



Preprocessor > Material Props > Material Models > Thermal > Density > DENS = 920  
MP, DENS, 1, 920

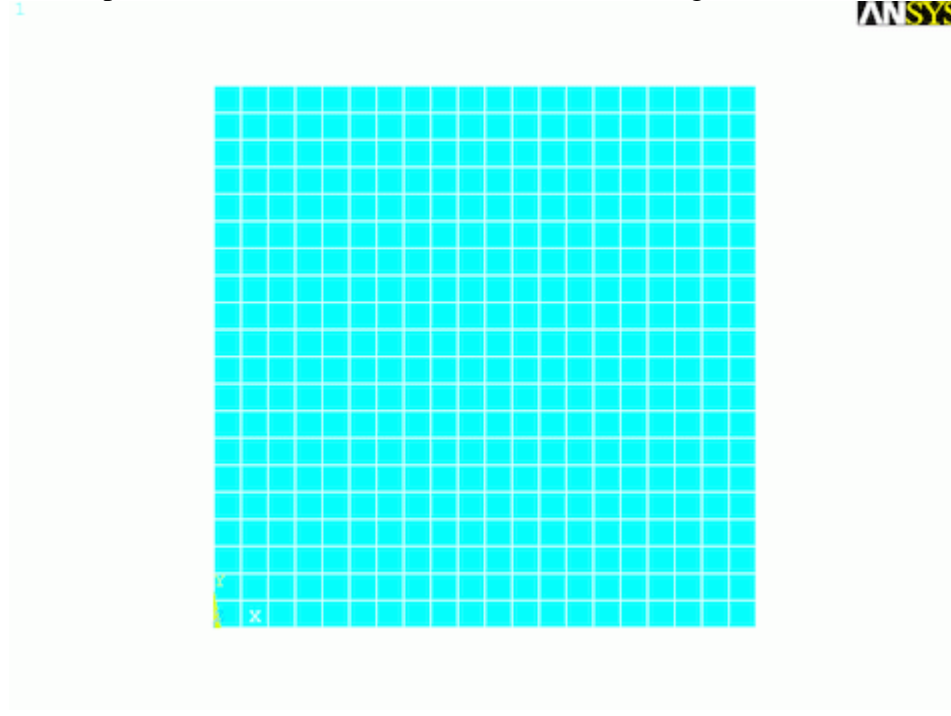
6. **Mesh Size**

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

7. **Mesh**

Preprocessor > Meshing > Mesh > Areas > Free > Pick All  
AMESH, ALL

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

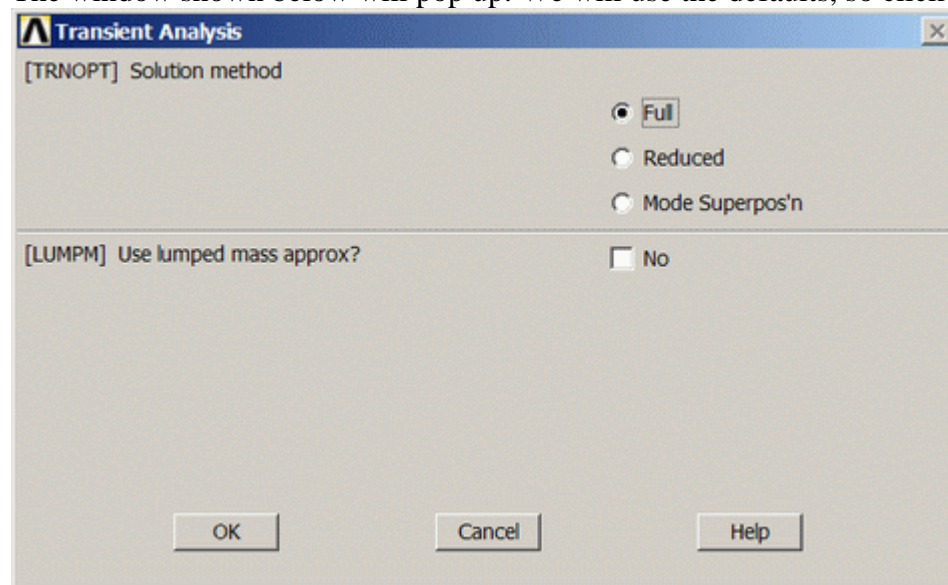


## Solution Phase: Assigning Loads and Solving

1. **Define Analysis Type**

Solution > Analysis Type > New Analysis > Transient  
ANTYPE, 4

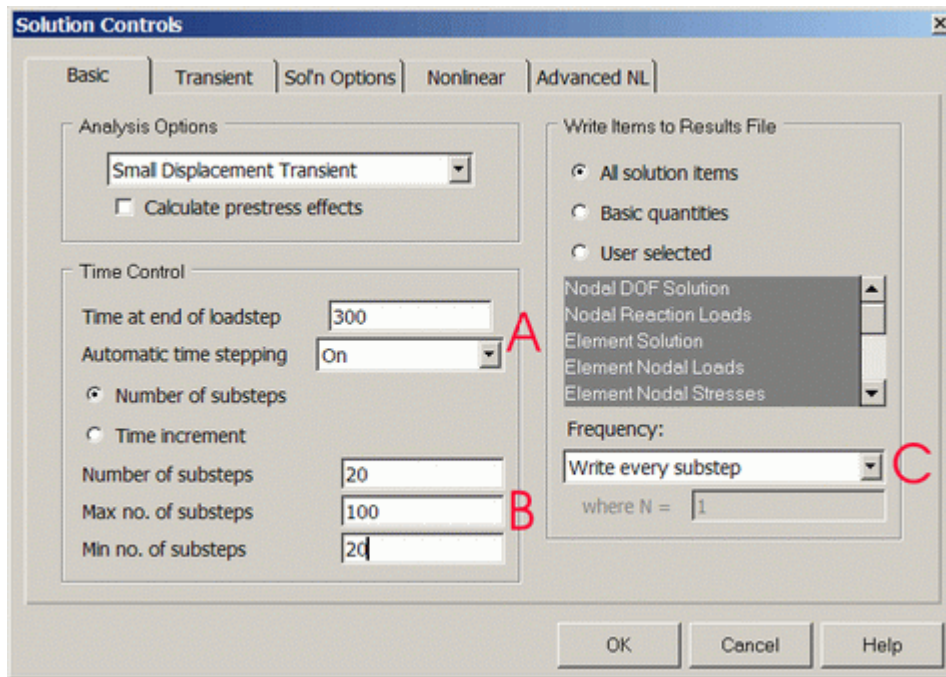
The window shown below will pop up. We will use the defaults, so click OK.



2. **Set Solution Controls**

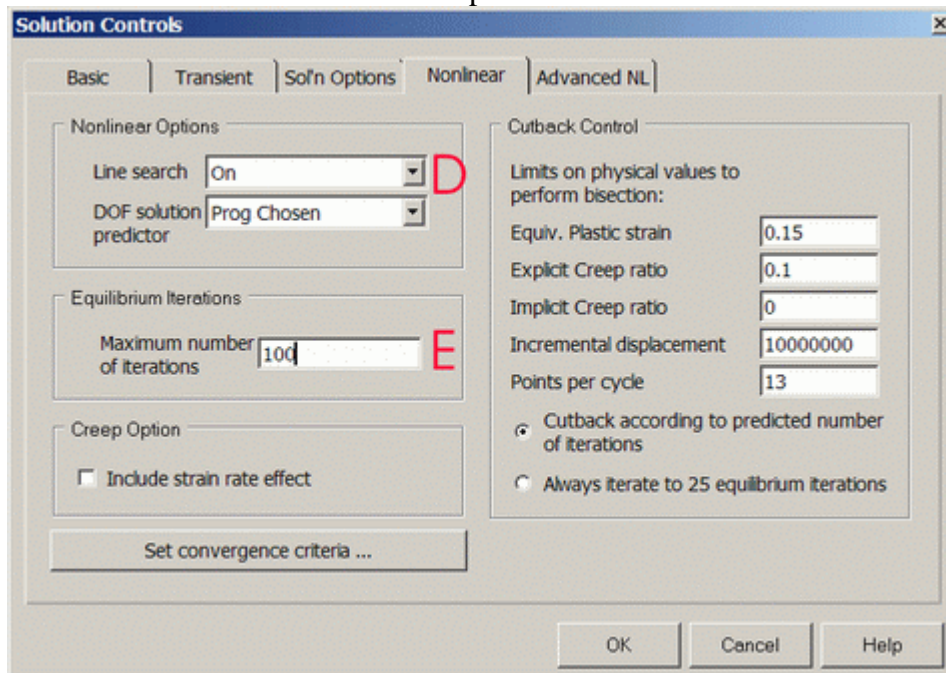
Solution > Analysis Type > Sol'n Controls

The following window will pop up.



- 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



- D) Set Line search to ON.  
 E) Set the Maximum number of iterations to 100.

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

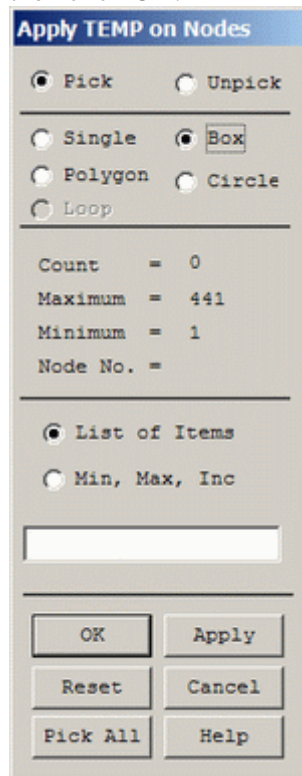
For thermal problems, constraints can be in the form of Temperature, Heat Flow, Convection, Heat Flux, Heat Generation, or Radiation. In this example, 2 sides of the block have fixed temperatures and the other two are insulated.

- **Solution > Define Loads > Apply**

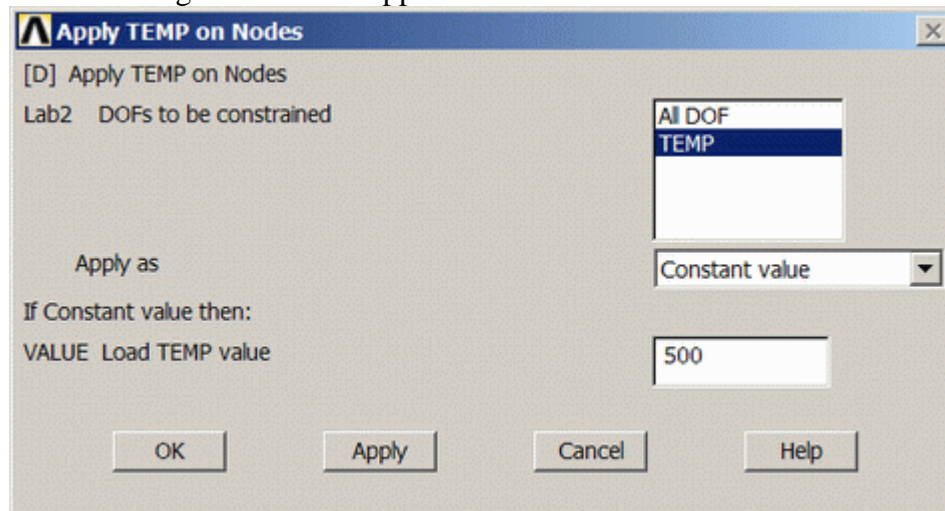
Note that all of the -Structural- options cannot be selected. This is due to the type of element (PLANE55) selected.

- **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.



The following window will appear:



- Fill the window in as shown to constrain the top to a constant temperature of 500 K

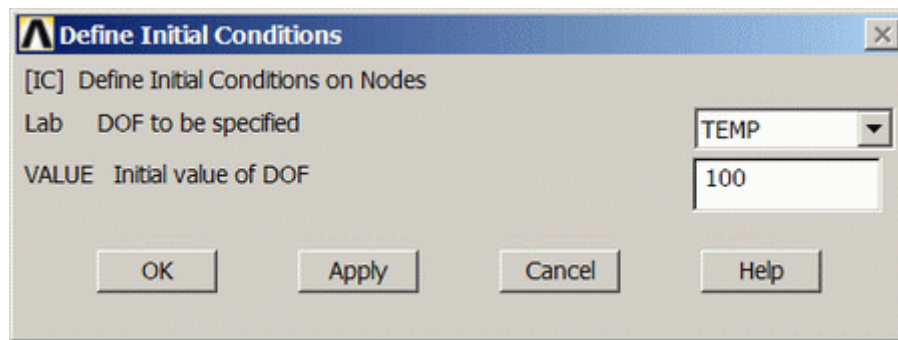
- Using the same method, constrain the bottom line to a constant value of 100 K

Orange triangles in the graphics window indicate the temperature constraints.

#### 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:

5. **Solve the System**

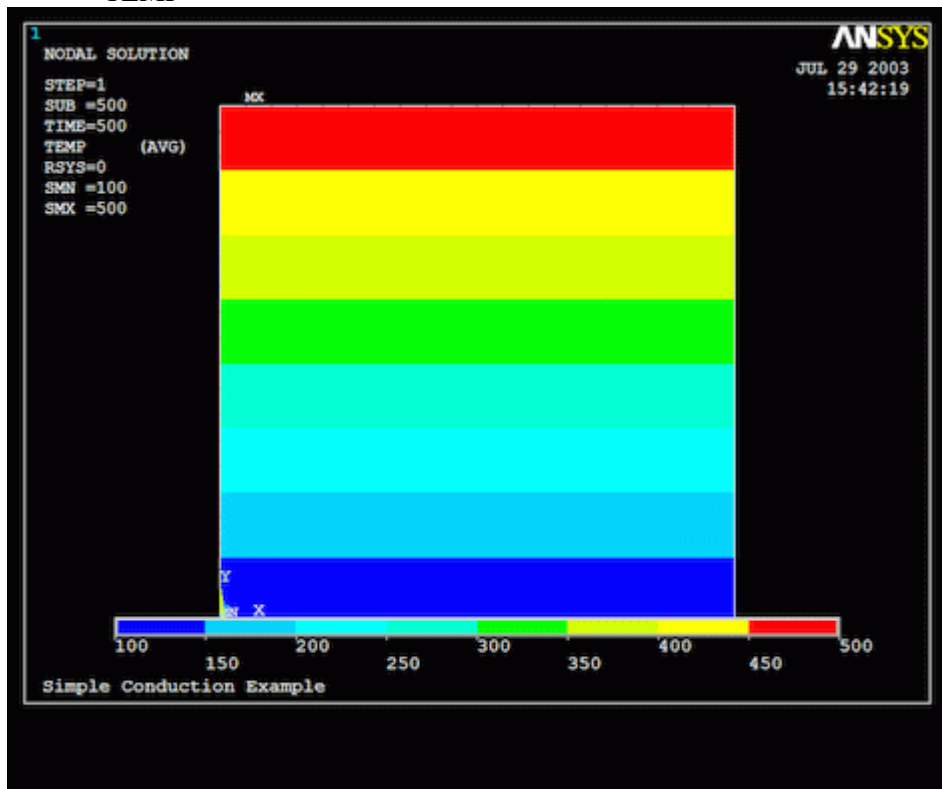
Solution > Solve > Current LS  
SOLVE

## Postprocessing: Viewing the Results

1. **Results Using ANSYS**

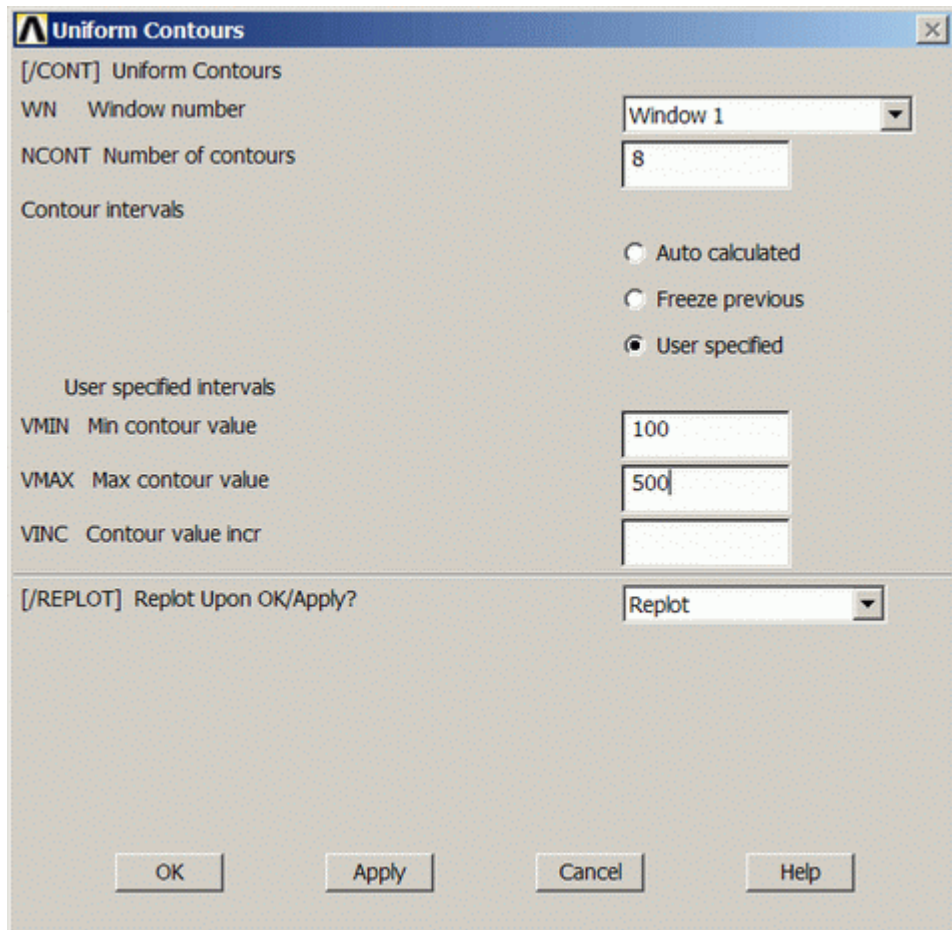
### Plot Temperature

General Postproc > Plot Results > Contour Plot > Nodal Solu ... > DOF solution, Temperature TEMP

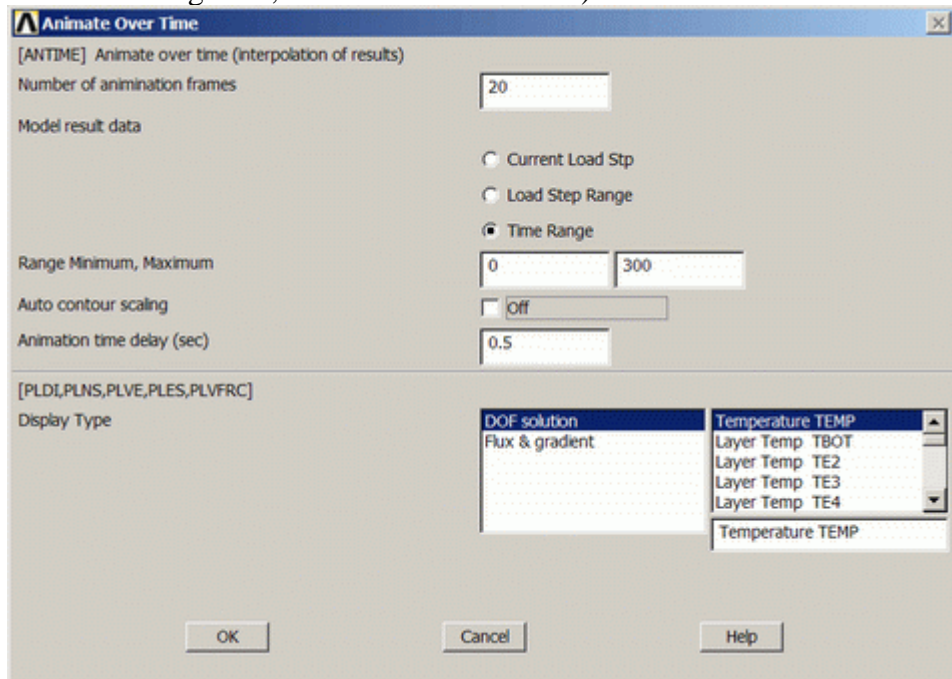


### Animate Results Over Time

- 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.

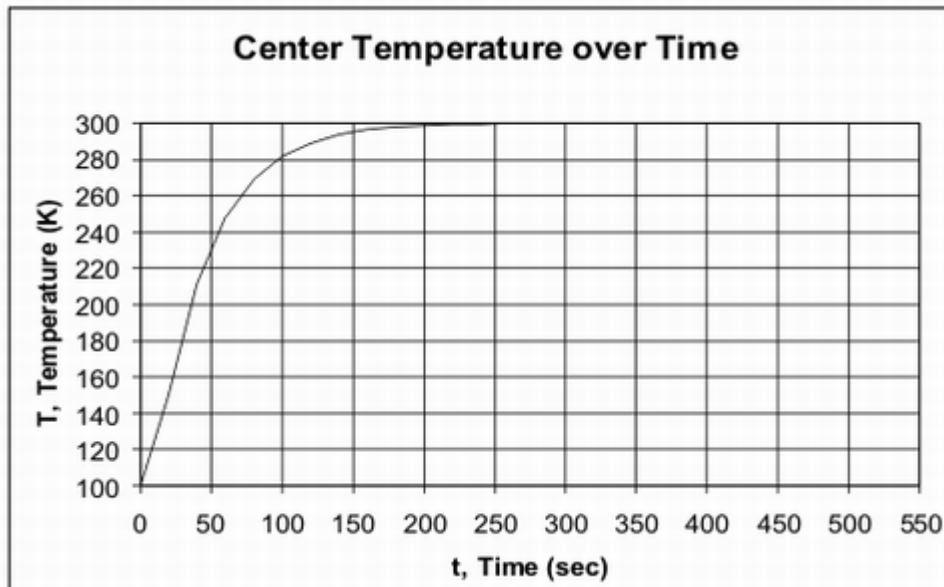


- Then animate the data.  
Utility Menu > PlotCtrls > Animate > Over Time...  
Fill in the following window as shown (20 frames, 0 - 300 Time Range, Auto contour scaling OFF, DOF solution > TEMP)

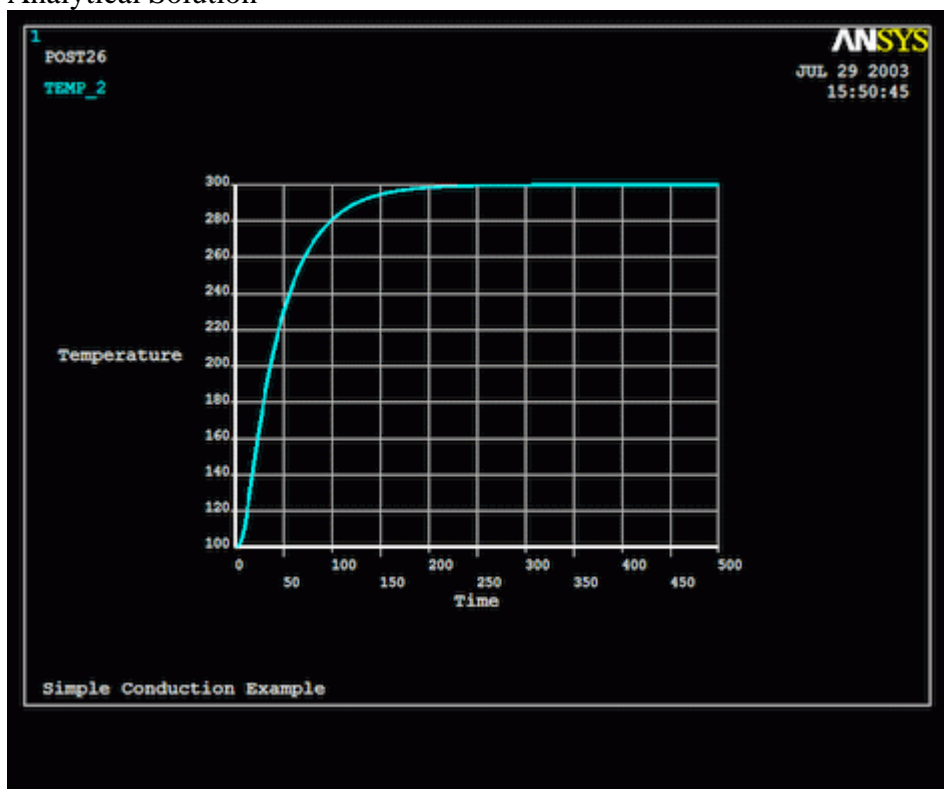


You can see how the temperature rises over the area over time. The heat flows from the higher temperature to the lower temperature constraints as expected. Also, you can see how it reaches equilibrium when the time reaches approximately 200 seconds. Shown below are analytical and

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.



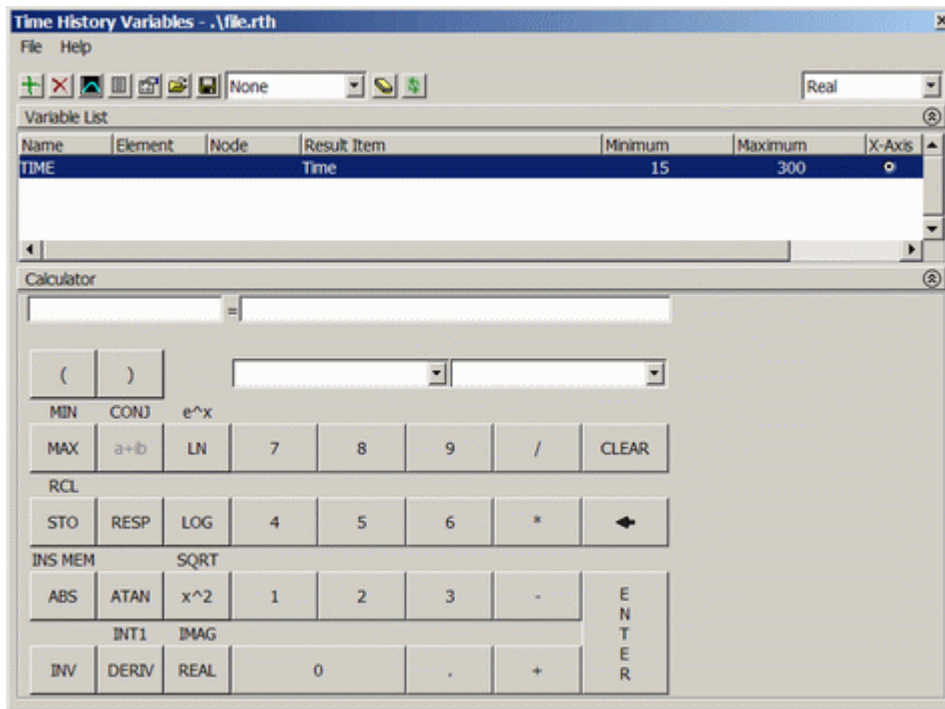
Analytical Solution




ANSYS Generated Solution

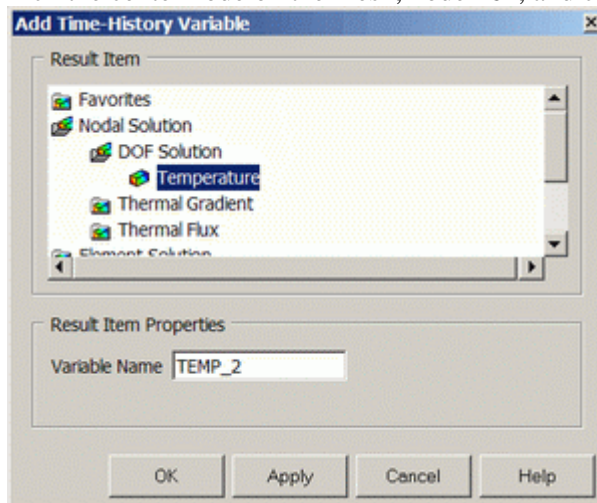
### Time History Postprocessing: Viewing the Results

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



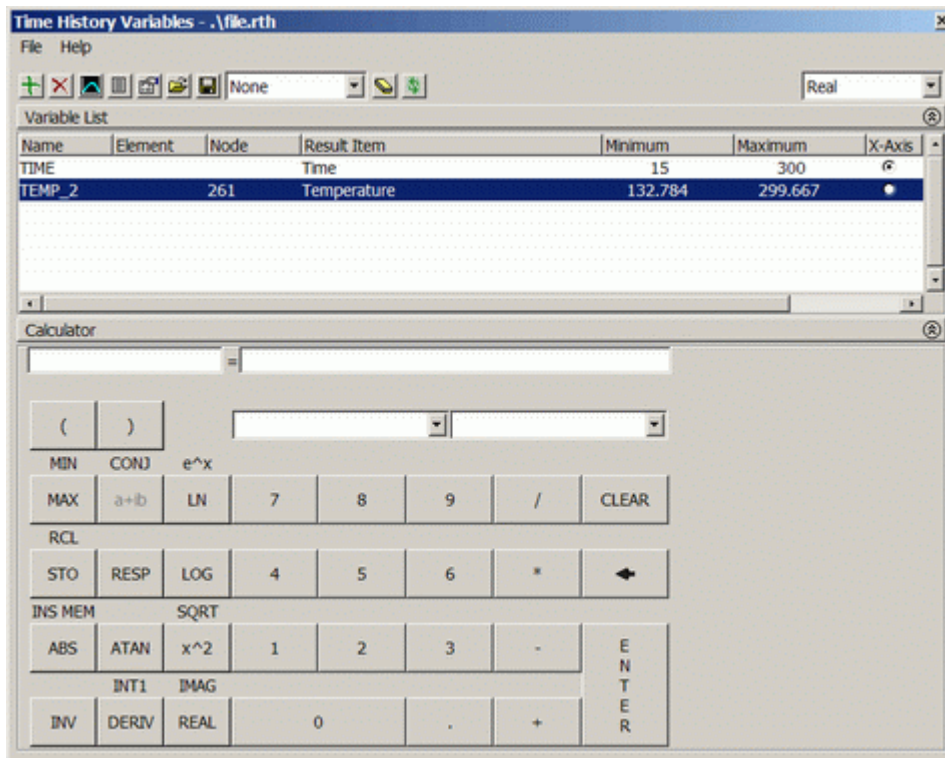
If it does not open automatically, select **Main Menu > TimeHist Postpro > Variable Viewer**

- 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.




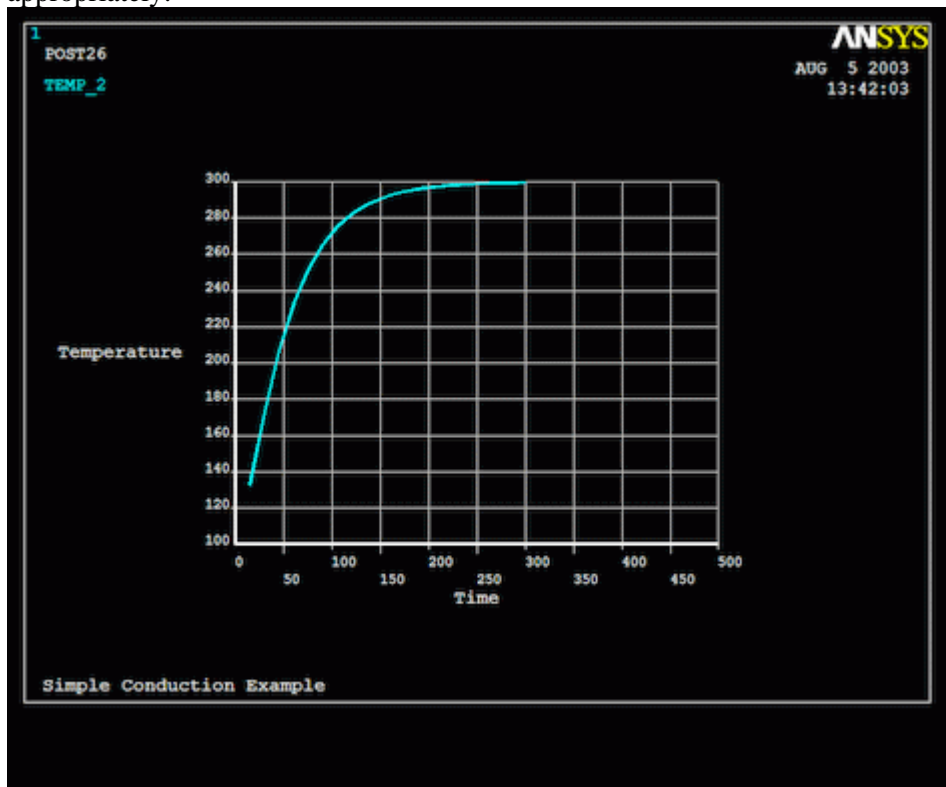
- The Time History Variables window should now look like this:





## 2. Graph Results over Time

- Ensure TEMP\_2 in the Time History Variables window is highlighted.
  - 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 manually.
- Select **Utility Menu > PlotCtrls > 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.

## Command File Mode of Solution

The above example was solved using a mixture of the Graphical User Interface (or GUI) and the command language interface of ANSYS. This problem has also been solved using the ANSYS command language interface that you may want to browse. Copy and paste following code into Notepad or a similar text editor and save it to your computer. Now go to '**File > Read input from...**' and select the file.

```
finish
/clear

/title, Simple Conduction Example
/PREP7                ! Enter preprocessor

! define geometry

length=1.0
height=1.0
blc4,0,0,length, height          ! area - one corner, then width and height

! mesh 2D areas

ET,1, PLANE55                ! Thermal element only
MP,Dens,1,920                ! Density
mp,c,1,2.040                 ! Specific heat capacity
mp,kxx,1,5                   ! Thermal conductivity
ESIZE,0.05                  ! Element size
AMESH,ALL                    ! Mesh area

FINISH
/SOLU

ANTYPE,4                     ! Transient analysis

time,300                      ! Time at end = 300

nropt,full                   ! Newton Raphson = full
lumpm,0                       ! Lumped mass approx off
nsubst,20                     ! 20 substeps
neqit,100                     ! Max no. of iterations = 100
autots,off                    ! Auto time search on
lnsrch,on                     ! Line search on
outres,all,all                ! Output data for all substeps
kbc,1

! fixed temp BC's
NSEL,S,LOC,Y,height          ! select nodes on top with y=height
D,ALL,TEMP,500               ! apply fixed temp of 500K
NSEL,ALL
NSEL,s,LOC,Y,0
D,ALL,TEMP,100               ! apply fixed temp of 100K
NSEL,ALL
IC,all,Temp,100              ! Initial Conditions: 100K

SOLVE
```

FINISH

```
/POST1          ! Enter postprocessor
/CONT,1,8,100,,500 ! Define a contour range
PLNSOL,TEMP     ! Plot temperature contour
ANTIME,20,0.5,,0,2,0,500 ! Animate temp over time
```

### Reference

<http://www.mece.ualberta.ca/tutorials/ansys>