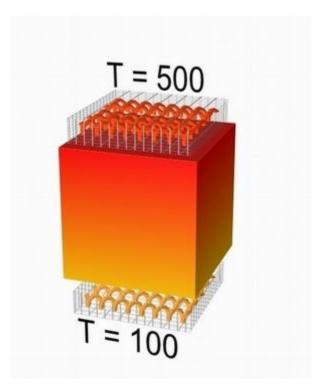
Transient Thermal Conduction Example

Introduction

This tutorial was created using ANSYS 7.0 to solve a simple transient conduction problem. Special thanks to Jesse Arnold for the analytical solution shown at the end of the tutorial.

The example is constrained as shown in the following figure. Thermal conductivity (k) of the material is 5 W/m*K and the block is assumed to be infinitely long. Also, the density of the material is 920 kg/m^3 and the specific heat capacity (c) is 2.040 kJ/kg*K.

It is beneficial if the <u>Thermal-Conduction</u> tutorial is completed first to compare with this solution.



Preprocessing: Defining the Problem

1. Give example a Title

Utility Menu > File > Change Title... /Title,Transient Thermal Conduction

2. Open preprocessor menu

ANSYS Main Menu > Preprocessor / PREP7

3. Create geometry

 $\label{eq:create} \begin{array}{l} Preprocessor > Modeling > Create > Areas > Rectangle > By 2 \ Corners \\ X=0, Y=0, Width=1, Height=1 \\ {\tt BLC4,0,0,1,1} \end{array}$

4. Define the Type of Element

Preprocessor > Element Type > Add/Edit/Delete... > click 'Add' > Select Thermal Mass Solid, Quad 4Node 55 ET, 1, PLANE55

For this example, we will use PLANE55 (Thermal Solid, Quad 4node 55). This element has 4 nodes and a single DOF (temperature) at each node. PLANE55 can only be used for 2 dimensional steady-state or transient thermal analysis.

5. Element Material Properties

 $\label{eq:source} \begin{aligned} & \text{Preprocessor} > \text{Material Props} > \text{Material Models} > \text{Thermal} > \text{Conductivity} > \\ & \text{Isotropic} > \text{KXX} = 5 \text{ (Thermal conductivity)} \\ & \text{MP, KXX, 1, 10} \\ & \text{Preprocessor} > \text{Material Props} > \text{Material Models} > \text{Thermal} > \text{Specific Heat} > \\ & \text{C} = 2.04 \\ & \text{MP, C, 1, 2.04} \\ & \text{Preprocessor} > \text{Material Props} > \text{Material Models} > \text{Thermal} > \text{Density} > \\ & \text{DENS} = 920 \\ & \text{MP, DENS, 1, 920} \end{aligned}$

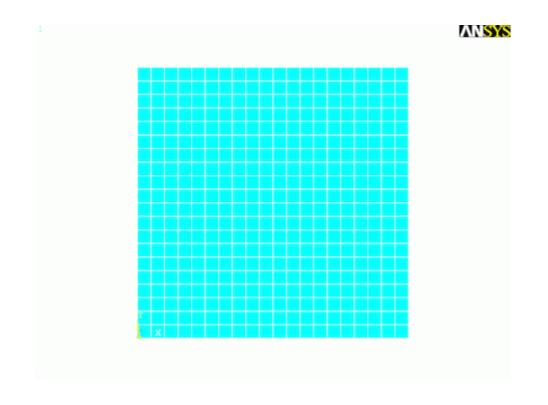
6. Mesh Size

Preprocessor > Meshing > Size Cntrls > ManualSize > Areas > All Areas > 0.05 AESIZE, ALL, 0.05

7. Mesh

 $\label{eq:preprocessor} Preprocessor > Meshing > Mesh > Areas > Free > Pick \mbox{ All AMESH, All}$

At this point, the model should look like the following:



Solution Phase: Assigning Loads and Solving

1. Define Analysis Type

Solution > Analysis Type > New Analysis > Transient $_{\tt ANTYPE}$, $_4$

The window shown below will pop up. We will use the defaults, so click OK.

▲ Transient Analysis			×
[TRNOPT] Solution method			
		• Ful	
		C Reduced	
		C Mode Superpos'n	
[LUMPM] Use lumped mass approx?		No	
ОК	Cancel	Help	

2. Set Solution Controls

Solution > Analysis Type > Sol'n Controls

The following window will pop up.

Solution Controls	×
Basic Transient Sol'n Options Nonlinear Ad	dvanced NL
Analysis Options Small Displacement Translent Calculate prestress effects Time Control Time at end of loadstep 300 Automatic time stepping On Automatic time stepping On Number of substeps Time increment Number of substeps 20 Max no. of substeps 20 Min no. of substeps 20 Min no. of substeps 20	Write Items to Results File Al solution Items Basic quantities User selected Nodel DOF Solution Nodel Reaction Loads Element Solution Element Nodel Loads Element Nodel Stresses Frequency: Write every substep Where N = 1
	OK Cancel Help

A) Set Time at end of loadstep to $300 \ and$ Automatic time stepping to ON.

B) Set Number of substeps to 20, Max no. of substeps to 100, Min no. of substeps to 20.

C) Set the Frequency to Write every substep.

Click on the NonLinear tab at the top and fill it in as shown

Solution Controls		×
Basic Transient Sol'n Options Nonline	ar Advanced NL	
Nonlinear Options Line search On DOF solution Prog Chosen predictor Equilibrium Iterations Maximum number 100 encode to the field of th	Cutback Control Limits on physical values to perform bisection: Equiv. Plastic strain Explicit Creep ratio Implicit Creep ratio Incremental displacement Points per cycle Cutback according to pr of iterations C Always iterate to 25 equi	
Set convergence criteria	ОК Са	ncel Help

For a complete description of what these options do, refer to the help file. Basically, the time at the end of the load step is how long the transient analysis will run and the number of substeps defines how the load is broken up. By writing the data at every step, you can create animations over time and the other options help the problem converge quickly.

3. Apply Constraints

For thermal problems, constraints can be in the form of Temperature, Heat Flow, Convection, Heat Flux, Heat Generation, or Radiation. In this example, 2 sides of the block have fixed temperatures and the other two are insulated.

- Solution > Define Loads > Apply Note that all of the -Structural- options cannot be selected. This is due to the type of element (PLANE55) selected.
- Thermal > Temperature > On Nodes
- Click the **Box** option (shown below) and draw a box around the nodes on the top line and then click OK.

Apply TEMP o	n Nodes
Pick	C Unpick
C Single	Box
C Polygon	C Circle
C Loop	
Count =	0
Maximum =	441
Minimum =	1
Node No. =	
(List of Min, Ma	
OK	Apply
Reset	Cancel
Pick All	Help

The following window will appear:

Apply TEMP on Nodes	×
[D] Apply TEMP on Nodes	
Lab2 DOFs to be constrained	A DOF TEMP
Apply as If Constant value then:	Constant value
VALUE Load TEMP value	500
OK Apply Can	Help

- Fill the window in as shown to constrain the top to a constant temperature of 500 K
- \circ $\,$ Using the same method, constrain the bottom line to a constant value of 100 K $\,$

Orange triangles in the graphics window indicate the temperature contraints.

4. Apply Initial Conditions

Solution > Define Loads > Apply > Initial Condit'n > Define > Pick All

Fill in the IC window as follows to set the initial temperature of the material to 100 K:

▲ Define Initial Conditions		×
[IC] Define Initial Conditions on Nodes		
Lab DOF to be specified		TEMP
VALUE Initial value of DOF		100
OK Apply	Cancel	Help

5. Solve the System

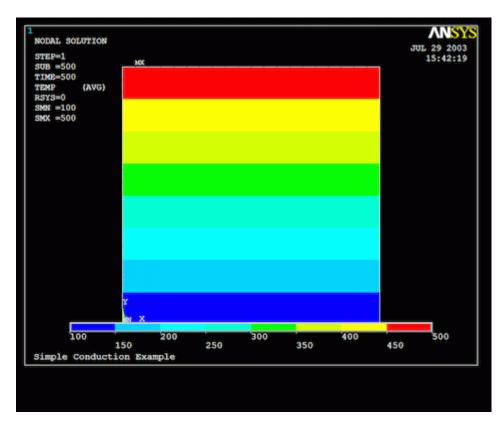
 $\begin{array}{l} Solution > Solve > Current \ LS \\ \texttt{SOLVE} \end{array}$

Postprocessing: Viewing the Results

1. Results Using ANSYS

Plot Temperature





Animate Results Over Time

• First, specify the contour range.

Utility Menu > PlotCtrls > Style > Contours > Uniform Contours...

Fill in the window as shown, with 8 contours, user specified, from 100 to 500.

Nuniform Contours	×
[/CONT] Uniform Contours	
WN Window number	Window 1
NCONT Number of contours	8
Contour intervals	
	C Auto calculated
	C Freeze previous
	 User specified
User specified intervals	
VMIN Min contour value	100
VMAX Max contour value	500
VINC Contour value incr	
[/REPLOT] Replot Upon OK/Apply?	Replot
OK Apply	Cancel Help

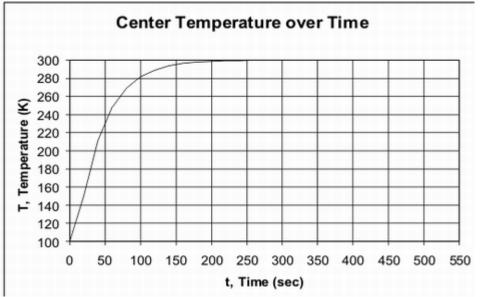
 \circ $\,$ Then animate the data.

Utility Menu > PlotCtrls > Animate > Over Time...

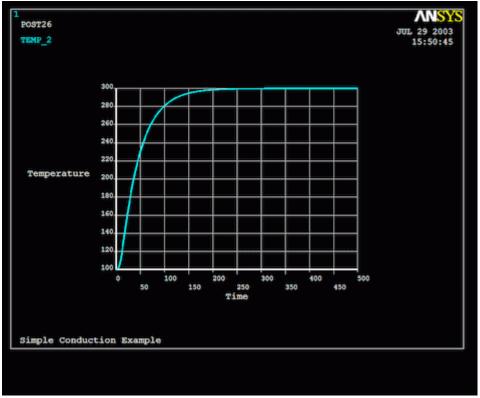
Fill in the following window as shown (20 frames, 0 - 300 Time Range, Auto contour scaling OFF, DOF solution > TEMP)

[ANTIME] Animate over time (interpolation of results)	
Number of animination frames	20
Model result data	
	C Current Load Stp
	C Load Step Range
	Time Range
Range Minimum, Maximum	0
Auto contour scaling	[□ off
Animation time delay (sec)	0.5
[PLDI,PLNS,PLVE,PLES,PLVFRC]	
Display Type	DOF solution Temperature TEMP Flux & gradient Layer Temp TBOT Layer Temp TE2 Layer Temp TE3 Layer Temp TE4
	Temperature TEMP

You can see how the temperature rises over the area over time. The heat flows from the higher temperature to the lower temperature constraints as expected. Also, you can see how it reaches equilibrium when the time reaches approximately 200 seconds. Shown below are analytical and ANSYS generated temperature vs time curves for the center of the block. As can be seen, the curves are practically identical, thus the validity of the ANSYS simulation has been proven.



Analytical Solution



ANSYS Generated Solution

Time History Postprocessing: Viewing the Results

- 1. Creating the Temperature vs. Time Graph
 - Select: Main Menu > TimeHist Postpro. The following window should open automatically.

		bles\fi	le.rth						2
le Help		 							
		B	None	- 2	4			Real	
/ariable L									۲
ame	Elemer	it No		Result Item			Minimum	Maximum	X-Axis A
ME				me			15	300	
									-
(<u> </u>
Calculator	r								۲
		-	=						
(1	1.1.1.1.1.1			1				
	1.00	See. 1			-		-		
MEN	CONJ	e^x					1		
MIN MAX	CONJ a+lb	e^x LN	7	8	9	1	CLEAR		
	[]		7	8	1	1			
MAX	[]		7	8	1	/			
MAX RCL STO	a+lb RESP	LN			9	/			
MAX RCL	a+lb RESP	LN LOG			9	/ *	CLEAR		
MAX RCL STO INS MEM	a+lb RESP	LN LOG SQRT	4	5	9				

If it does not open automatically, select **Main Menu > TimeHist Postpro > Variable Viewer**

- Click the add button in the upper left corner of the window to add a variable.
- Select Nodal Solution > DOF Solution > Temperature (as shown below) and click OK. Pick the center node on the mesh, node 261, and click OK in the 'Node for Data' window.

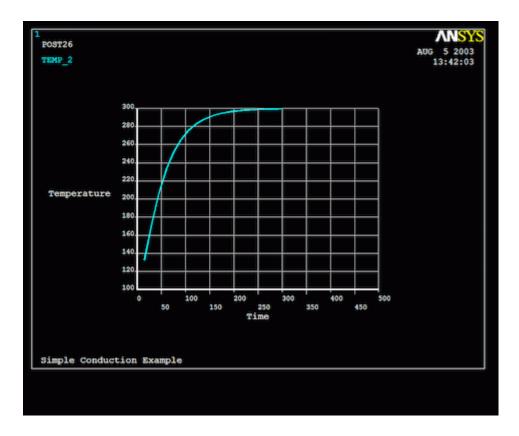
dd Time-History \	Variable		
Result Item			
Favorites Folder Nodal Solutio OF Sol OF Sol OF Sol OF Thermal Thermal Thermal Support Solution	ution mperature Gradient Flux		
Result Item Prop	erties		
Variable Name	TEMP_2		
OK	Apply	Cancel	Help

• The Time History Variables window should now look like this:

	් ග ්ස්	2 B N	lone		4			Real	
ariable L									
me	Elemer	nt Nod	le	Result Item			Minimum	Maximum	X-Axis
1E			1	Time			15	300	e
MP_2		261	viete internet	remperature	e a la l		132.784	299.667	•
									<u>D</u>
alculato	r								
		=							
			11111						
,		l r					-		
. (1				-		<u> </u>		
MEN	CONJ	e^x							
MAX	a+b	LN	7	8	9	1	CLEAR		
RCL	1			and the second sec	and the second second	and the second second second			
RCL STO	RESP	LOG	4	5	6		+		
STO			4	5	6		+		
STO NS MEN	1	SQRT					•		
STO			4	2	6	*	E		
STO NS MEN	1	SQRT					E N T		
STO NS MEN	ATAN	SQRT x^2				•	E N T E R		

2. Graph Results over Time

- Ensure TEMP_2 in the Time History Variables window is highlighted. 0
- 0
- Click the graphing button in the Time History Variables window. The labels on the plot are not updated by ANSYS, so you must change them 0 manually. Select Utility Menu > Plot Ctrls > Style > Graphs > Modify Axes and re-label the X and Y-axis appropriately.



Note how this plot does not exactly match the plot shown above. This is because the solution has not completely converged. To cause the solution to converge, one of two things can be done: decrease the mesh size or increase the number of substeps used in the transient analysis. From experience, reducing the mesh size will do little in this case, as the mesh is adequate to capture the response. Instead, increasing the number of substeps from say 20 to 300, will cause the solution to converge. This will greatly increase the computational time required though, which is why only 20 substeps are used in this tutorial. Twenty substeps gives an adequate and quick approximation of the solution.