

OPTISTRUCT FOR LINEAR ANALYSIS, V2019  
CHAPTER 6: THERMAL STRESS STEADY STATE ANALYSIS

# AGENDA

1. Introduction to Linear Analysis
  - Type of Analysis
  - Type of Elements and Materials
  - Type of Loads & Boundary Conditions
2. Linear Static Analysis
3. Inertia Relief Analysis
4. Modal Analysis
5. Linear Buckling Analysis
6. Thermal Stress Steady State Analysis
7. Advanced Topics
  - Debugging Guide
  - Parameters
  - Transitioning Elements
  - Introduction to Parallelization
  - Run Options
  - Output Management
8. Optimization in Linear Analysis
  - OptiStruct Optimization
  - DRCO Approach
  - Setting up Optimization
  - Optimization Responses for Linear Analysis



# THERMAL STRESS STEADY STATE ANALYSIS

Not only forces or moments, but also **temperature changes** causes bodies to expand or contract. The total strain of a body is the sum of mechanical strain and heat strain:

$$\varepsilon = \varepsilon_{\sigma} + \varepsilon_T = \frac{\sigma}{E} + \alpha_T \Delta T \quad \text{with thermal expansion coefficient } \alpha_T \text{ and temperature change } \Delta T$$

If a body can not expand unrestricted, there are constraints which lead to (**thermal**) **stress**:

$$\sigma = E(\varepsilon - \alpha_T \Delta T)$$

Example values for  $\alpha_T$  are  $1.2 \cdot 10^{-5} \frac{1}{K}$  for steel and  $2.4 \cdot 10^{-5} \frac{1}{K}$  for aluminum.

The thermal expansion coefficient  $A$  and a reference temperature  $T_{REF}$  for thermal loading can be defined on the material cards, e.g. MAT1:

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)
MAT1	MID	E	G	NU	RHO	A	TREF



# THERMAL ANALYSIS

In general, the temperature field is not known, but calculated with a thermal analysis and can be referred by a structure solution to perform a coupled thermal/structural analysis.

OptiStruct does the following thermal analysis types

- Linear Steady-State Heat Transfer Analysis
- Linear Transient Heat Transfer Analysis
- Nonlinear Transient Heat Transfer Analysis (Beta)
- Nonlinear Steady-State Heat Transfer Analysis
- One Step Transient Thermal Stress Analysis
- Contact-based Thermal Analysis

For more information regarding thermal analysis, please review the following tutorials

- OS-T: 1080 Coupled Linear Heat Transfer/Structure Analysis
- OS-T: 1085 Linear Steady-state Heat Convection Analysis
- OS-T: 1090 Linear Transient Heat Transfer Analysis of an Extended Surface Heat Transfer Fin
- OS-T: 1100 Thermal Stress Analysis of a Printed Circuit Board with Anisotropic Material Properties
- OS-T: 1385 Heat Transfer Analysis on Piston Rings with GAP Elements



# THERMAL STRESS STEADY STATE ANALYSIS

TEMP card (represented as a temperature load in HyperMesh) defines temperature at grid points or a SET of grid points for determination of thermal loading and stress recovery.

TEMPD card (represented as a load collector in HyperMesh) defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.

Temperature sets may be selected for use in a subcase by the TEMPERATURE Subcase Information Entry.

```
SUBCASE      2
  LABEL Temp
  SPC =      1
  TEMPERATURE (LOAD) =      3
```

(1)	(2)	(3)	(4)
TEMP	SID	GIGSETID	T

(1)	(2)	(3)
TEMPD	SID	T

```
TEMPD      3      20.0
[... ]
TEMP      3      3      40.0
[... ]
TEMP      3      2      30.0
[... ]
```



# EXERCISE 6A: THERMAL STRESS ANALYSIS OF A BEAM

## File Name and Location

...\STUDENT-EXERCISE\6a\_Simple\_Beam\beam.hm



## Objectives (1/2)

This exercise runs a thermal stress analysis on a simple supported beam modeled with shell elements, known from exercises 3b and 5b.

The objective is to calculate the deformed shape and maximum displacement due to the SPCs and the following temperature load

- Reference temperature of 20
- Lower/middle/upper row of nodes has a temperature of 20/30/40



1. Open the model in HyperMesh Desktop and review the model



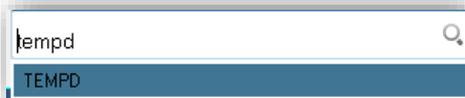
# EXERCISE 6A: THERMAL STRESS ANALYSIS OF A BEAM

## Objectives (2/2)

3. Add the thermal expansion coefficient and a reference temperature to the material card.
4. Create a load collector `TEMPERATURE` (card image `TEMPD`) with `T = 20` and create `TEMP` loads with a value of 30 & 40 for the middle upper row of nodes respectively.
5. Create a load step `Temp` with `SPC` referencing `SPC` load collector and `TEMP(LoadCOL)` referencing new `TEMPERATURE` load collector.
6. Run the analysis with OptiStruct, review the results in HyperView and check the maximum displacement

## Hints (1/3)

3. As the material is steel, take  $1.2e-5$  as `A`. `TREF` is given as 20.
4. Use HyperMesh's Quick Access Tool (`Ctrl+f`) to create a `TEMPD` card



A screenshot of the Material Properties table for 'steel'. The table has columns 'Name' and 'Value'. The rows are: Solver Keyword (MAT1), Name (steel), ID (1), Color (green), Include ([Master Model]), Defined (checked), Card Image (MAT1), User Comments (Do Not Export), E (210000.0), G, NU (0.3), RHO (7.85e-009), A (1.2e-005), TREF (20.0), and GF.

Name	Value
Solver Keyword	MAT1
Name	steel
ID	1
Color	<span style="color: green;">■</span>
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
Card Image	MAT1
User Comments	Do Not Export
E	210000.0
G	
NU	0.3
RHO	7.85e-009
A	1.2e-005
TREF	20.0
GF	

A screenshot of the Load Collectors table for 'TEMPERATURE'. The table has columns 'Name' and 'Value'. The rows are: Solver Keyword (TEMPD\*), Name (TEMPERATURE), ID (3), Color (green), Include ([Master Model]), Card Image (TEMPD), User Comments (Hide In Menu/Export), and T1 (20.0).

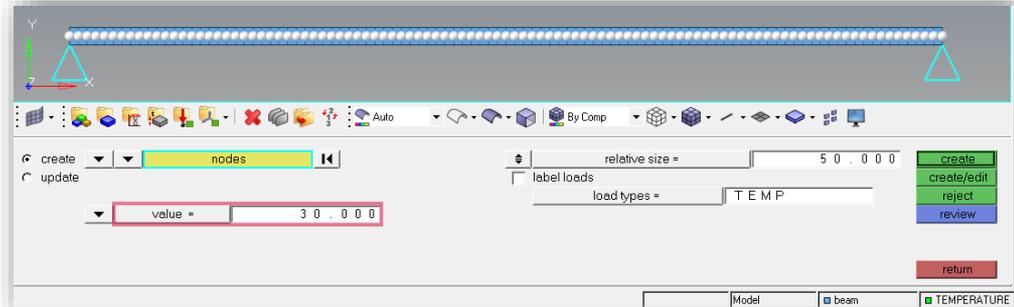
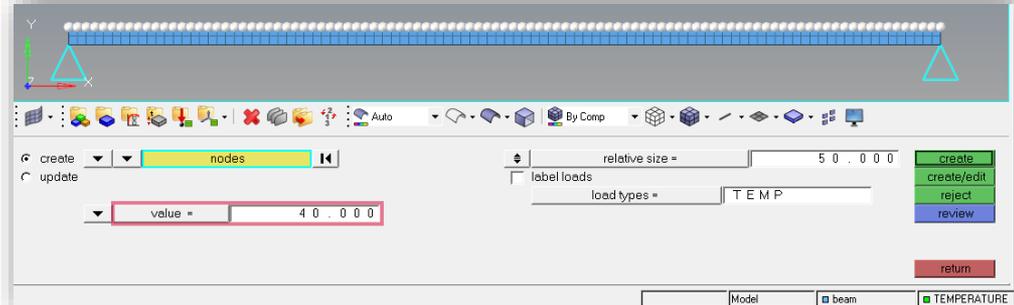
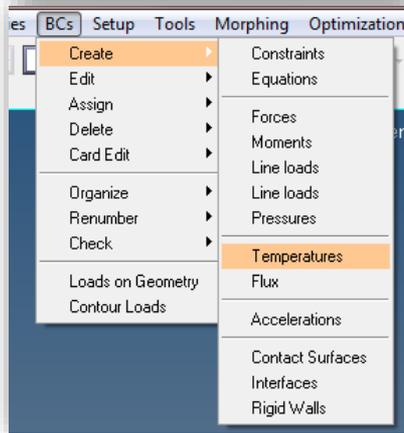
Name	Value
Solver Keyword	TEMPD*
Name	TEMPERATURE
ID	3
Color	<span style="color: green;">■</span>
Include	[Master Model]
Card Image	TEMPD
User Comments	Hide In Menu/Export
T1	20.0



# EXERCISE 6A: THERMAL STRESS ANALYSIS OF A BEAM

## Hints (2/3)

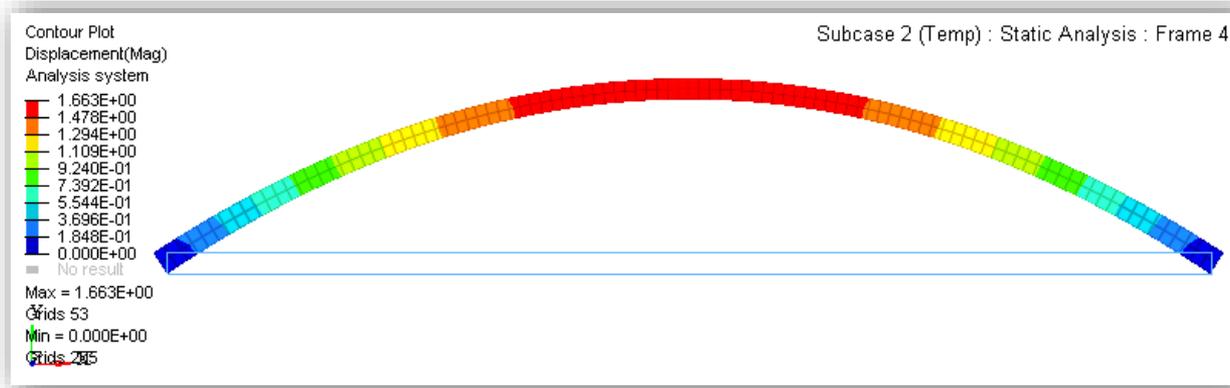
- Due to TEMPD card with  $T = 20$ , there is not need to create TEMP loads for the lower row of nodes.



# EXERCISE 6A: THERMAL STRESS ANALYSIS OF A BEAM

## Hints (2/3)

5. Use Generic as Analysis type.
6. Maximum displacement is 1.663 (deformed shape inflated by 100)



Name	Value
Solver Keyword	SUBCASE
Name	Temp
ID	2
Include	[Master Model]
User Comments	Hide In Menu/Export
<b>Subcase Definition</b>	
Analysis type	Generic
SPC	(1) SPC
LOAD	<Unspecified>
MPC	<Unspecified>
FREQ	<Unspecified>
TEMP (LOADCOL)	(3) TEMPERATURE
TEMP (SUBCASE)	<Unspecified>
METHOD (STRUCT)	<Unspecified>
SUPPORT	<Unspecified>



# QUESTIONS & ANSWERS

1. What is the maximum stress of a statically determined supported model?
  - a) zero
  - b) 42 MPa
  - c) infinite

