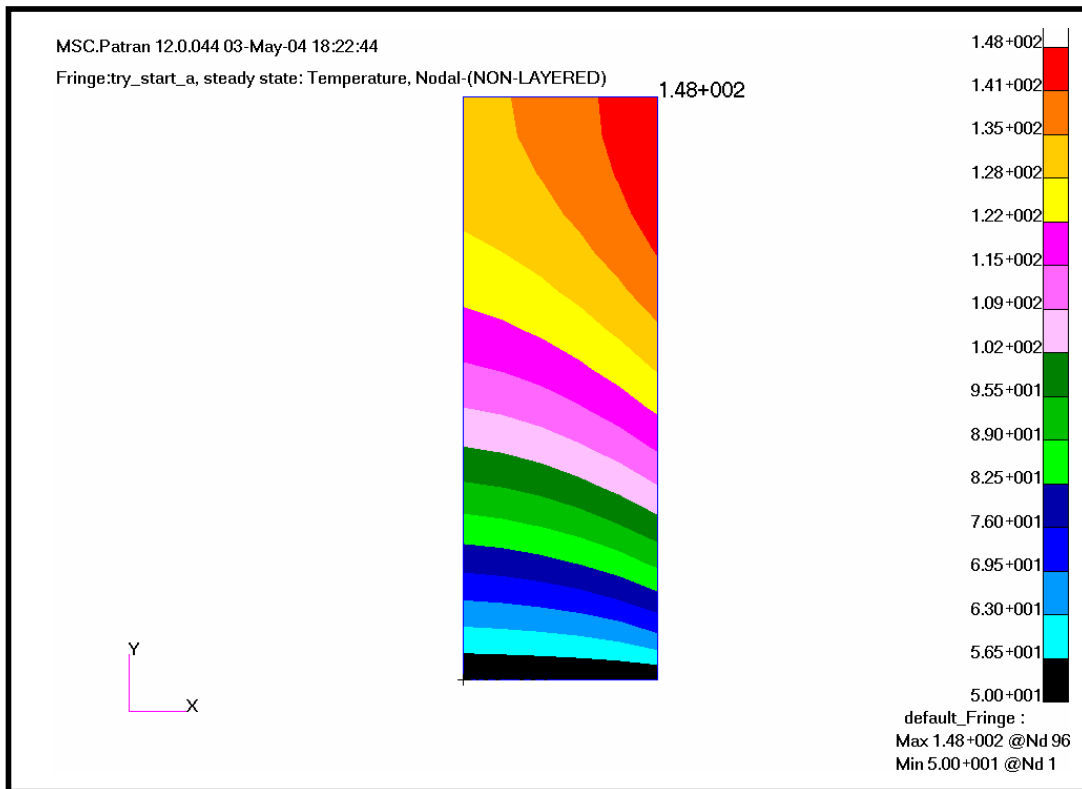


Getting Started Sinda/G for Patran



Objectives:

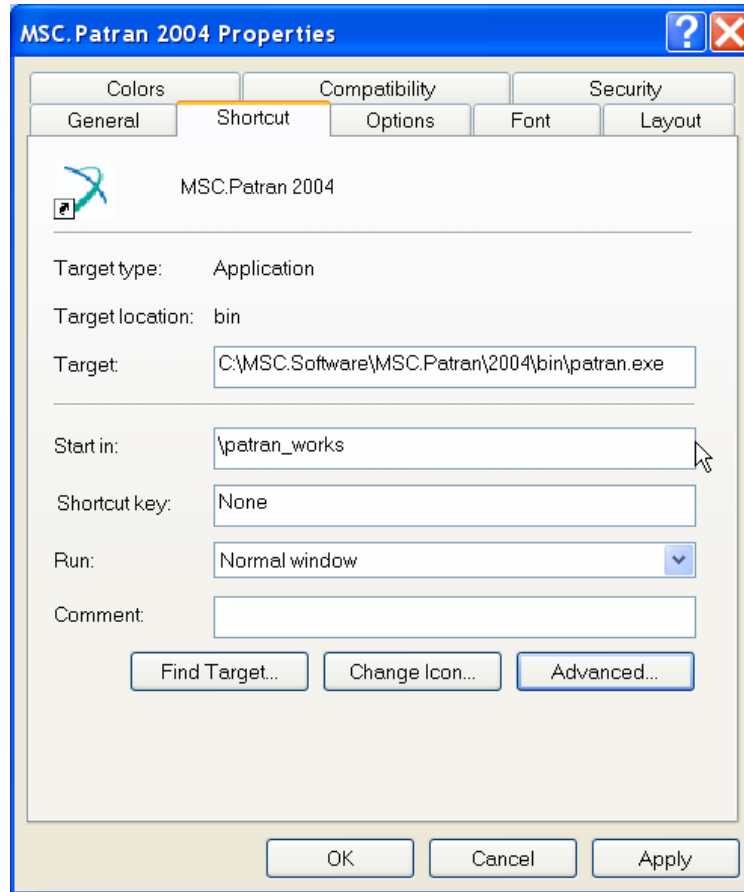
- Before getting started creating a model.
- Create a new model for steady state thermal analysis.
- Modify the model for transient state thermal analysis.
- Sinda/G for Patran QA test and example models.

Getting Started - SINDA/G for PATRAN

Before getting started creating a model

● If you are running Sinda/G for Patran on PC

1. Right click the Patran icon on the PC desktop, then a window pops up. On the bottom of the window, select Properties, the MSC.Patran properties window pops out as follows:



2. Modify the path in the **start in** data box. This directory is where you read or save your Patran models and all relative files. This directory can be any existing directories, such as **C:\patran_works**
3. If you want to see the status of Sinda/G compiling and running, you can add an option at the end of **Target** data box: **C:\MSC.Software\MSC.Patran\2004\bin\patran.exe -stdout**
4. If you had some customized settings in p3epilog.pcl before Sinda/G for Patran installation, you need to add them manually. The original p3epilog.pcl is renamed to p3epilog_old.pcl.

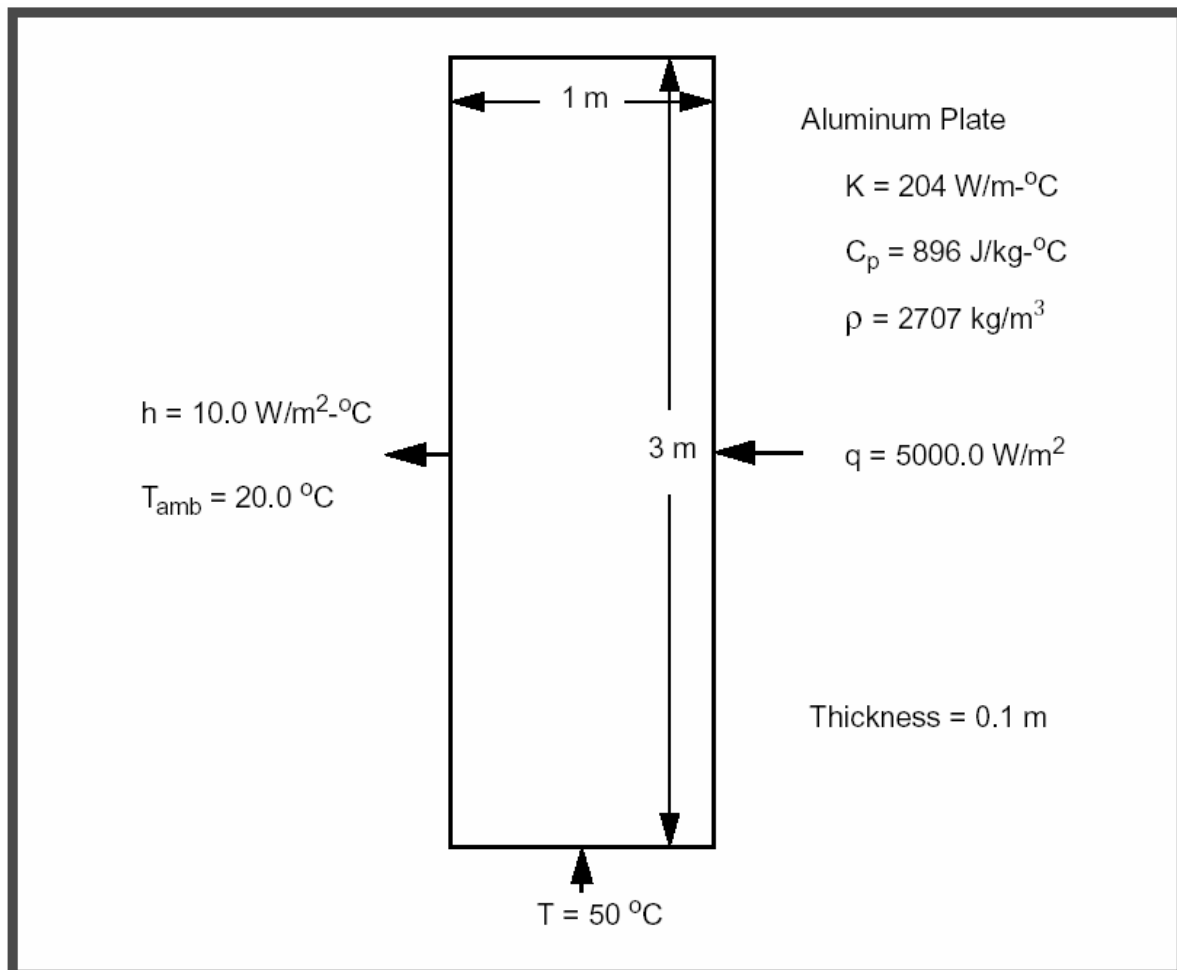
● If you are running Sinda/G for Patran on Unix

1. The Unix version links to external radiation codes (like Thermica, Nevada...) are developing and coming soon.
2. Each user can save his own default Sinda/G settings in user_account_directory/**sindaprefs/nas2sg.ini**.

Create a new model for steady state thermal analysis

Model Description:

In this exercise you will first create an aluminum plate. Shown below is a drawing of the model you will be building and suggested steps for its construction.



Getting Started - SINDA/G for PATRAN

Suggested Exercise Steps:

- Create a new database defined for Sinda/G for Patran thermal analysis.
- Define geometry for a rectangular plate.
- Mesh the surface with quadrilateral elements.
- Define the plate's material as aluminum. Specify a thermal conductivity of 204 W/m · °C, specific heat of 896 J/kg · °C, and a density of 2707 kg/m³.
- Define the plate's property as 2D shell with a thickness 0.1 m.
- Apply a temperature of 50°C to the bottom edge of the plate.
- Apply heat flux of 5000 W/m² to the right edge of the plate.
- Apply to the left edge of the surface a convection boundary condition with heat transfer coefficient of 10.0 W/m²·°C and ambient temperature of 20°C.
- Set the thermal solution setup and Sinda/G option for steady state analysis
- Save the current settings as the default, and check the nas2sg.ini file.
- Perform a steady-state thermal analysis using Sinda/G for Patran.
- Check the result files in Edit/Manager Files form.
- Read the .nrf result file into Sinda/G for Patran
- Visualize the temperature distribution in the post procession.

Getting Started - SINDA/G for PATRAN

Exercise Procedure:

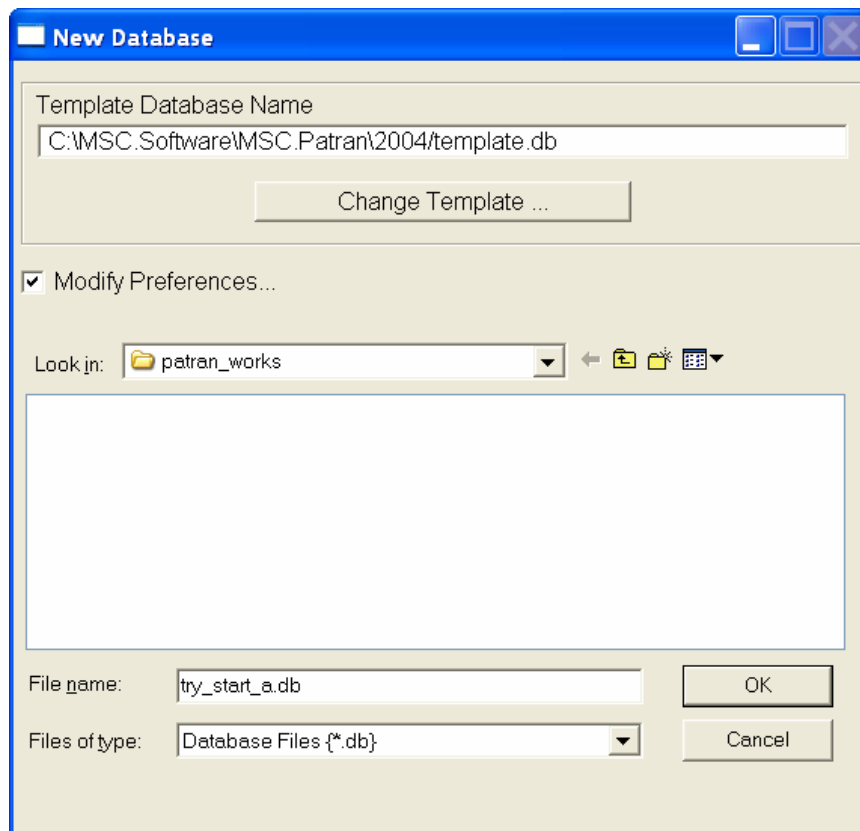
1. Open a new database. Name it **try_start_a.db**

File/New...

New Database Name:

try_start_a.db

OK



The viewport (PATRAN's graphics window) will appear along with a **New Model Preference** form. The **New Model Preference** sets all the code specific forms and options inside MSC.PATRAN.

In the **New Model Preference** form set the *Analysis Code* to **SINDA/G**

Tolerance:

◆ **Based on Model**

Approximate Maximum Model Dimension:

10.0

Analysis Code:

SINDA/G

Getting Started - SINDA/G for PATRAN

Analysis Type:

Thermal

OK

2. Create the Model

◆ Geometry

Action:

Create

Object:

Surface

Method:

XYZ

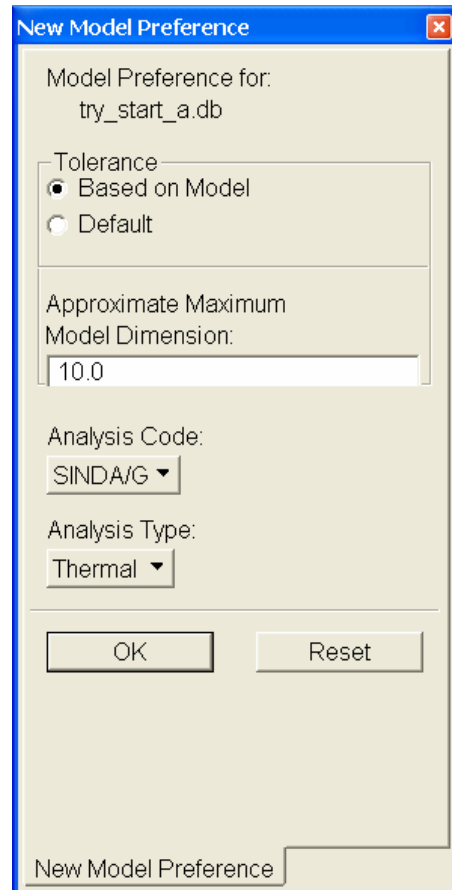
Vector Coordinates List:

<1 3 0>

Origin Coordinates List:

[0 0 0]

Apply



3. Mesh the surface with elements

◆ Elements

Action:

Create

Object:

Mesh

Type:

Surface

Mesher:

IsoMesh

Surface List:

Surface 1

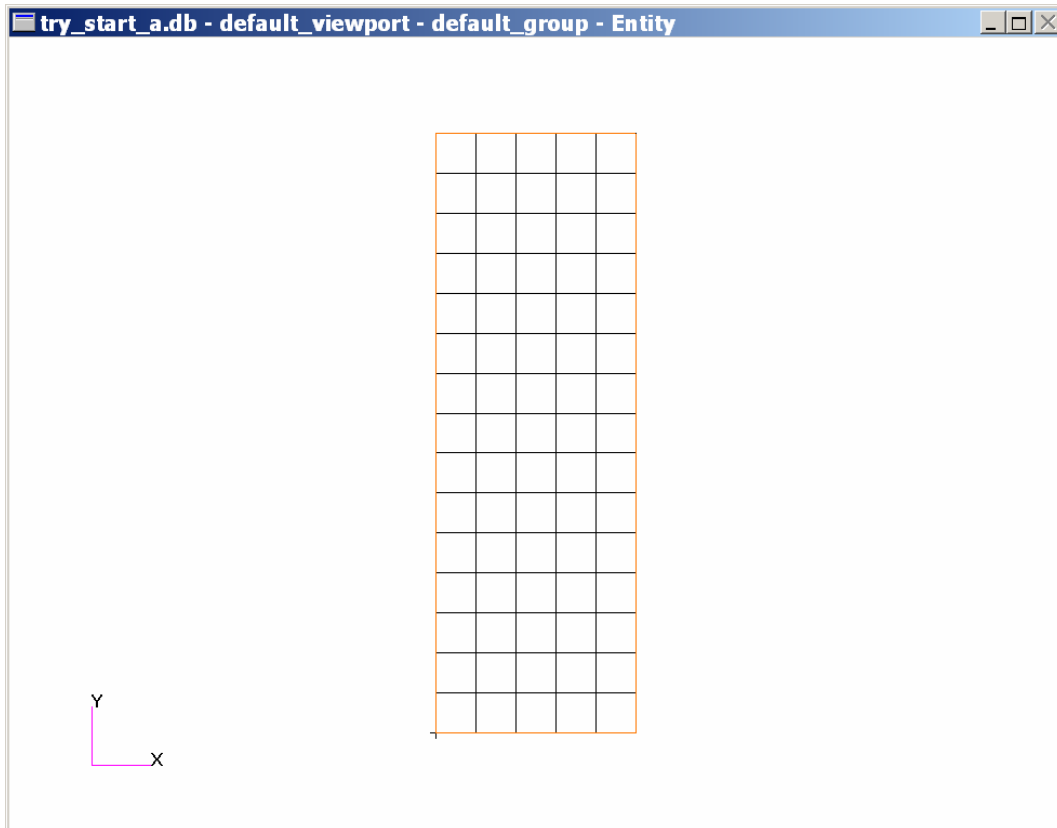
Global Edge Length:

0.2

Apply

Your model should look like the following figure.

Getting Started - SINDA/G for PATRAN



4. Specify the Material

Our material for this model will be aluminum. Click on the **Materials** button. The **Material** form will appear with certain default options.

◆ Materials

Action:

Create

Object:

Isotropic

Method:

Manual Input

Material Name:

alum

Input Properties...

Thermal Conductivity:

204

Specific Heat:

896

Density:

2707

Getting Started - SINDA/G for PATRAN

OK

Apply

5. Specify the Property

◆ Properties

Action:

Create

Object:

2D

Type:

Shell

Property Set Name:

plate

Input Properties...

Material Name:

m:alum

Thickness:

0.1

OK

From the **Element Properties** form, click on the Select Members data box. MSC.PATRAN will display 4 icons to the left of the **Element Properties** form. The first icon represents **surface or face**, the others represents **2D elements**. The 4 options allow you to apply properties either on the geometric entity (in this case, the surface) or on the finite elements.

Click on the **Surface or Face** icon.



Surface or Face

Now click anywhere on the geometric surface. The surface will be highlighted in red. The Select Members databox will now appear as *Surface 1*.

Add

Apply

6. Apply the load and boundary conditions

◆ Load/BCs

Getting Started - SINDA/G for PATRAN

| | |
|---|--|
| Action: | <input type="text" value="Create"/> |
| Object: | <input type="text" value="Temp(Thermal)"/> |
| Type: | <input type="text" value="Nodal"/> |
| New Set Name: | <input type="text" value="tempbc"/> |
| <input type="button" value="Input data..."/> | |
| Temperature: | <input type="text" value="50"/> |
| <input type="button" value="OK"/> | |
| <input type="button" value="Select Application Region..."/> | |

Geometry Filter: ◆ **Geometry**

Click on the **Curve or Edge** icon.



Curve or Edge

With your mouse, position the cursor on the bottom edge of the surface. Click on the edge. You will see *Surface 1.4* appear in the **Select Geometry Entities** databox. This means we have selected Edge number 4 in *Surface number 1*.

| |
|--------------------------------------|
| <input type="button" value="Add"/> |
| <input type="button" value="OK"/> |
| <input type="button" value="Apply"/> |

7. Now apply heat flux to the model using the Loads/Boundary Conditions form.

◆ **Load/BCs**

| | |
|---------|--|
| Action: | <input type="text" value="Create"/> |
| Object: | <input type="text" value="Applied Heat"/> |
| Type: | <input type="text" value="Element Uniform"/> |

Getting Started - SINDA/G for PATRAN

Option:

Normal Fluxes

New Set Name:

flux

Target Element Type:

2D

Because the model is a 2D plate, we need to toggle that **Target Element Type** setting to 2D. Even though we are applying heat flux along an edge, which we normally think of as 1D, our finite element type is 2D; i.e., we are modeling heat conduction in two dimensions.

Input data...

Form Type:

Basic

Surface Option:

Edge

Edge Heat Flux:

5000

OK

Select Application Region...

Geometry Filter:

◆ Geometry

Click on the **Edge** icon.



Edge

Position the cursor over the right edge of the surface and click on this edge with the mouse. MSC.PATRAN will insert *Surface 1.3* in the data box under the heading Select Surfaces or Edges.

Add

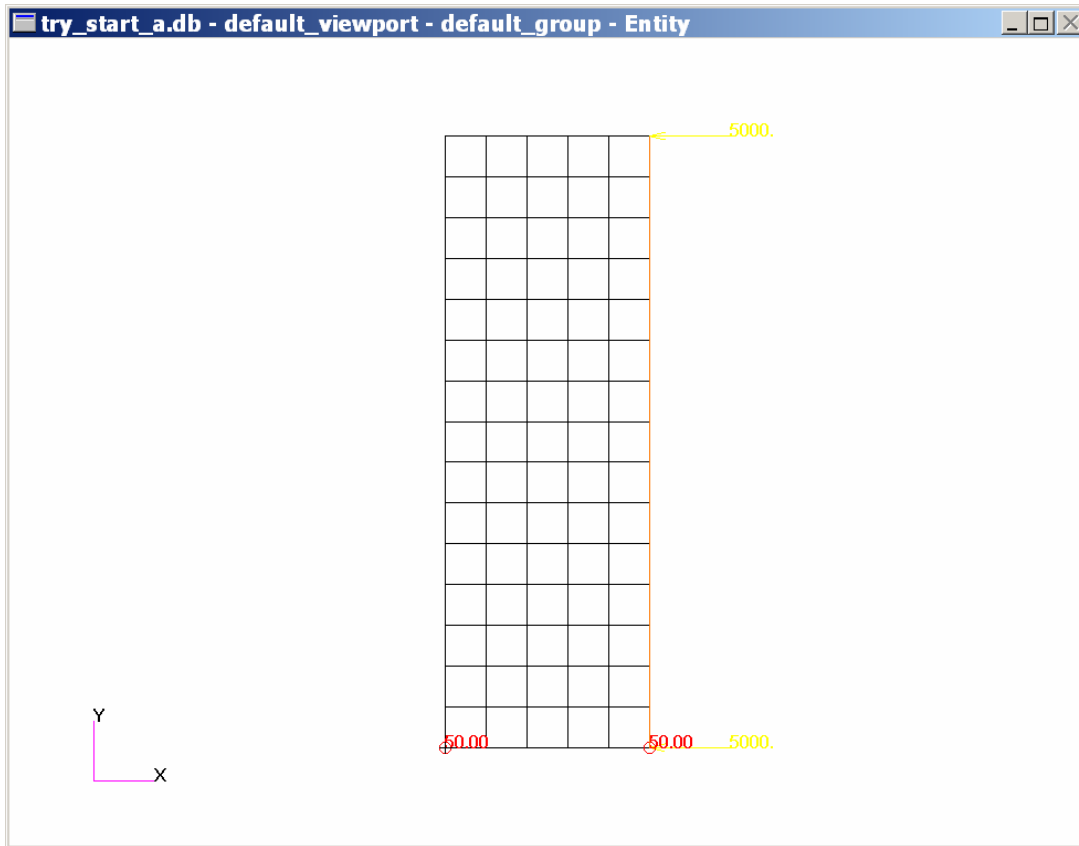
OK

Apply

A yellow flag will appear on the right edge of your surface indicating that a heat flux of 5000 W/m² has been applied along the right edge.

Your model should look like the following figure.

Getting Started - SINDA/G for PATRAN



8. Apply a convection boundary condition to the left edge of the plate-- again, using the **Loads/BCs** form.

◆ **Load/BCs**

Action:

Create

Object:

Convection

Type:

Element Uniform

Option:

To Ambient

New Set Name:

conv

Target Element Type:

2D

Input data...

Surface Option:

Edge

Getting Started - SINDA/G for PATRAN

Form Type:

Basic

Edge Convection Coef:

10

Ambient Temperature:

20

OK

Select Application Region...

Geometry Filter:

◆ Geometry

Click on the **Edge** icon.



Edge

Position the cursor over the left edge of the surface and click on the edge with the mouse. MSC.PATRAN will insert *Surface 1.1* in the data box under **Select Surfaces** or **Edges**

Add

OK

Apply

A green label will appear confirming that you have applied a convection coefficient of 10.0W/m²·°C at this location of your model.

Your model should look like the figure on the next page

9. We are now ready to submit the model for Sinda/G steady state thermal analysis. Click on the **Analysis** application located on the MSC.PATRAN main form.

◆ Analysis

Action:

Analyze

Object:

Entire Model

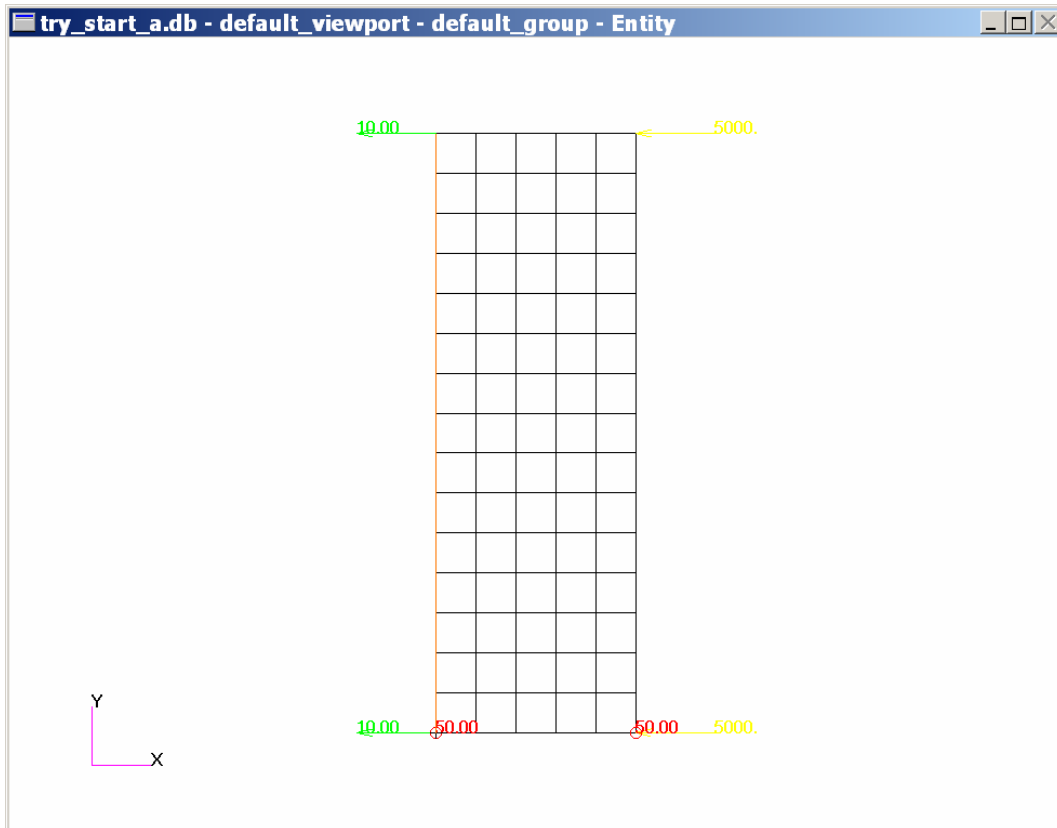
Method:

Translate and Run

Job Name:

try_start_a

Getting Started - SINDA/G for PATRAN



Thermal Solution Setup...

Steady State Setup(SNSOR)

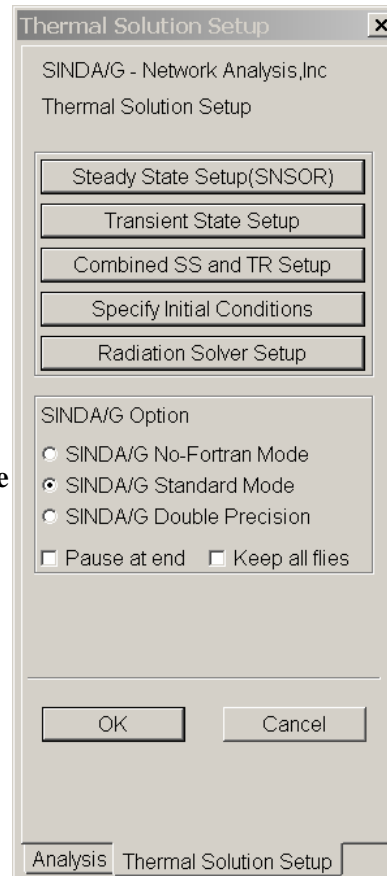
Choose Solution Routine: ◆ **SNSOR**

OK

SINDA/G Option: ◆ **SINDA/G Standard Mode**

OK

Apply



Getting Started - SINDA/G for PATRAN

Note: after you hit the “Apply” button

- If you run this model again with the same job name, some small windows will pop out to ask if you want to re-run this model, if the old files are overwritten, or deleted, just answer Yes.
- If you run this model on PC, some status windows will pop out to show the status of pre-processing and Sinda/G calculation.
- The Analysis Form may flash during the analysis. When you find “**Translation and Sinda/G Model Complete**” in Patran command window, the translation and analysis is completed. You can also view the Patran output window to see the status of Sinda/G analysis. For PC, you need to set the **-stdout** in the target data box in MSC.Patran properties form (see page 2), for Unix, you can see these messages in the window where you started Patran.

10. Check the result files in Edit/Manager Files Form

Edit/Manage Files...

In the **Edit/Manage File** Form, you can edit/view any files, including the SINDA/G input file (.sin), result file (.sot), etc. The default file editor can be changed to your preference.

All the settings in Analysis Form and its sub-forms will be saved in SINDA/G configuration file (.ini & ini2). Next time, when you open this model, these settings will be automatically recovered. If you are satisfied with the current settings, you can save them in the default configuration file (**nas2sg.ini**). It is recommended to do so when you are creating your first model. The **nas2sg.ini** file will be created or updated by clicking the following icon inside **Edit/Manager Files** form.

Save Settings as Default...

Note: about the default configuration file (**nas2sg.ini**):

- If you are running SINDA/G for Patran on a PC, the default configuration file **nas2sg.ini** will be saved in Patran home directory, such as C:\MSC.Software\MSC.Patran\2004\.
- If you are running SINDA/G for Patran on an Unix system, each user can have his own **nas2sg.ini** file which is saved in user_account_home/sindaprefs/.
- SINDA/G for Patran will search the configuration file in the following order:
 - 1) job_name.ini2
 - 2) db_name.ini2
 - 3) nas2sg.ini
 - 4) program built-in configuration file

11. Read the .nrf result file into SINDA/G for Patran

◆ Analysis

Action:

Access Result

Object:

Result Entities

Getting Started - SINDA/G for PATRAN

Method:

Translate

Job Name:

try_start_a

Apply

In the Patran command line, you will find “**Result file imported into Patran**”. If you want to read the newest result file that you just had, you do not need to hit the “Select Result file...” button to select the result file. Just click the “**Apply**” button, job_name.nrf file will be read into Patran by default.

12. Display the Result

◆ Results

Action:

Create

Object:

Quick Plot

Select Result Cases:

try_start_a, steady state

Select Fringe Result:

Temperature, nodal

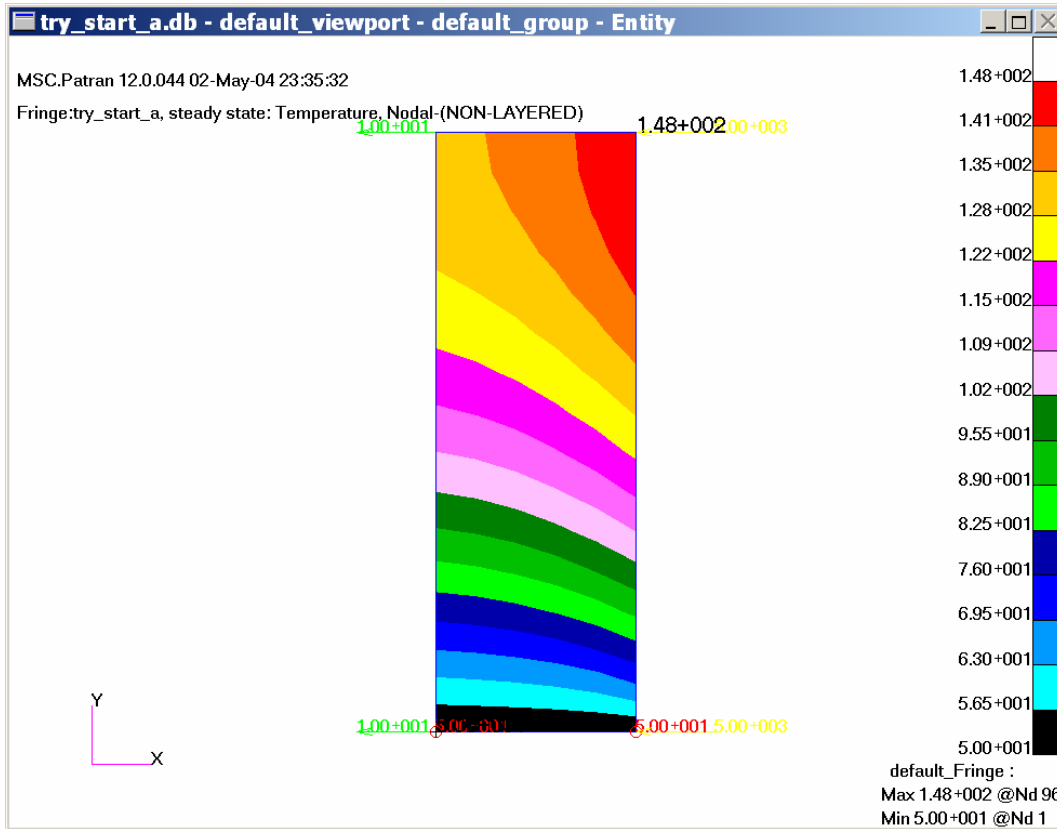
Apply

A contour plot displaying temperature distributions will appear. Your model should look like the following figure.

13. Close database and quit MSC.Patran to complete this exercise.

File/Quit...

Getting Started - SINDA/G for PATRAN

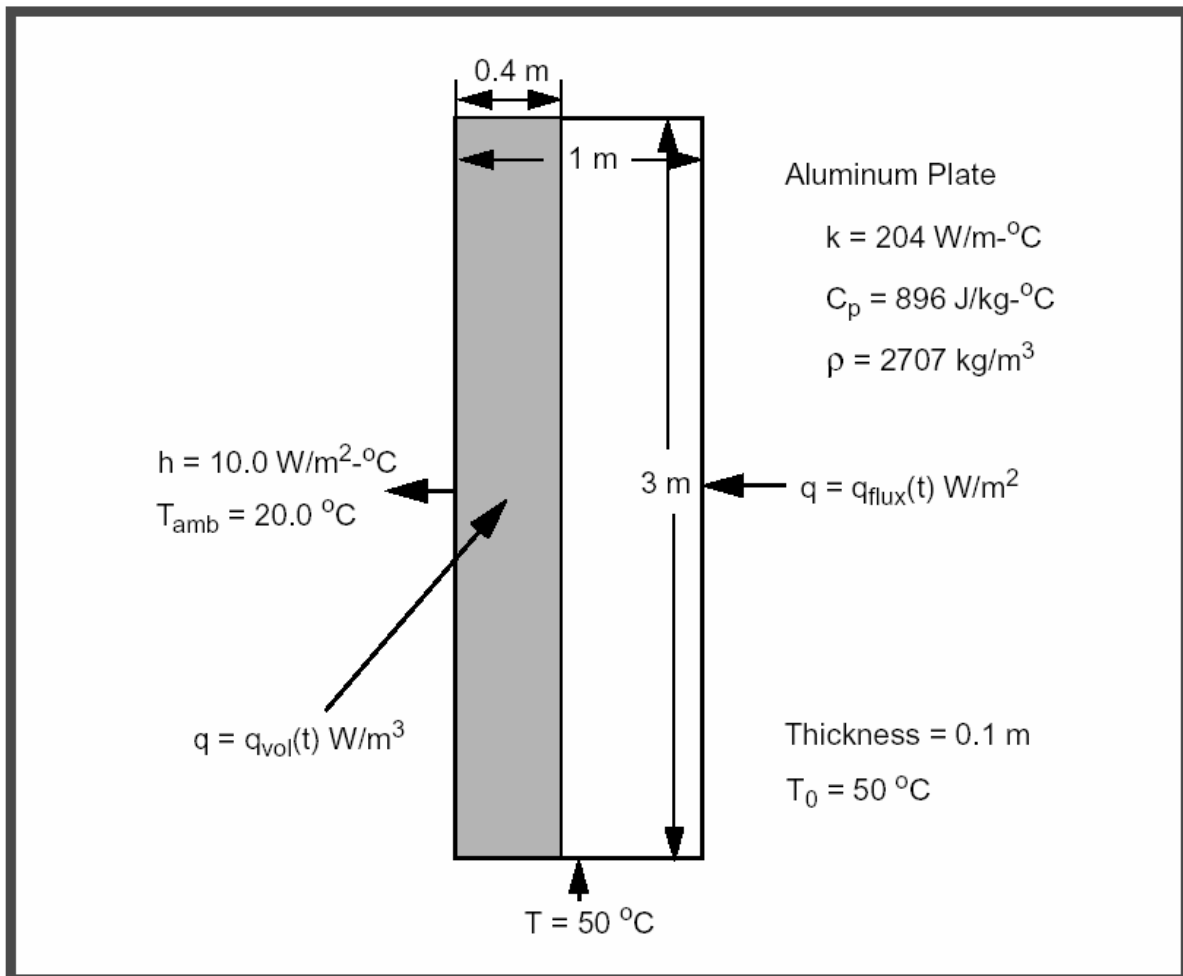


Modify the model for transient state thermal analysis

Model Description:

This exercise describes transient thermal analysis, it is an extension of the steady state modeling exercise given in the above exercise. This contains step-by-step descriptions of the menu picks involved in the modeling process.

Shown below is a drawing of the model you will be building and suggested steps for its construction



Getting Started - SINDA/G for PATRAN

Suggested Exercise Steps:

- Open the database try_start_a.db which was created in the above exercise.
- Define time dependent functions using the Field application.
- Create a transient load case. Add two existing load sets (temperature and convection boundary conditions) to this transient load case.
- Apply time varying heat flux to the right edge of the plate.
- Apply a transient volumetric heat generation inside the shaded area of the plate.
- Set the transient state solution setup.
- Specify the default initial temperature.
- Perform a transient thermal analysis using SINDA/G within the MSC.PATRAN system.
- Postprocess the transient results (Contour and Animation).

Getting Started - SINDA/G for PATRAN

Exercise Procedure:

1. Open the database **try_start_a.db**

File/Open...

File name:

try_start_a.db

OK

2. Define Time Dependent Functions

Before applying time varying loads and boundary conditions, we need to define time dependent functions using the Field application. In this model, two time fields are defined, one for applied heat flux and one for volumetric heat generation.

◆ Fields

Action:

Create

Object:

Non Spatial

Method:

Tabular Input

Field Name:

flux_time

Input data...

Fill in the table with the following values using the RETURN or ENTER key.

OK

Apply

| | Time(t) | Value |
|---|---------------|---------------|
| 1 | 0.000000E+000 | 1.000000E+000 |
| 2 | 1.000000E+001 | 1.250000E+000 |
| 3 | 3.000000E+001 | 1.750000E+000 |
| 4 | 5.000000E+001 | 2.000000E+000 |
| 5 | 1.000000E+002 | 2.000000E+000 |
| 6 | | |
| 7 | | |
| 8 | | |

Similarly, a time dependent function for volumetric heating is defined as follows.

◆ Fields

Action:

Create

Object:

Non Spatial

Getting Started - SINDA/G for PATRAN

Method:

Tabular Input

Field Name:

qvol_time

Input data...

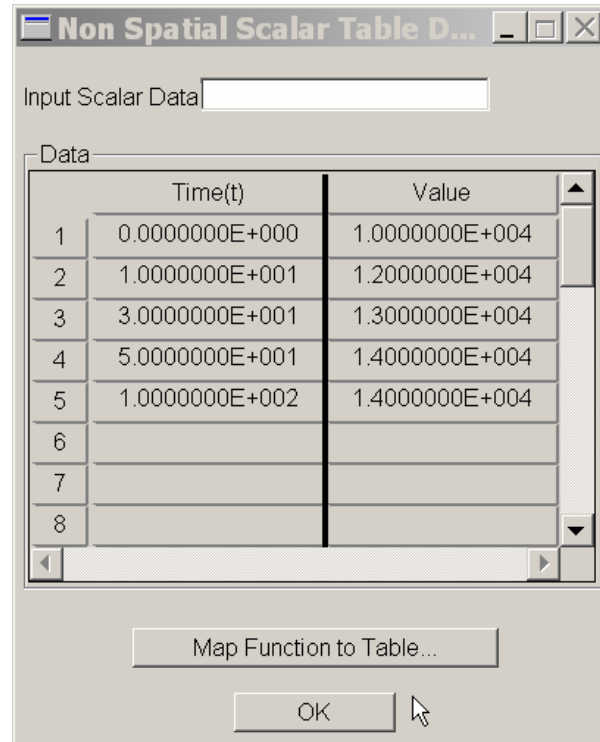
Fill in the table with the following values using the RETURN or ENTER key.

OK

Apply

3. Create a transient load case for transient load boundary conditions.

Since the temperature and convection boundary conditions are not changed, we can associate these two load sets with the new load case directly.



◆ Load Cases

Action:

Create

Load Case Name:

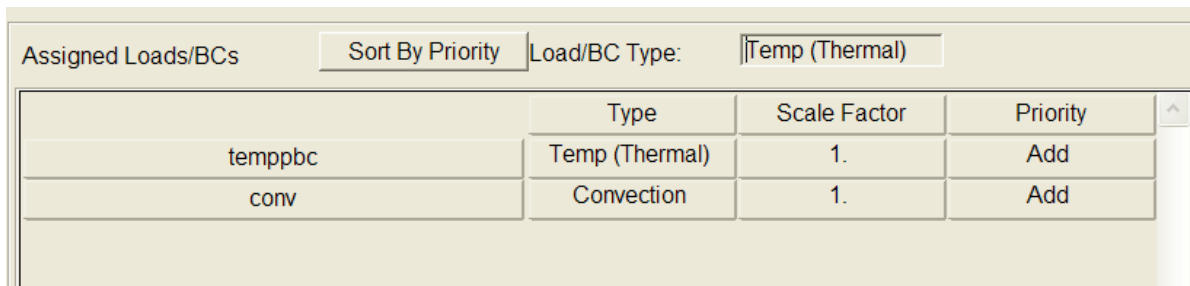
transient

Load Case Type:

Time Dependent

Assign/Prioritize Loads/BSs...

Highlight **Conve_conv** and **Temp_tempbc** within the *Select Individual Loads/BCs Sets* listbox. The middle part of the load case window should look like the follows.



Getting Started - SINDA/G for PATRAN

OK

Apply

At this point, we will impose a transient flux load on the plate's right edge. The magnitude of this flux load is 5000 W/m^2 multiplied by the time dependent function **flux_time** defined earlier under the Fields application. Click on the Loads/BCs application.

4. Apply time varying heat flux using the Loads/Boundary Conditions form.

◆ **Load/BCs**

Action:

Create

Object:

Applied Heat

Type:

Element Uniform

Option:

Normal Fluxes

New Set Name:

tran_flux

Target Element Type:

2D

Input data...

Form Type:

Basic

Surface Option:

Edge

Edge Heat Flux:

5000

Time Function:

f:flux_time

OK

Select Application Region...

Geometry Filter:

◆ **Geometry**

Click on the **Edge** icon.

Getting Started - SINDA/G for PATRAN



Edge

Position the cursor over the right edge of the surface and click on the edge with the mouse. MSC.PATRAN will insert *Surface 1.3* in the data box under **Select Surfaces** or **Edges**

Add

OK

Apply

5. Apply Transient Volumetric Heat Generation Inside the Plate.

The volumetric heating can be applied in a similar way, using the Loads and Boundary Conditions form as follows.

◆ Load/BCs

| | |
|----------------------|-----------------------|
| Action: | Create |
| Object: | Applied Heat |
| Type: | Element Uniform |
| Option: | Volumetric Generation |
| New Set Name: | tran_qvol |
| Target Element Type: | 2D |
| Input data... | |
| Form Type: | Basic |
| Time Function: | f:qvol_time |

You do not have to input *Volumetric Heat Generation*. By default, SINDA/G for Patran will assign 1.0 to this data box if there is a time function available and this data box is empty.

OK

Getting Started - SINDA/G for PATRAN

Next, click on Select Application Region located on the **Loads and Boundary Conditions** form. We want to apply an internal heat generation inside a section of the plate from $x=0.0$ m to $x=0.4$ m. This application region will be selected by graphical cursor using the **FEM** geometry filter.

Select Application Region...

Geometry Filter:

◆ FEM

Use the mouse cursor to drag a rectangle covering the elements located between $x=0.0$ m and $x=0.4$ m. Release the mouse cursor. The first two columns of the elements will turn red indicating the selection. Also, a list of elements will appear in the **Select 2D Elements** databox.

Add

OK

Apply

Note: A square yellow marker will appear on the center of the selected element indicating that a volumetric heating has been applied on this element.

6. Now we are ready to set the analysis controls for transient thermal analysis.

◆ Analysis

Action:

Analyze

Object:

Entire Model

Method:

Translate and Run

Job Name:

try_start_b

Please note that a new job name **try_start_b** is input instead of **try_start_a**.

Thermal Solution Setup...

Transient State Setup

Choose Solution Routine: ◆ SNDUFR

Getting Started - SINDA/G for PATRAN

TIMEND:

1000

OUTPUT:

100

OK

7. Still in **Thermal Solution Setup** form, we will specify the default initial temperature for transient thermal analysis.

Specify Initial Conditions

Initial Temperature:

50

OK

SINDA/G Option: ◆ **SINDA/G Standard Mode**

OK

Apply

8. Read the .nrf result file into SINDA/G for Patran

◆ **Analysis**

Action:

Access Result

Object:

Result Entities

Method:

Translate

Job Name:

try_start_b

Apply

In the Patran command line, you will find “**Result file imported into Patran**”. Sinda/G for Patran will automatically distinguish the steady state and transient result. Again, if you want to read the newest result file that you just created, you do not need to hit the “Select Result file...” button. Just click the “**Apply**” button, job_name.nrf file will be read into Patran by default.

Getting Started - SINDA/G for PATRAN

9. Create a contour plot temperature distributions at $time=700$ sec using the **Result Display** form.

Before create the contour, Click on the **Reset Graphics** icon to clear the load marks on the figure.



Reset Graphics

◆ Results

Action:

Create

Object:

Quick Plot

Select Result Cases:

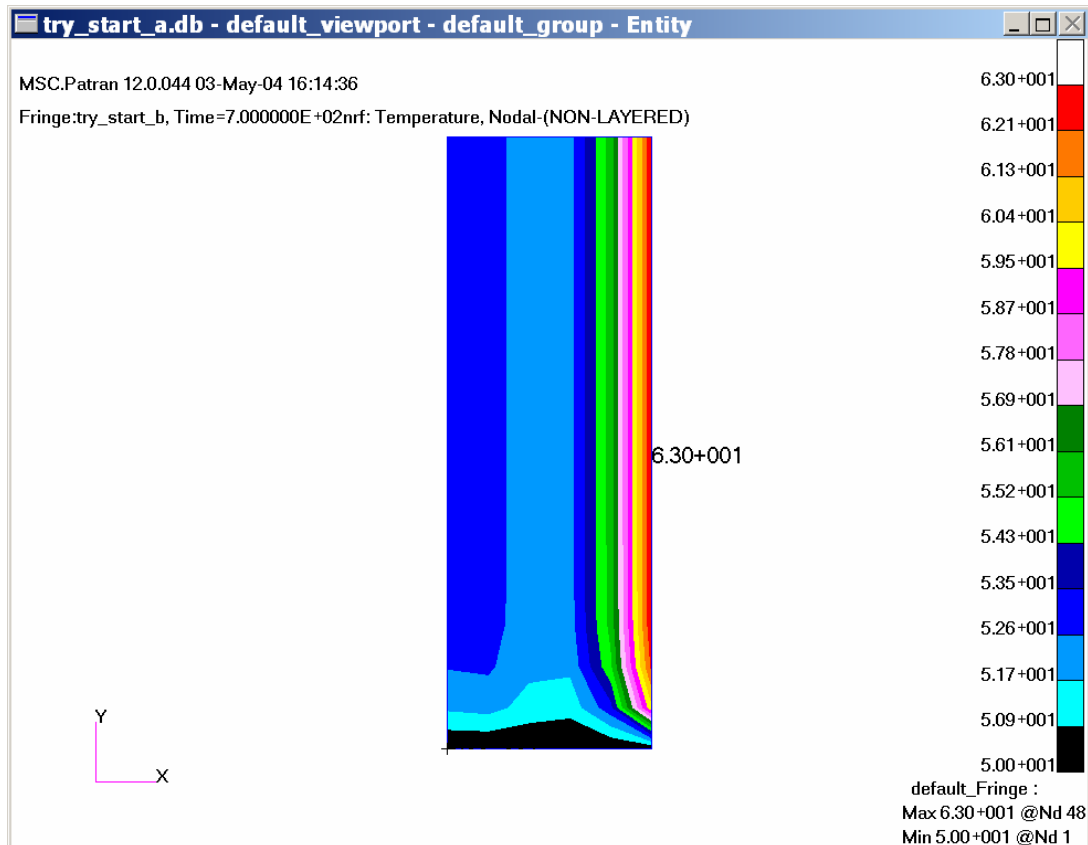
try_start_b, Time=7.000000E+02nrf

Select Fringe Result:

Temperature, nodal

Apply

A contour plot displaying temperature distributions will appear. Your model should look like the following figure.



Getting Started - SINDA/G for PATRAN

10. Now we will create an animation for the transient result.

◆ Results

Action:

Create

Object:

Fringe

In the **Select Result Case(s)** listbox, click and drag mouse to select all the transient cases from *try_start_b, Time=0.000000E+00nrf* to *try_start_b, Time=1.000000E+03nrf*.

Select Result Cases:

try_start_b, Time=7.000000E+02nrf
try_start_b, Time=7.000000E+02nrf
.....
try_start_b, Time=1.000000E+03nrf

Select Fringe Result:

Temperature, nodal

Apply

Then also inside the **Result** form, switch **Fringe** to **Animation**.

◆ Results

Action:

Create

Object:

Animation

Method:

2D Graphics

Select Fringe Result:

GV-FRI_default_fringe

Animation Method:

Global Variable

Select Global Variable:

LOAD CASE INDEX

Number of Frames:

20

Apply

Then the **Animation Control** form will show up. The first time, the animation looks very slow caused by the animation frames generation. It will be much faster after all the frames are generated. You can adjust the **Speed of Animation** and try different **Animation Sequence**. Stop

Getting Started - SINDA/G for PATRAN

the animation in the following way.

Stop Animation and...:

No Graphics Refresh

11. Close the database and quit MSC.Patran when you have completed this exercise. Patran will automatically save your model.

File/Quit...