# HEAT TRANSFER PROBLEMS AND THERMAL STRESSES

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x} \left( \lambda_x \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \lambda_y \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( \lambda_z \frac{\partial T}{\partial z} \right) + q_v(x, y, z, t) ,$$

where: T(x, y, z, t) - temperature,  $q_v$  - heat generation (W/m<sup>3</sup>),  $\rho$  - density (kg/m<sup>3</sup>),  $\lambda_x, \lambda_y, \lambda_z$  - thermal conductivity (W/mK), c - specific heat (J/kg).

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most materials do vary with temperature, so the analysis usually is nonlinear.

**Transient thermal analysis determines** temperatures and other thermal quantities that vary over time. Many heat transfer applications--heat treatment problems, nozzles, engine blocks, piping systems, pressure vessels, etc.--involve transient thermal analyses.

Temperature distribution that a transient thermal analysis calculates is often used as input to structural analyses for thermal stress evaluations.

A transient thermal analysis follows basically the same procedures as a steady-state thermal analysis. The main difference is that most applied loads in a transient analysis are functions of time. To specify time-dependent loads the load-versus-time curve should be divided into load steps.

If you use individual load steps, each "corner" on the load-time curve can be one load step, as shown in the following sketches.



Examples of load-versus-time curves - stepped and ramped loads

For each load step, you need to specify both load values and time values, along with other load step options such as stepped or ramped loads, automatic time stepping, etc. You then write each load step to a file and solve all load steps together.

### Temperature-dependent coefficient of thermal expansion $\alpha_t(T)$

If  $\alpha_t$  is the thermal expansion coefficient then the typical component of thermal strain is  $\epsilon_{th} = \alpha_t (T) (T-T_0)$ .

If  $T_0 = T_{ref}$ , where  $T_{ref}$  is the reference temperature at which zero strains exist such a coefficient is correctly used. If this condition is not true an adjustment must be made (MPAMOD command in the Preprocessor).

#### **Thermal stresses**

Thermal stresses usually are analysed using sequential method.

The *sequential method* involves two or more sequential analyses, each belonging to a different field. You couple the two fields by applying *results* from the first analysis as *loads* for the second analysis. In the case of thermal-stress analysis the nodal temperatures from the thermal analysis are applied as "body " loads in the subsequent stress analysis

Ansys enables also using *direct method* which involves just one analysis that uses a *coupled-field element type* containing all necessary degrees of freedom.



Data flow for a sequential coupled field analysis.

### Example 1 - thermal stresses in steady state heat flow

In the steel thick pipe we have the internal temperature  $T_w=100^{\circ}C$  end external temperature  $T_z=20^{\circ}C$ . The inner radius is a=30mm, and outer b=40mm. Show the temperature distribution, von Mises stress and stress components in the cylidrical coordinate system. E=2e11Pa, v=0.3,  $\alpha_t=1.2e-5$  1/K, k=50 W/mK. Consider the pipe constrained in the axial direction at both ends.

Analytical solution (plane strain problem,  $T_{ref}=0^{\circ}C$ ):



$$T(r) = T_w + \frac{T_z - T_w}{\ln\left(\frac{b}{a}\right)} \ln\left(\frac{r}{a}\right)$$

$$\sigma_r(r) = C \left[ \ln\left(\frac{b}{r}\right) / \ln\left(\frac{b}{a}\right) - \left(\frac{b^2}{r^2} - 1\right) / \left(\frac{b^2}{a^2} - 1\right) \right]$$

$$\sigma_t(r) = C \left[ \left( \ln\left(\frac{b}{r}\right) - 1\right) / \ln\left(\frac{b}{a}\right) + \left(\frac{b^2}{r^2} + 1\right) / \left(\frac{b^2}{a^2} - 1\right) \right]$$

$$\sigma_z(r) = v \left(\sigma_r + \sigma_t\right) - \lambda ET$$

$$C = \frac{-E\alpha_t \left(T_w - T_z\right)}{2(1 - v)}$$

From the above formulas we have e. g.  $\sigma_t$  (a)=-150.2 MPa,  $\sigma_t$  (b)=124.1 MPa.





TEMP

Temperure along the thickness of the pipe



Radial stress (Sx) ,hoop stress (Sy) , axial stress (Sz) and von Mises stress (Sred) along the thickness of the pipe (MPa)



Distribution of radial stress (Sx) ,hoop stress (Sy) , axial stress (Sz) and von Mises stress (Sred) Cylindrical coordinate system

### Approach

The analysis may be performed using plane strain model (cross-section of the pipe), axisymmetric model, or 3D model. In each case only a segment of the pipe may be analysed with the adequate symmetry conditions.

### Summary of steps in numerical analysis (3D):

#### Preprocessor

-define geometry of the analysed region (part of the cylinder),

-define the material properties  $(E,v,\lambda,k)$ 

-define the element type (thermal solid)

-specify the boundary conditions (temperatures  $T_w$  and  $T_z$  applied on the adequate areas)



-mesh the model

#### Solution

-solve the current load step (Solve>Current Load Step)

### General Postprocessor

-review the results- plot the maps of interest and the temperature as the graph along the path (thickness).

### Preprocessor

- change the element type from *thermal solid* to *structural solid* (Element Type>Switch Element Type)

- switch element technology to Enhanced Strain Preprocessor

B Elem	ent Type		
🖬 🗛	dd/Edit/De	elete	
🖬 S	T Element	Types	X
🖬 🗛			
🖾 R			
📼 E	Define	d Element Types:	
⊞ Rea	Туре	1 SOLID185	
⊞ Mat			
Sec		A SOLID185 element type options	
⊞ Moc		Options for SOLID185, Element Type Ref. No. 1	
. Mes			
⊞ Che		Element technology K2	Full Integration
. Nun ₪		Laver construction K3	Full Integration
Arcl			Reduced integration
Cou		Element formulation K6	Simple Enhanced Strn
FLO			Simple Enhanced Still
. Mult		User defined initial stress K10	No USTRES routn 💌
🗉 Loa			
Phy		OK Cancel	Help
Patł			

#### Solution

- apply the boundary conditions for stress analysis (*Define Loads>Displacements> Symmetry B.C.>On Areas*)

- apply the nodal tempertatures as a load in stress analysis (file *jobname.rth* from thermal analysis)



- solve the current load step

#### General Postprocessor

- review the results - plot the stresses as graphs along the path (thickness). Use cylindrical coordinate system: (*Options For Output>Results Coordinate System*)

### **Tasks and questions:**

1. Repeat the analysis using 2D plane strain or axisymmetric model. Compare the obtained results with the results corresponding to the 3D model.

2. Perform the adequate analysis for the pipe with unconstrained ends (without the axial compression). Explain the differences.

3. Find the results corresponding to point 2, but with the 10mm insulation (E=1\*10<sup>3</sup> MPa, v=0.35, k= 0.1 W/m<sup>2</sup>K)

3. Repeat the calculations using the model of convection b.c. : At the int. surface : bulk temp. 100C, film coefficient  $h=500W/(m^2K)$ At the ext. surface: bulk temperature 20C, film coefficient is  $h=10W/(m^2K)$ . Explain the results.

In each case save the results: FE mesh, temperature distribution and stress components distributions in the form of contour plots and graphs along the path (thickness).

### Example 2 - thermal stresses in transient heat flow

Steel balls of diameter d=12mm are heated to  $T_1$ =850°C and then quenched in oil. The temperature of the oil is assumed as constant  $T_0$ =40°C. Heat exchange coefficient at the surface oil – the ball is h=400W/(m<sup>2</sup>K).

How long should the balls stay within the oil bath to get the temperature  $T_2=100^{\circ}C$  in the centre?

What is the maximum von Mises stress during the process?

Material properties of the steel : density  $\rho$ =7800kg/m<sup>3</sup>, specific heat c=444J/(kgK), thermal conductivity k=40W/mK, coefficient of thermal expansion  $\alpha_t$ =1.2e-5 1/K, Young modulus E=2e11Pa, Poisson's ratio v=0.3.



b) the difference DT=Tw-Tz



Temperature distribution (°C) and von Mises stress (Pa) after 1 s of cooling



Temperature distribution (°C) and von Mises stress (Pa) after 48 seconds of cooling.

# Approach

The analysis may be performed using axisymmetric or 3D model. In each case only a segment of the ball may be analysed with the adequate symmetry conditions.

### Summary of steps in numerical analysis (3D):

Preprocessor

-define geometry of analysed region (part of the sphere by intersection of sphere and block),

-define material properties (E,v, $\lambda$ ,k,  $\rho$ ,c)

-define element type (*thermal solid*)

-specify the boundary conditions (convection on external surface with *bulk temperature*  $40^{\circ}$ C, *film coefficient* 400W/m<sup>2</sup>K)



-mesh the model

#### Solution

- set type of analysis - ANALYSIS TYPE - transient (solution)-



 set initial condition TREF= 850°C (being the initial temperature)
 set time, time step and related parametrs (Load step options), e.g.: time=50sec, time step 0.2 sec., stepped, auto time step OFF

INEVE Main Manuel (	all —		
	<b>쐭</b>		
Preferences			
Preprocessor			
E Solution			
East Sol'n Onth		Time and Time Step Options	
Define Loads		Time and Time Step Options	
🗆 Load Step Opts			
Output Ctrls		[TIME] Time at end of load step	50
Solution Ctrl			
Time/Frequenc		[DELTIM] Time step size	250
🧰 Time - Time Step			
Time and Substps		[KBC] Stepped or ramped b.c.	
Damping			C Ramped
			() Ramped
⊞ Nonlinear			<ul> <li>Stepped</li> </ul>
ExpansionPass     Other			
Reset Ontions		[AUTOTS] Automatic time stepping	
Read LS File			C 01
🔤 Write LS File			O ON
Initial Stress			OFF
Physics			9 011
SE Management			Prog Chosen
Results Tracking		[D]] TIMI Mining up time step size	
Solve		[DEL IIM] Minimum ume step size	
E FLUTRAN SET UP		Maximum time atop aiza	
E CI Cot Up		Maximum ume step size	
E MultiField Set Un		Line province atom size?	
ADAMS Connection		Use previous step size?	Yes
Diagnostics			
Abridged Menu		[TSRES] Time step reset based on specific time points	

output controls (file write sequency to every substep or every Nth step)

ANSYS Main Menu 🛞	Variation	
🗟 Preferences		
Solution		
Analysis Type		
Define Loads	M Solution Printout Controis	
E Load Step Upts	[OUTPR] Solution Printout Controls	
Uutput Utris		
Solu Printouu	Item Item for printout control	Basic quantities 📃 💌
B Chow Status	EBEO Print fraguency	
B DGP File		
Incl Mass Summry		🔿 None
Integration Pt		
Other		🔘 Last substep
Stop Solution		C. D. unit in the term
Reset Options		C Every substep
📾 Read LS File		Every Nth substr
🔤 Write LS File		
Initial Stress	Value of N	2
E SE Management		
Results Tracking	(Use negative N for equally spaced data)	
Solve		
⊞ FSI Set Up	Cname Component name -	
MultiField Set Up		
ADAMS Connection	<ul> <li>for which above setting is to be applied</li> </ul>	
Diagnostics		
I Conorol Postaros		
I deneral Postproc	OV I Instru	Canaal
Interist Postpro		Help
Topological opt		

-solve Current Load Step

# General Postprocessor

Read the results corresponding to the chosen time point

ANSYS Main Menu						
■ Preferences		Command				
🗄 Solution		Sommania				
🗆 General Postproc	General Postproc					
Data & File Opts						
Results Summary	Available Da	ata Sets:				
E Read Results	Set	Time	Load Step	Substen	Cumulative	
In Heat Set	1	0.20000	1	1	1	
Brouious Sot	1	0.20000	1	1	1	
I ast Set	2	0.40000	1	2	2	
m By Pick	3	0.60000	1	3	3	
🔜 By Load Step	4	0.80000	1	4	4	
By Time/Freq	5	1.0000	1	5	5	
🔤 By Set Number	6	1.2000	1	6	6	
ELOTRAN 2.1A	7	1.4000	1	7	7	
Plot Results	8	1.6000	1	8	8	
List Results	9	1.8000	1	9	9	
Uuery Results     Element     Continue for Outer     Continue     Contin     Contin     Continue     Contin     C	10	2.0000	1	10	10	
Options for Outp     Results Viewer	11	2 2000	- 1	11	11	
Write DGP File	12	2,2000	1	12	12	
Nodal Calcs	12	2,4000	1	12	12	
Element Table	15	2,6000	1	15	15	
Path Operations	14	2,8000	1	14	14	
	15	3.0000	1	15	15	
🗉 Load Case	16	3,2000	Ţ	16	16	
Check Elem Shape		Read			Next	
Write Results				_		
ROM Operations		2				
⊞ Submodeling		, Cl	ose			
⊞ raugue ⊞ Define/Modify						
Nonlinear Diagnostics	10 0	0000	1	10	10	
	19 3	.8000	1	19	19	

-review the results corresponding to the arbitrary point of time -animate the results (General Postprocessor – PlotCTrl)

# TimeHist Postprocessor

-review the results e.g. variation of temperature with respect to time at chosen points (commands *Define Variables i Graph Variables*):

SAVE_DB RESUM_DB	QUIT POWRGRPH ANSYSWEB TUTORIAL WHATSNEW			
	\Lambda Time History V	ariables - aug28a.rth		
ANSYS Main Menu	File Help			
<ul> <li>Preferences</li> <li>Preprocessor</li> </ul>		😭 🖆 📕 None 💽 💊 💲		
E Solution	Variable List			
H General Postproc	Name Elemen	t Node Result Item		
TimeHist Postpro     Wariable Viewe	] TimeHist Postpro TIME			
Settings		Result Item		
📰 Store Data		Result Item		
📰 Define Variabl		🛜 Favorites		
Read LSDYNA		need Nodal Solution		
List Variables				
List Extremes				
📰 Graph Variabl				
Math Operatic	4	🚰 Thermal Gradient		
Table Uperation     Table Uperation	Calculator	🛃 Thermal Flux		
Smooth Data		Element Solution		
Generate Spe     Deset Postpro	1			
FI ROM Tool		Result Item Properties		

-find the difference between the temperature in the centre of the ball and the surface – delta T:

🔨 Time I	listory ¥ari	iables - au	ıg28a.rth			
File Help	I.					
$+\times$		1 🛋 日	None 🗾 💊 🕸			
Variable	 List					
Name	Element	Node	Result Item	Minimum	Maximum	X-Axis
TIME			Time	0.05	50	•
Tzew		22	Temperature	87.6881	842.994	0
Twew		2	Temperature	89.1317	849.959	0
deltaT			Calculated	1.44368	22.8599	0
-						
· ·						
Calculate	νr.					
Calculato	or					
Calculato	or deltaT	= <mark>{T</mark> )	wew}-{Tzew}			

-show the defined functions in the form of graphs

#### Preprocessor

-change the element type from *thermal solid* to *structural solid* 



-apply the boundary conditions for stress analysis (symmetry)

-apply the nodal tempertatures as a load in stress analysis at the arbitrary point of time  $t_{arb}$  (file *jobname.rth*)

![](_page_12_Picture_6.jpeg)

#### Solution

- set analysis type to static

-set time=1, analysis using 1 substep (Time/Frequency) -solve Current Load Step

#### General Postprocessor

-Review the results for the time t<sub>arb</sub>

In the presented way of analysis the thermal stresses corresponding to the chosen time  $t_{arb}$  were presented. If we want to find the history of stresses in a period of time the corresponding history of loads for structural analysis should be built in the form of Load Step files (Files *Jobname.s i*, where i is the number of the load step).

It may be also performed automatically using for example the presented below macro, which is set of commands written using Ansys Parametric Design Lenguage (APDL). The file may be read by the program in Solution phase using the INPUT command.

```
/COM, Creation of files jobname.si for history of linear thermal stresses
/COM, The commands may be input after changing elements to structural and
/COM, after applying the adequate structural boundary conditions
     *ask,case,name of the file rth (jobname) =,'file'
     *ask,time_in,initial time =,1
     *ask,time e,final time =,1
     *ask,liczba_p,number of time segments = ,1
     przyr=(time e-time in)/liczba p
     chwila=time in
     ANTYPE,0
     *DO,k,1,liczba p
     LDREAD, TEMP, , , chwila, , case, 'rth',
     TIME, chwila
     AUTOTS,0
     NSUBST,1, , ,1
     KBC,1
     LSWRITE, k,
     chwila=chwila+przyr
     *ENDDO
```

# Tasks and questions:

-Repeat the analysis using axisymmetric model. Compare the results.

-Find the influence of heat exchange coefficient and thermal conductivity on the results (maximum stresses and time of cooling)

-Perform the analysis using the elasto-plastic model of the material. Why in this case residual stresses may be expected?