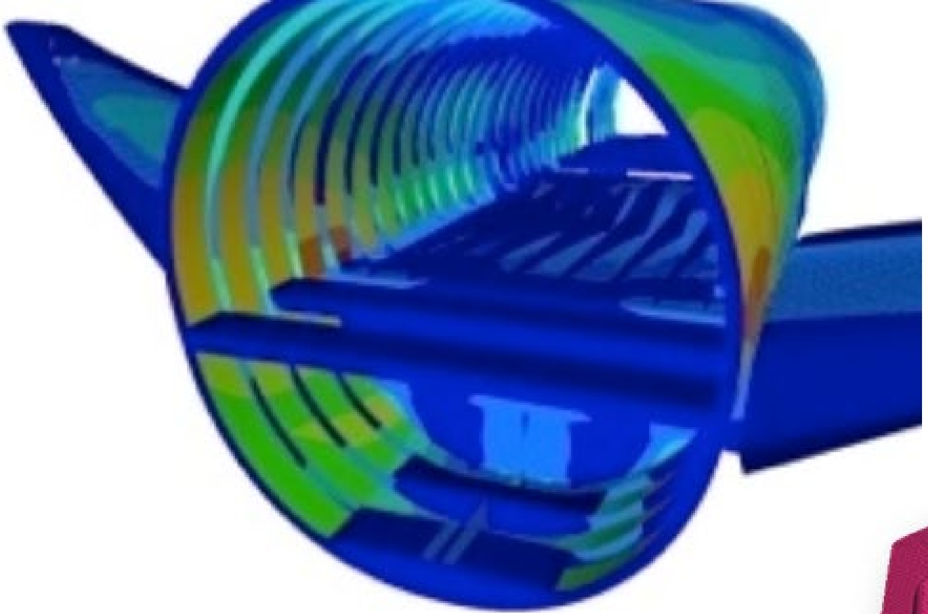
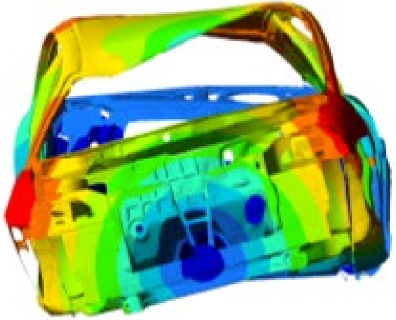
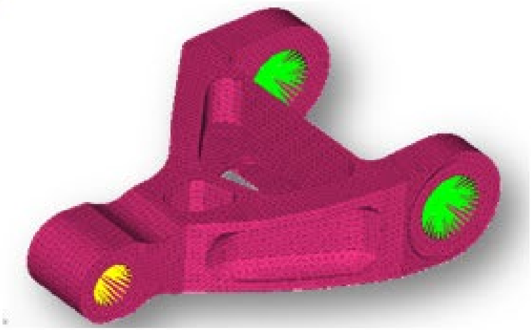






OPTISTRUCT FOR LINEAR ANALYSIS, V2019
CHAPTER 6: THERMAL STRESS STEADY STATE ANALYSIS

Altair

AGENDA

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:

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

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

The thermal expansion coefficient `A` and a reference temperature `TREF` 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.

(1)	(2)	(3)	(4)
TEMP	*SID*	*G/GSETID*	*T*

`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.

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

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
```

```
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

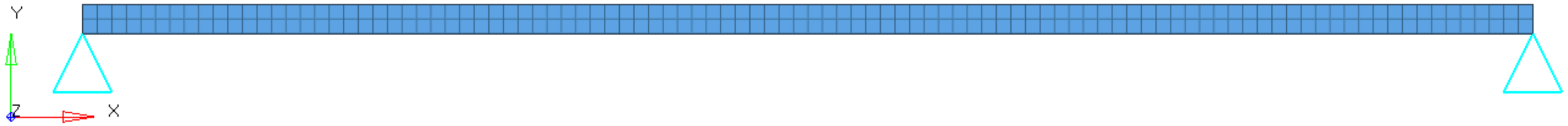
…\STUDENT-EXERCISE\6a_Simple_Beam\beam.hm

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

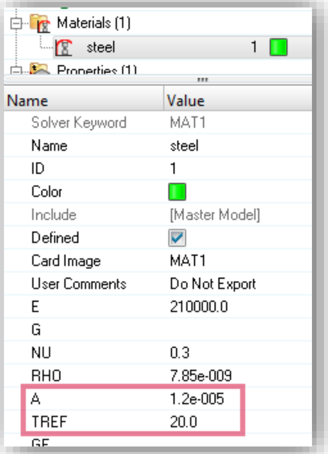


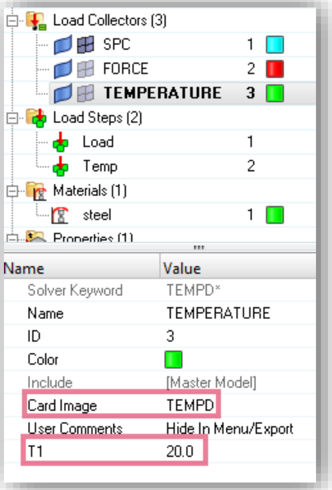
1. Open the model in HyperMesh Desktop and review the model

EXERCISE 6A: THERMAL STRESS ANALYSIS OF A BEAM

Objectives (2/2)

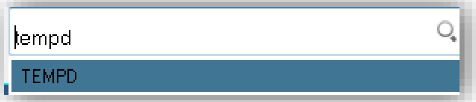
3. Add the thermal expansion coefficient and a reference temperature to the material card.
4. Create a load collector `TEMPERATURE` (card image `TEMPD`) with `T` = `20` and create `TEMP` loads with a value of `30` &. `40` for the middle upper row of nodes respectively.
5. Create a load step `Temp` with 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)

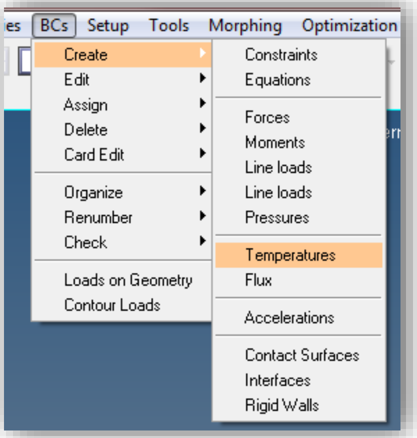
3. As the material is steel, take `1.2e-5` as `A`. `TREF` is given as `20`.
4. Use HyperMesh's Quick Access Tool (`Crtl+f`) to create a `TEMPD` card

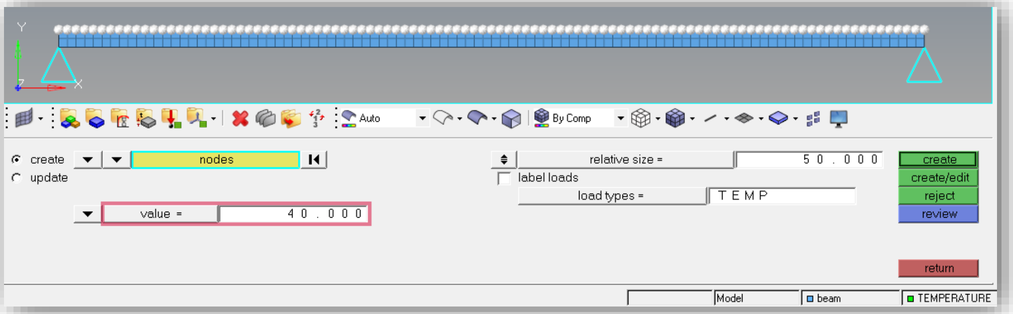


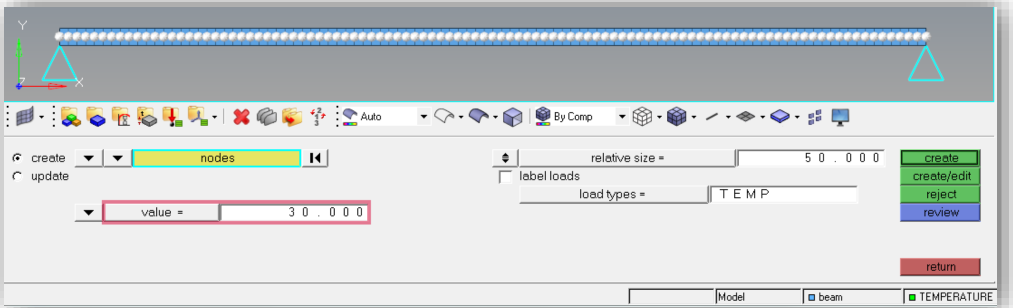
EXERCISE 6A: THERMAL STRESS ANALYSIS OF A BEAM

Hints (2/3)

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



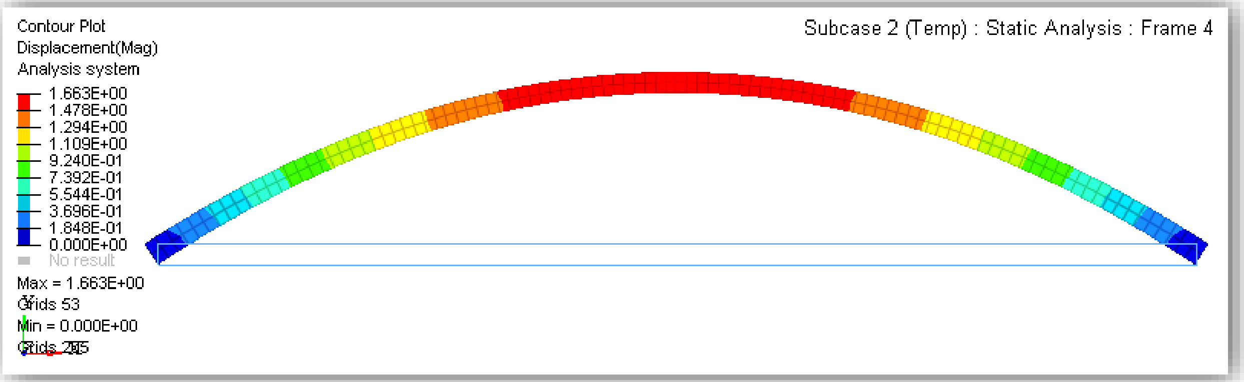


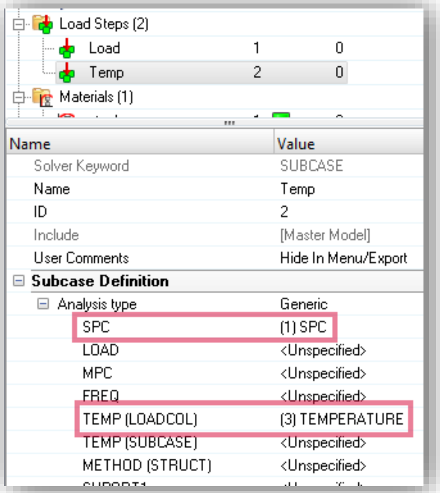


EXERCISE 6A: THERMAL STRESS ANALYSIS OF A BEAM

Hints (2/3)

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





QUESTIONS & ANSWERS

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