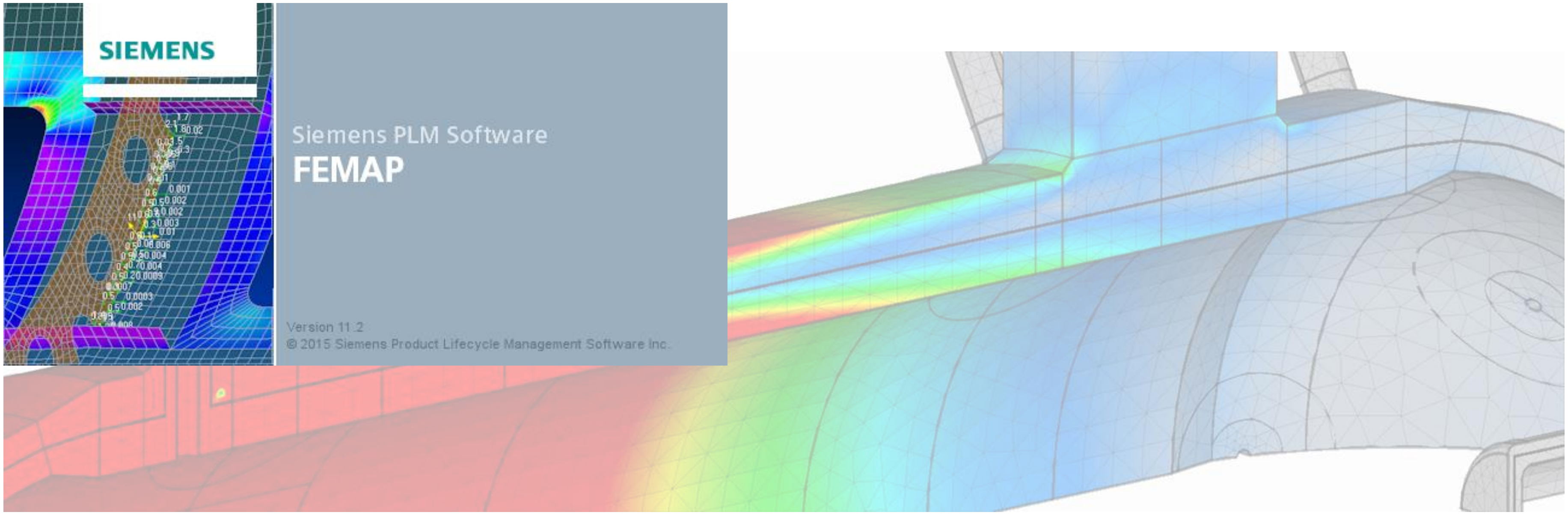


Thermal-Stress Analysis

A Technical Seminar for Femap and NX Nastran Users

George Laird, PhD, PE, Principal Mechanical Engineer

Finite Element Analysis
Predictive Engineering
LS-DYNA Sales, Support & Consulting
www.PredictiveEngineering.com

Applied CAx
Siemens PLM Software Sales & Support
CAD I CAM I CAE I Teamcenter
www.AppliedCAx.com

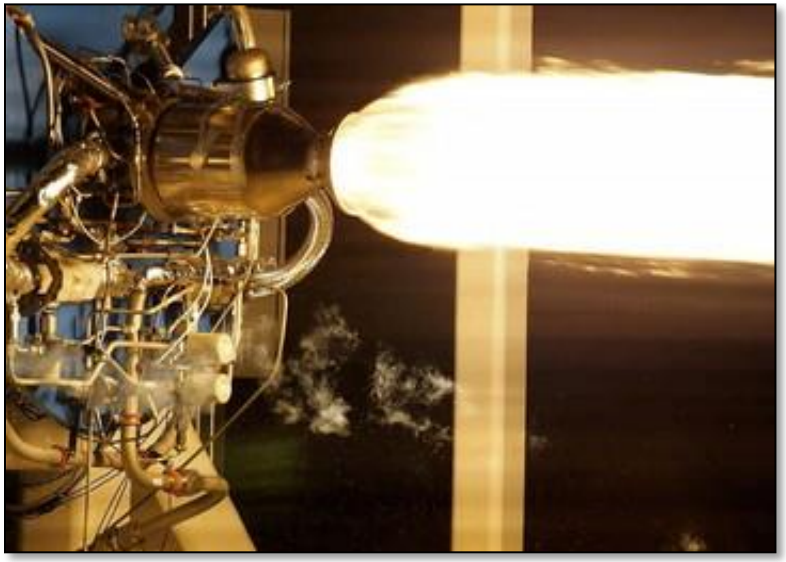
What this white paper covers:

This paper is intended for inquisitive types that would like to know more about the fundamentals of thermal-stress analysis and how to solve such problems using Femap and NX Nastran. This note is our general outline and is intended to accompany the live technical seminar.

- Thermal-Stress Fundamentals
  - Thermally driven stress requires CTE mismatch between materials and/or fixed boundary conditions
- Thermal Analysis
  - Constant Delta T
  - Steady-State and Transient Thermal Analysis
- Thermal-Stress Analysis
  - FEA Process (Thermal Strains to Mechanical Stress)
  - Converting Temperature Results to Thermal-Stress Load Set
  - Brazed Joint Analysis
  - Glued Connections and RBE's in Thermal-Stress Analysis
- Example of Transient Thermal-Stress Analysis
  - Analysis Checklist
  - Processing of Thermal Results
  - Stress Result Visualization

This white paper serves as the foundation for our Femap and NX Nastran technical seminar and can be used independently or dependently with the Seminar's YouTube video. Additionally, examples models used in this White Paper can be found within the Technical Seminar's section of the AppliedCAx.com website.

Femap and NX Nastran Technical Seminar on Thermal-Stress Analysis



Thermal-Stress and Thermal-Deflection analyses are an important subset of general finite element analysis (FEA) modeling. Such analyses are common in the development of rocket motors, ASME pressure vessels, electronics (PCB), electronic systems (automotive lamp systems), composite curing mandrels, generators, satellites and etc.

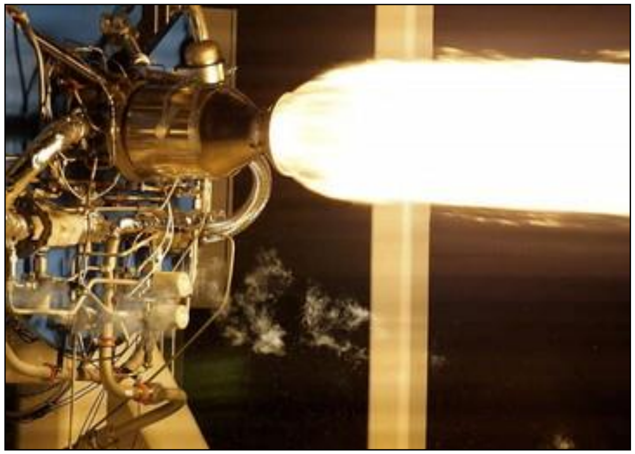
This technical seminar will present the basic principles of linear, thermal-stress and thermal-deflection analysis. We say “linear” since it is starting point if one endeavors to move forward with more complex type of analyses. For this seminar we will use thermal-stress to cover any type of mechanical behavior, stress or deflection introduced by a fixed temperature rise (delta) or an induced temperature gradient. The resulting strain from this temperature load is based on the material’s coefficient of thermal expansion (CTE). The development of stress or deflection within the structure due to this fixed strain and/or variable strain is dependent upon many factors that will be discussed with easy-to-follow basic examples.

With this background, the creation of temperature loads will be discussed using simple boundary conditions or running a steady-state conduction analysis to map out an imposed temperature gradient. These thermal results will then be converted to a temperature load for the thermal-stress or –deflection analyses.

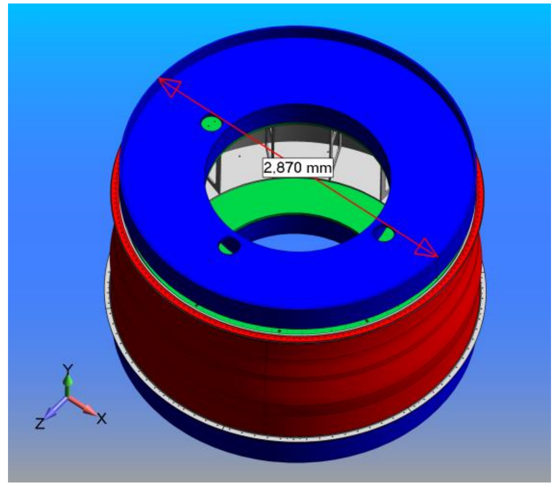
This seminar will close with several examples of thermal-stress work that we have done at Predictive Engineering.

1. SOME EXAMPLES FROM OUR WORK AT PREDICTIVE

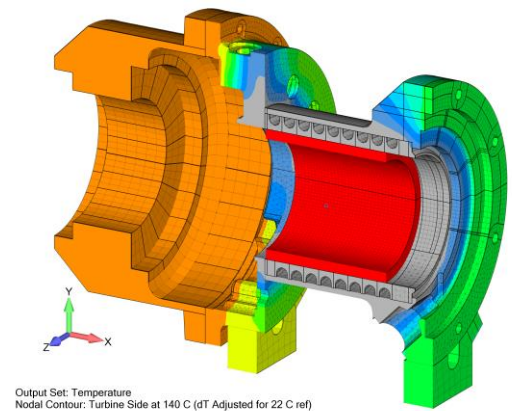
LOX and RP1 Fuel Tanks



Composite Curing Mandrel

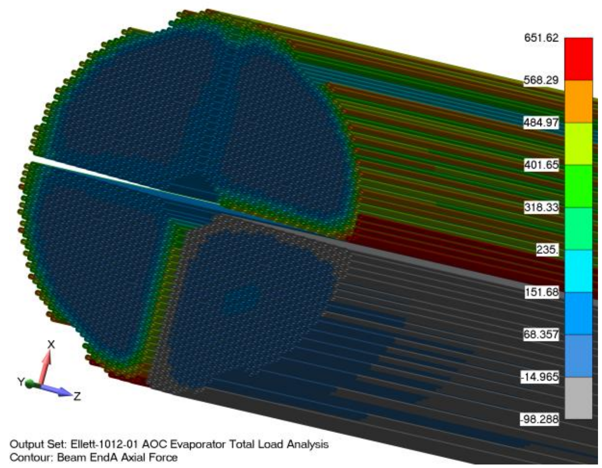


500kW Turbine Generator

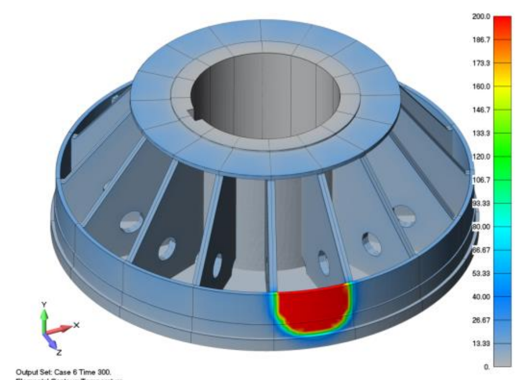


Solar Panel DC to AC Power Converter

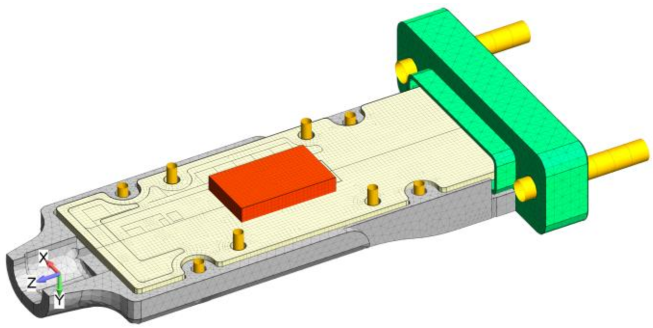
Thermal Differential Expansion ASME Tube Sheet Pressure Vessel



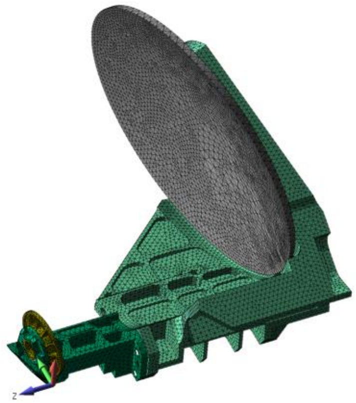
Thermal-Stress Fracture Hydroelectric Generator Thrust Collar



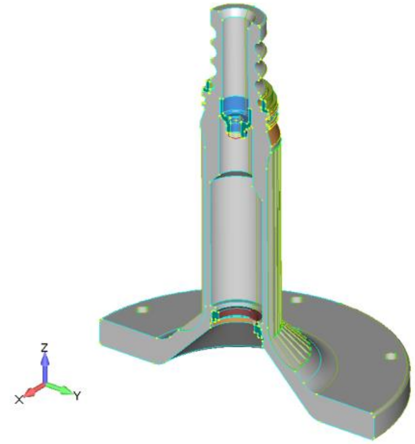
Thermal-Fatigue Analysis of Active Optic Cable



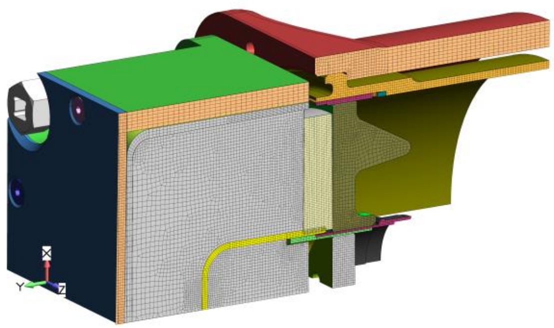
Thermal-Deflection Optical Telescope



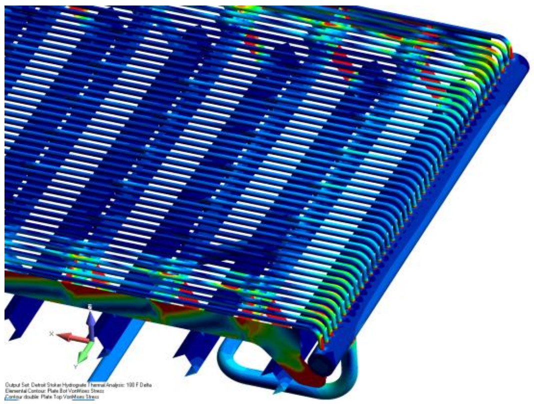
Plasma Tube with Brazed Inserts



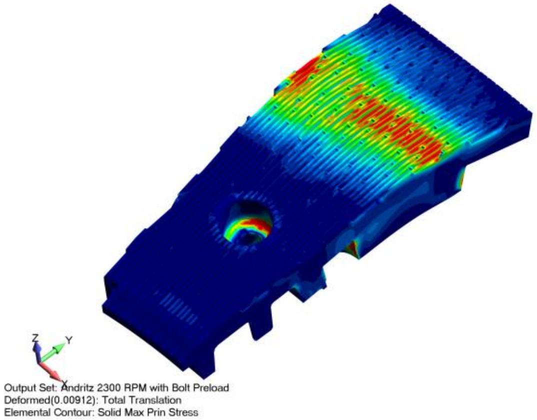
High-Voltage Thermal-Shock Deflection Simulation



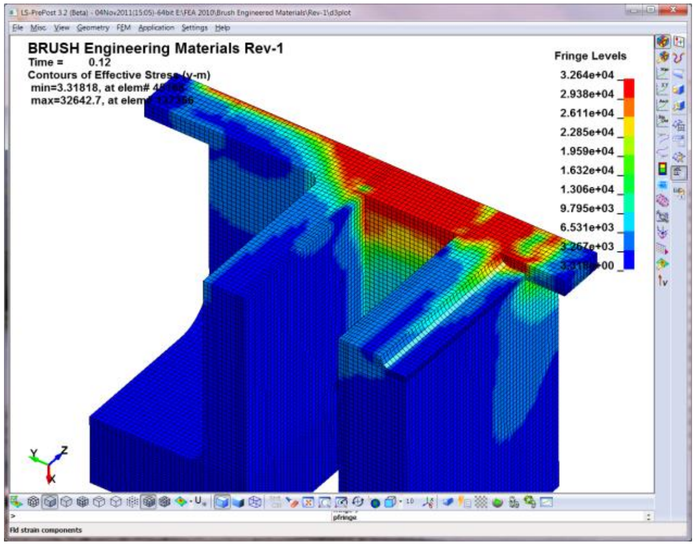
Thermal-Stress Analysis of Water Cooled Furnace Grate



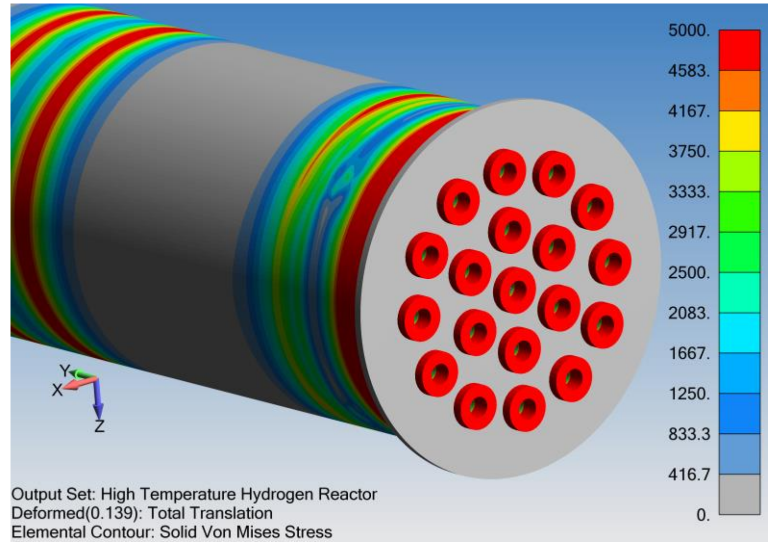
Thermal-Shock Stress Analysis of Pulp Refiner Plate for the Paper Industry



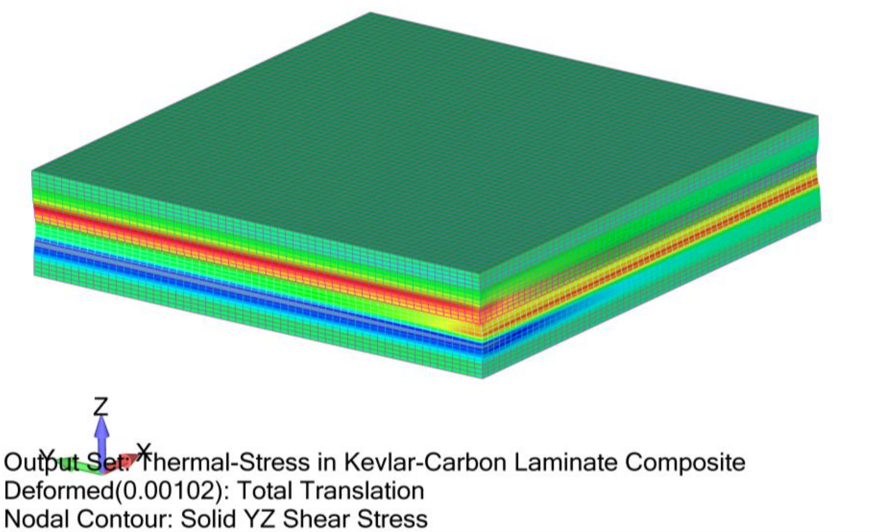
Thermal-Residual Stress Electron Beam Welding



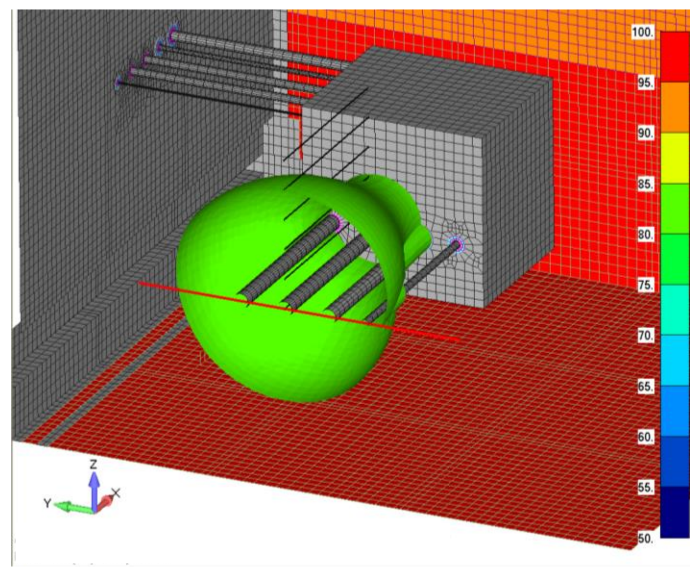
Thermal-Stress in High-Temperature Hydrogen Reactor



Thermal-Stress in Kevlar-Carbon Laminate Composite



Thermal-Stress of Steam Power Supply Piping



2. FUNDAMENTALS OF THERMAL-STRESS ANALYSIS

Thermal-stress or thermal-deflection analysis is driven by strains created in the structure by a temperature load. One of the complexities of this loading is that stresses only develop if the structure is prevented from expanding or contracting or materials with different coefficient of thermal expansion (CTE) are bonded together (e.g., brazed) or mechanical connected (e.g., bolted).

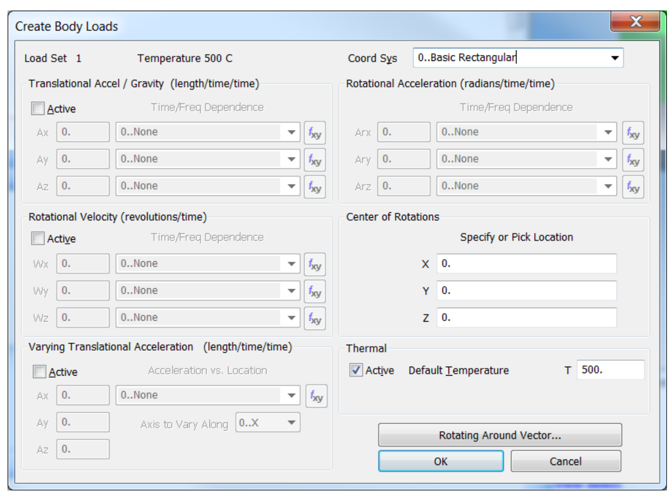
One simple way to think about this is to just picture a chunk of aluminum or steel or some homogeneous material (i.e., not a composite) floating in space. As the structure expands or contract due to temperature, the thermally induced strains do not create any stresses but only deflections, as given by this equation:

$$\varepsilon = \propto (T_{load} - T_{reference})$$

where the strain (ϵ) is determined by the change in temperature and the CTE (α) of the material.

Let's do a simple investigation on how thermal-stresses are developed by looking at some FEA models where stainless steel is brazed onto an alumina structure. The braze material is a nickel alloy but we'll ignore that for the moment. The temperature delta is 500 C and we'll use standard published values of CTE for the two materials. To keep things simple, we'll use beam elements. The temperature differential is applied as a body load of 500 C and the with the reference temperature (strain free) within the material card at 0 C. One could also apply the temperature as a regular nodal or elemental load.

A beam FEA model will be used to illustrate how the construction of the system determines the thermal-stresses.



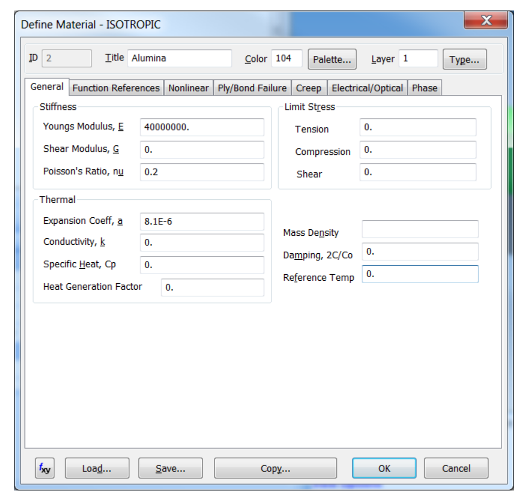


Figure 1 shows the thermal stress results for the two configurations of alumina (gray) and stainless steel (green) beam models. With a linear arrangement, one has deflection but no stress. If they are stacked or layered, then stresses develop due to the differential strain between the layers.

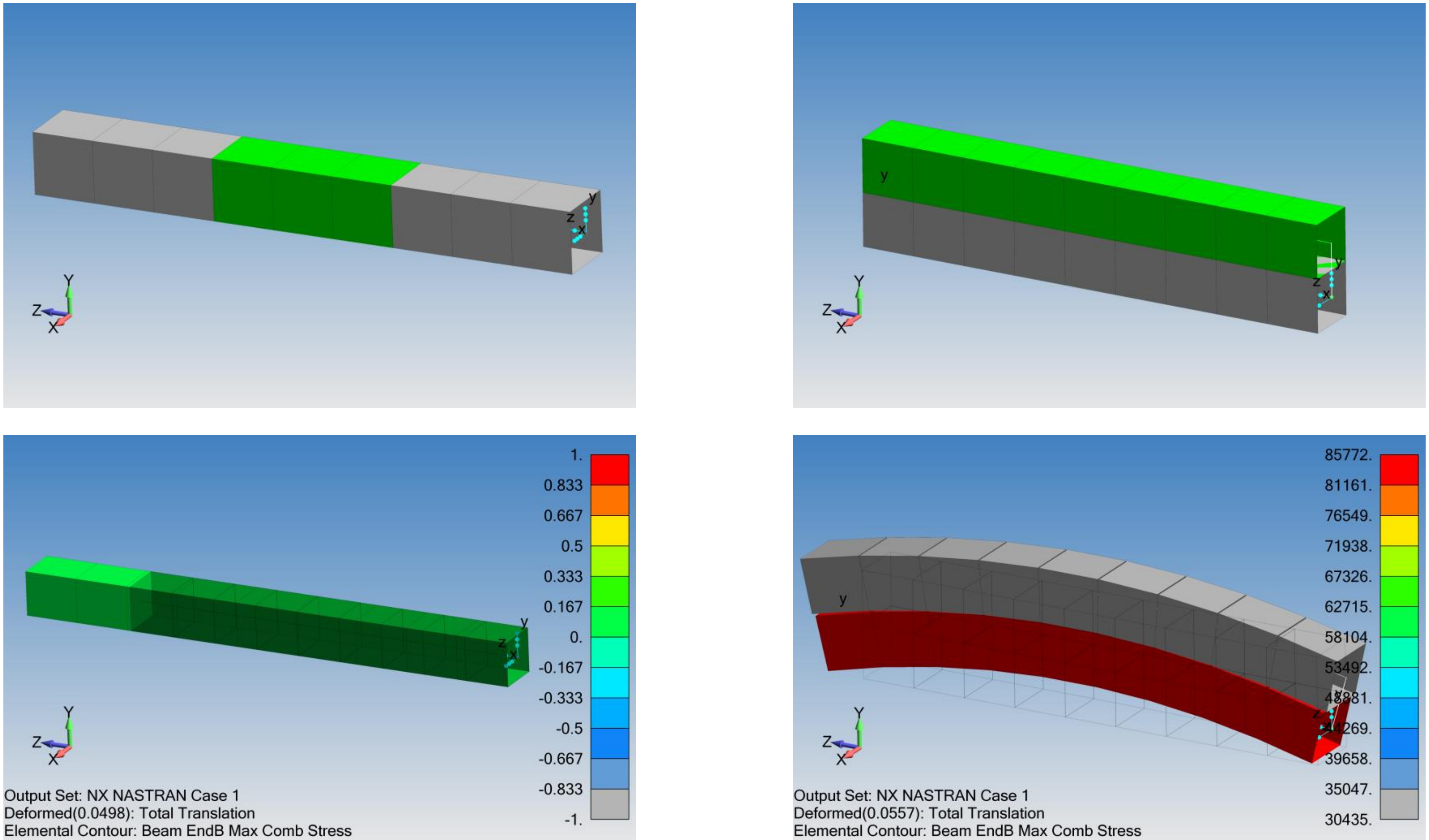


Figure 1: Given a uniform load, how the components are constructed determines their thermal-stresses

The development of thermal-stresses requires two factors: (i) thermal load and (ii) constraint. The constraint factor can be quite obvious or be a product of differential expansion or contraction of connecting parts. The model shown in Figure 2 shows an example of the stainless steel part (green) constrained by the alumina part.

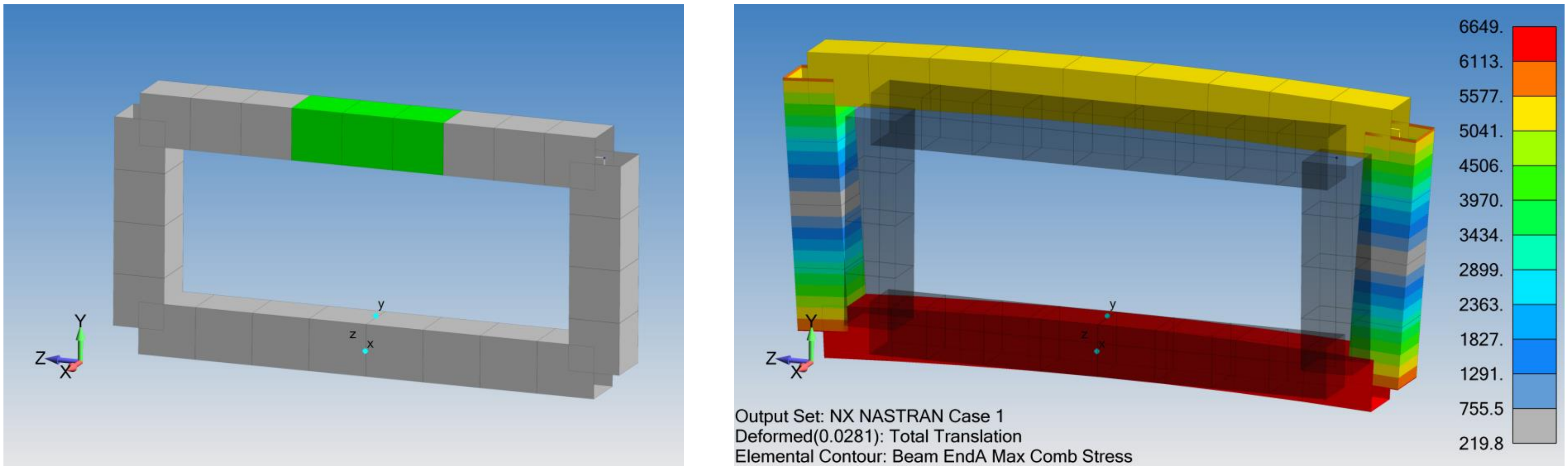


Figure 2: Given constraint by a surrounding part, thermal-stresses can develop due to differential strains

2.1 Shrink-Fit

We would be remiss not to include a shrink-fit model as an example of useful thermal-stress analysis. In this model, the pin is dipped in liquid nitrogen and allowed to cool to -200 C. The block is heated to 200 C. For ease of assembly the pin is sized to have a radius 0.025 mm smaller than the hole in the block. The goal of this example is to determine if this shrink fit will cause plastic deformation in the block.

Figure 3 shows a maximum von Mises stress near 1,000 MPa. Given our stainless steel material model, the design is not adequate to prevent plastic yielding.

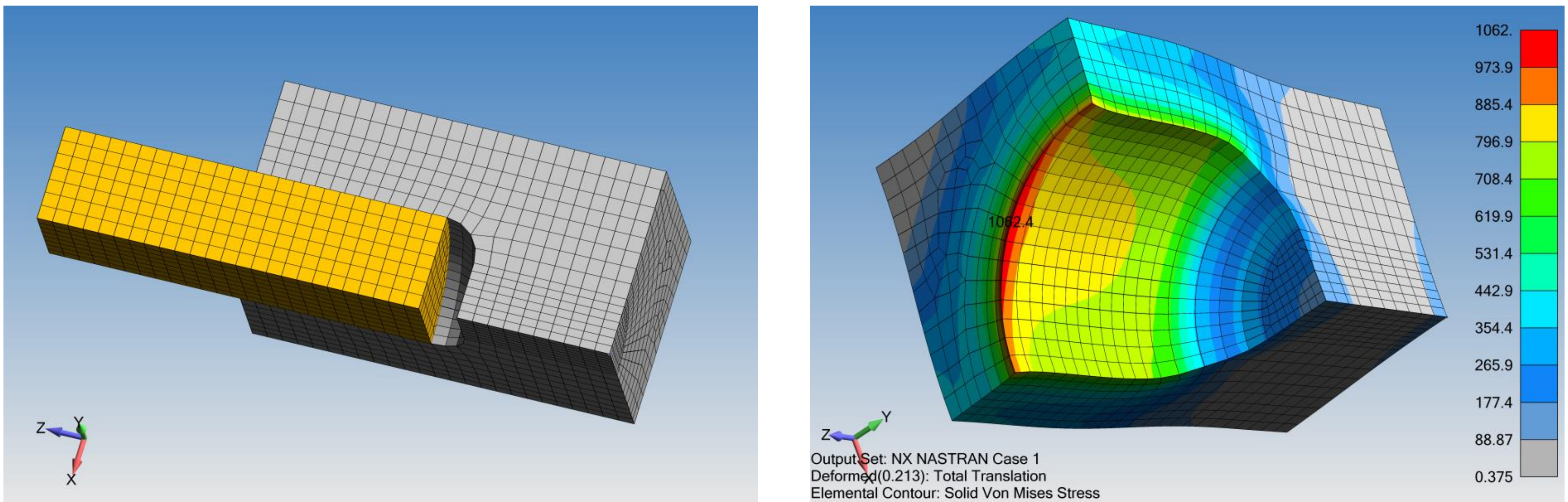


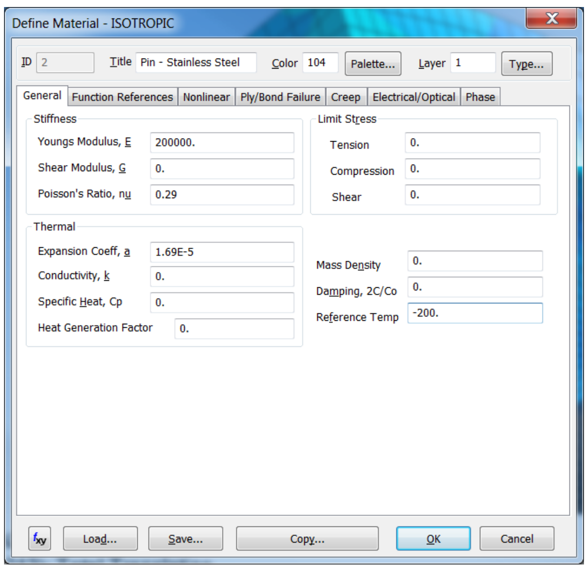
Figure 3: Thermal-stress shrink fit using linear contact

3. THERMAL-ANALYSIS

Setting up the thermal profile to use as a load case to a structural model can be easy or hard and there is obviously dependent upon the thermal event that is being is being simulation. In our work, we have mapped thermal profiles from static and transient CFD simulations and have done fully coupled electron-beam welding simulations where phase change and residual plastic strains are captured within the work piece. These examples represent complex thermal analyses that are beyond the scope of this discussion and if your work entails such needs, please contact us and we can provide some guidance. For this discussion, we will stick with simple thermal loads that can be obtained using a fixed temperature delta or from steady-state temperature gradient or using a data surface.

3.1 Uniform Temperature Delta (ΔT)

In the material card, a "Reference Temp" can be applied. This can be thought of as the material's strain-free temperature. To create the ΔT from this strain-free temperature, a load case is created where the Body Loads / Thermal Default Temperature represents the final temperature state (see Figure 4).



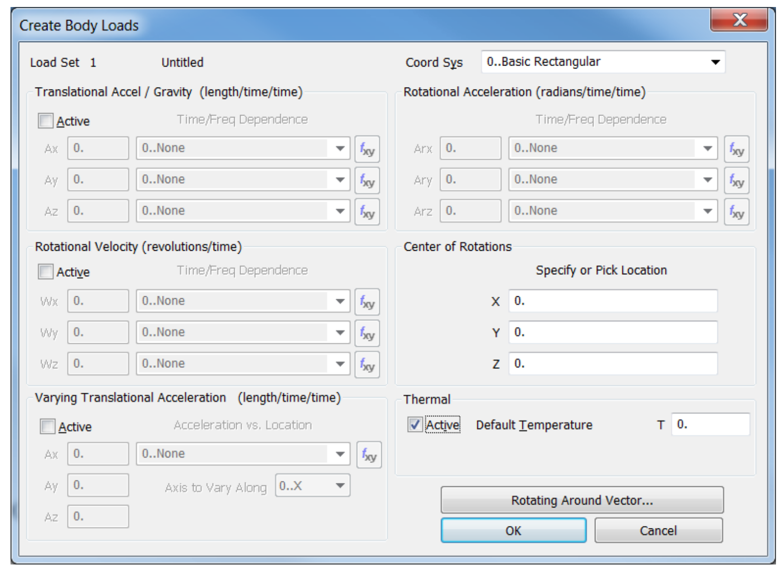


Figure 4: A quick way to define a fixed ΔT in a model

Alternatively, one can set the reference temperature to 0 and use nodal temperatures to drive the thermal strains in any number of ways within the model. For example, in the Shrink Fit example, one material model could have been used with the Reference Temp = 0.0 with a temperature load case of +200 for the pin and -200 for the block. This setup is shown in Figure 5.

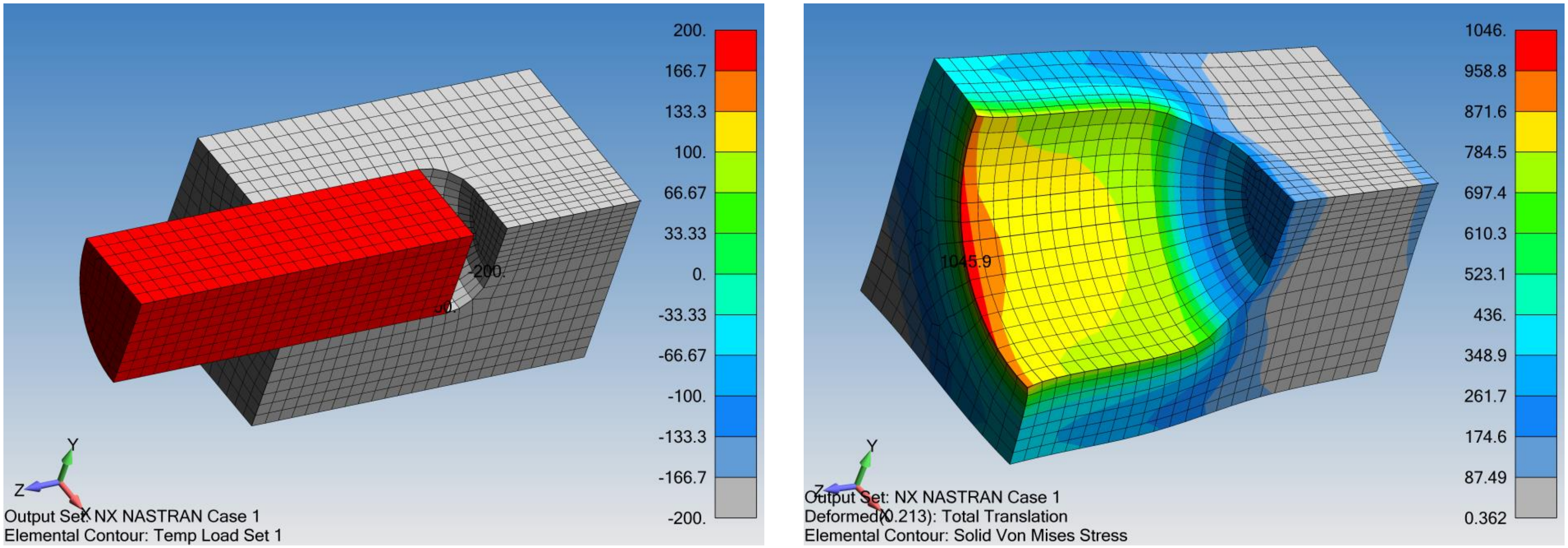


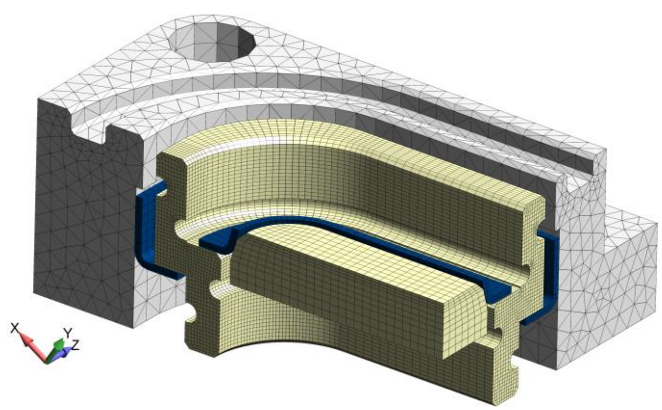
Figure 5: Alternative thermal load application using nodal temperatures rather than Body Load

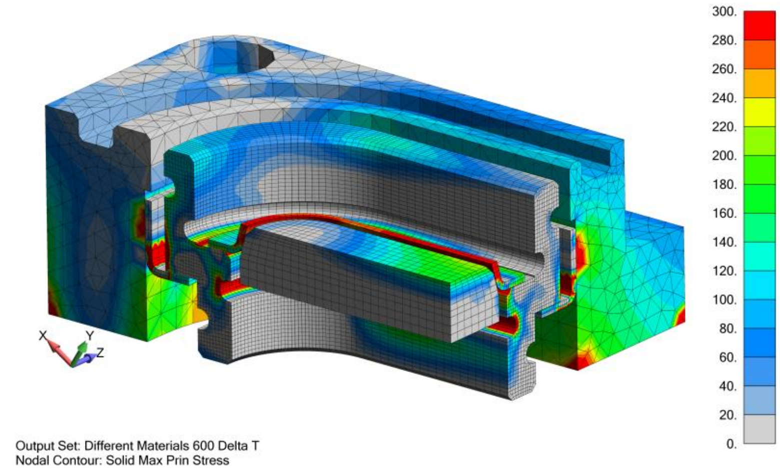
3.2 Summary of Thermal Load Application

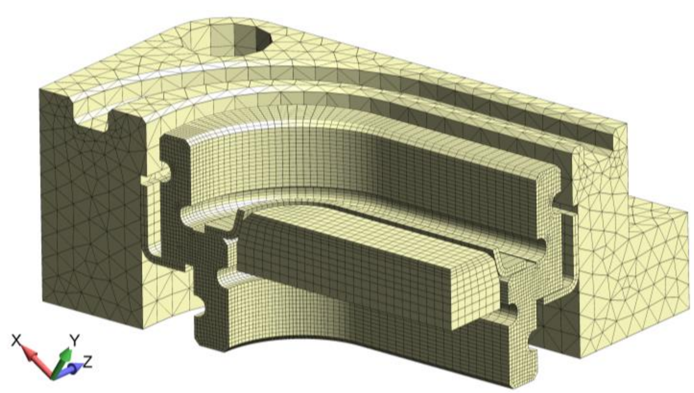
- Material Card (Reference)
- Body Load (Default)
- Nodal Load (Steady-State or Transient Load Application)

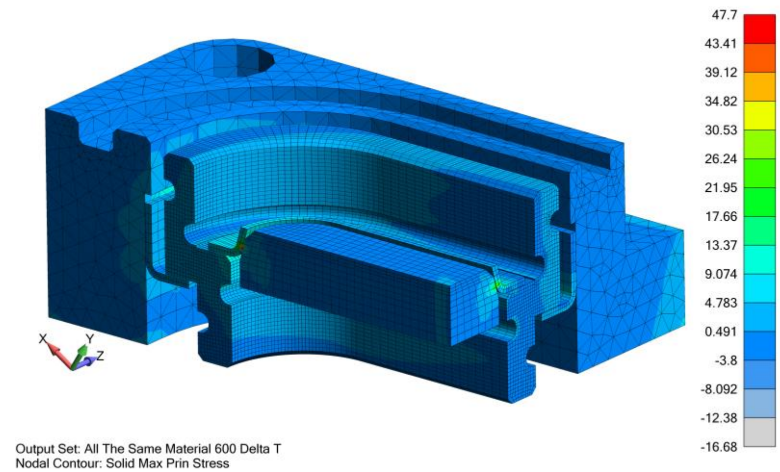
3.3 Glued Connections and RBE's in Thermal-Stress Analysis

Interfaces and RBE's can create challenges in a thermal-stress analysis due to their abrupt discontinuity of the mesh. Physically one can imagine that any sharp interface in a FE model is not reflected in physical world. This is most clearly apparent in brazed connections where differences in CTE between the components will create a sharp stress discontinuity. Even with the same materials, the glued connection creates a bogus stress of 50.

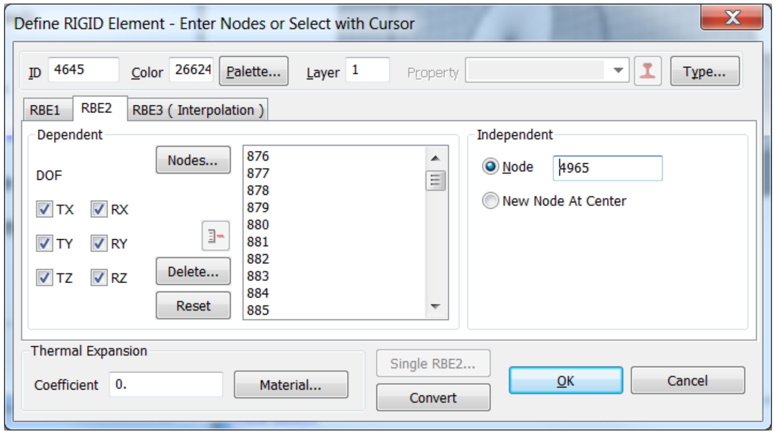


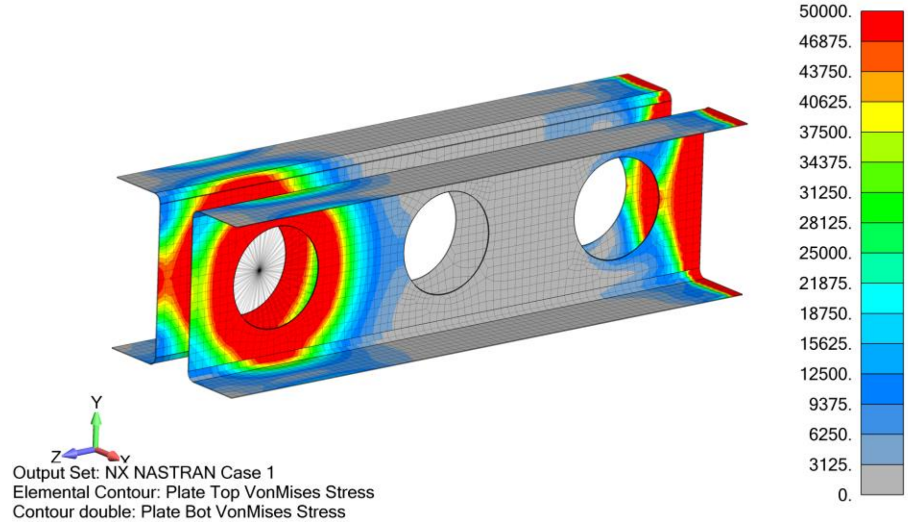


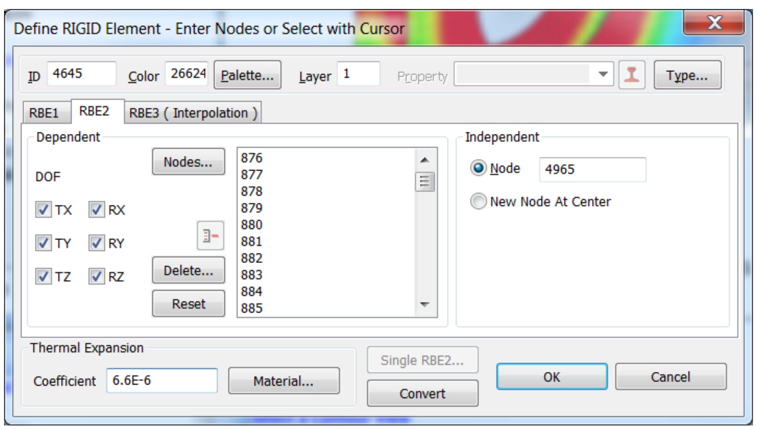


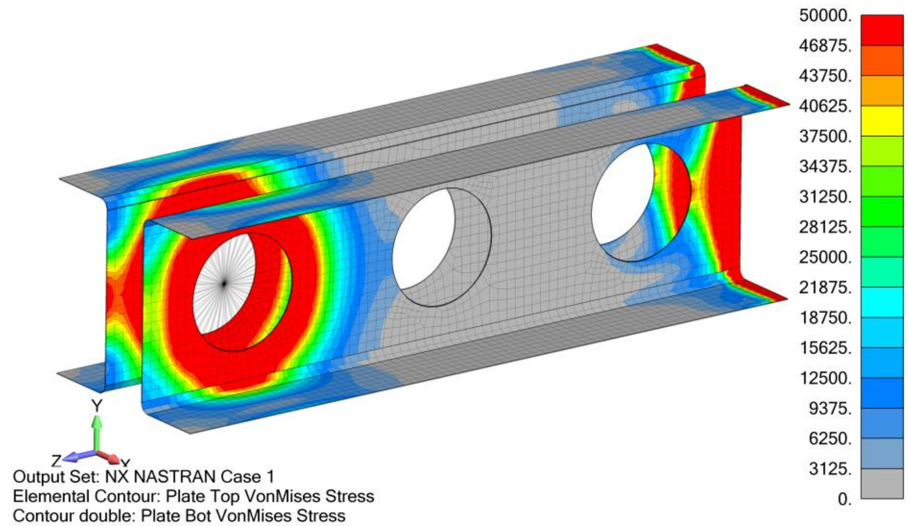


If one understands that RBE's are multi-point constraint equations, then the following results might seem logical except for the non-responsive behavior upon requesting that the RBE2 use the CTE of the parent material.







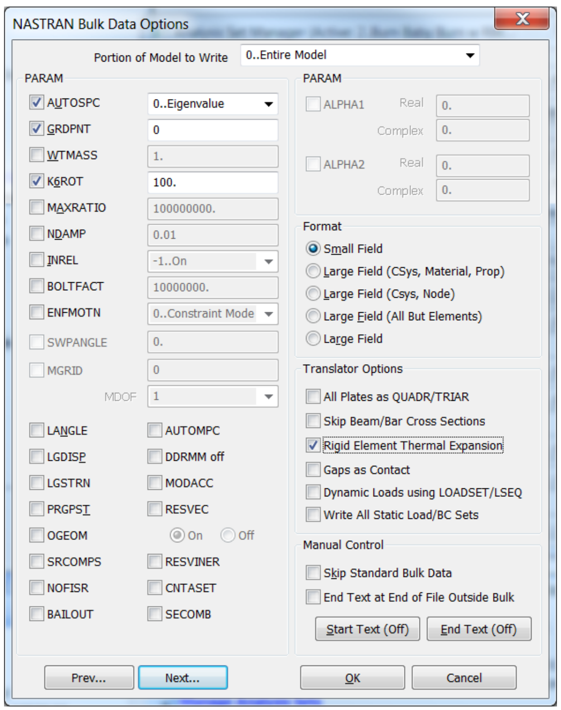


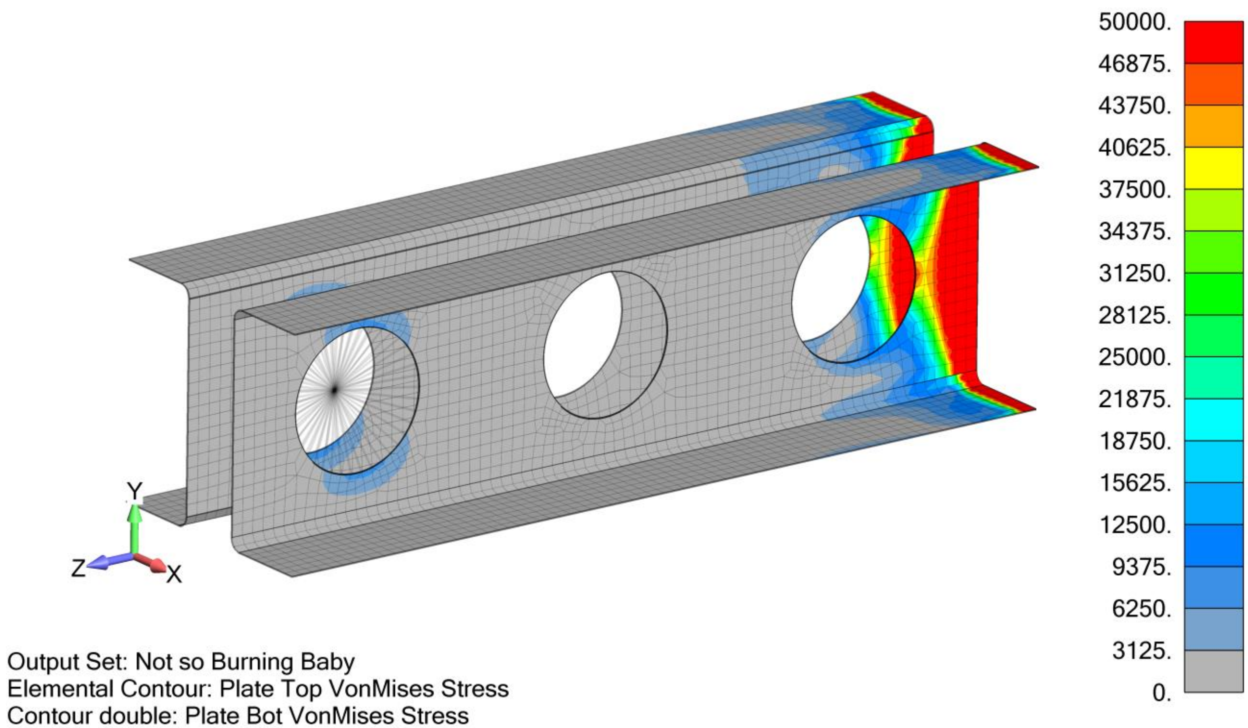
Well, it all depends upon your load application. In the prior example, we didn't switch over the formulation within the analysis manager. What is going on is that the RBE2 element is no longer a MPC element but a rigid spring-type element.

One could also switch to a RBE3 element but then it is a force interpolation and not "rigid".

Analysis Setting Required to Activate RBE2 Formulation Switch

Not so Hot



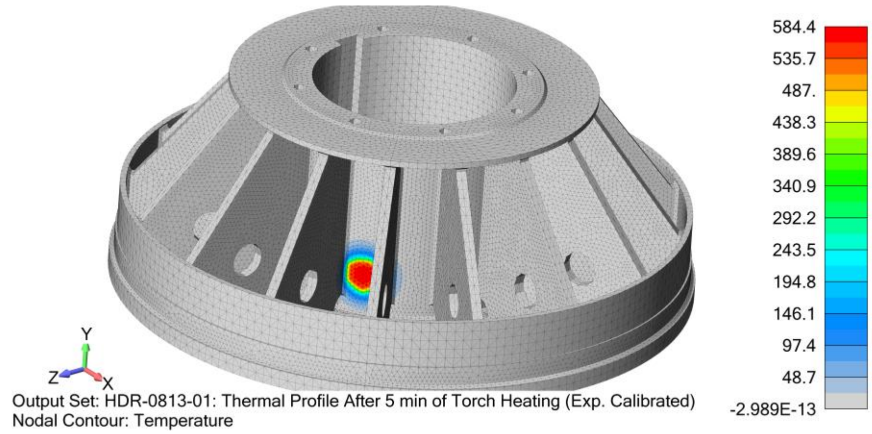


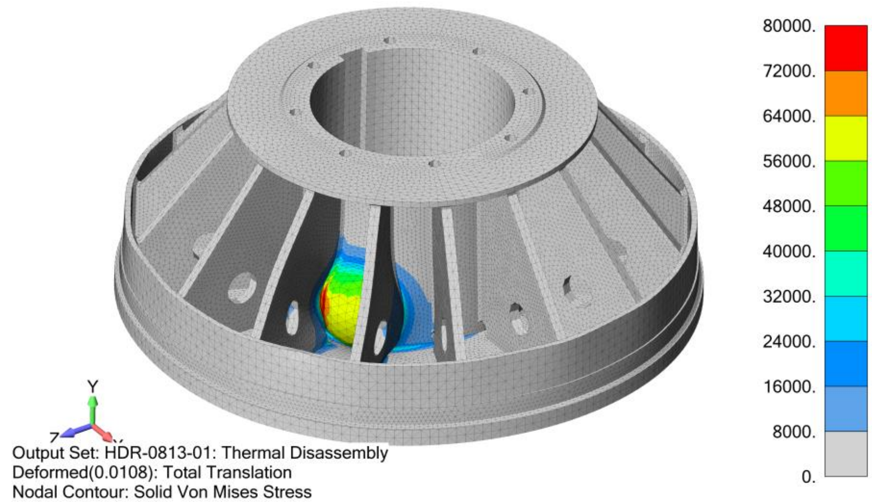
4. TRANSIENT THERMAL-STRESS ANALYSIS

If specific temperatures are known at boundaries, a steady-state conduction analysis can be used to map these temperatures into the structure. Likewise, if the geometry is relatively simple, Data Surfaces can directly create the temperature field to drive the structural load case. In this example, we'll show how to do a transient thermal analysis and convert the temperature results into a structural load case.

Workflow:

- Create temperature load with or without time function
- Set k (thermal conductivity) and C_p (specific heat) for material. Ensure consistent structural/thermal units – please note it can be messy with English units.
- Run transient thermal analysis and request outputs at desired time intervals
- Convert thermal results into load cases (Model / Load / From Output
- Run static stress analysis





4.1 Transient Thermal Stress Analysis in SI and English

This example will walk thru a basic transient thermal analysis that can be done in either unit system. To get comfortable with the units, it is helpful to see how the units of a thermal analysis work via the basic equations:

$$q = k \cdot L \cdot (\Delta T) \text{ and } q = c_p \cdot mass \cdot \frac{dT}{dt}$$

In SI units, q is J/s or Watts which then leads to k in (N·m)/(s·m·°C) and c_p in units of (N·m)/(kg·°C), while in English units we have q as lbf·in/s and k as lb_f·in/in·s·°F and c_p as lb_f·in/snail·°F.

Note: As one can see, it is why many simulation engineers prefer to work in SI units for thermal analysis work.

Here's our conversion table from SI to English:

Thermal Conductivity	1 W/m·°K	0.1249 lb_f·in/in·s·°F
Specific Heat	1 J/kg·°K	861.1 lb_f·in/snail·°F
Convection Coefficient	1 W/m^2·°K	3.172E-03 lb_f·in /s·in^2·°F
Temperature	1 °K	1.8 °F

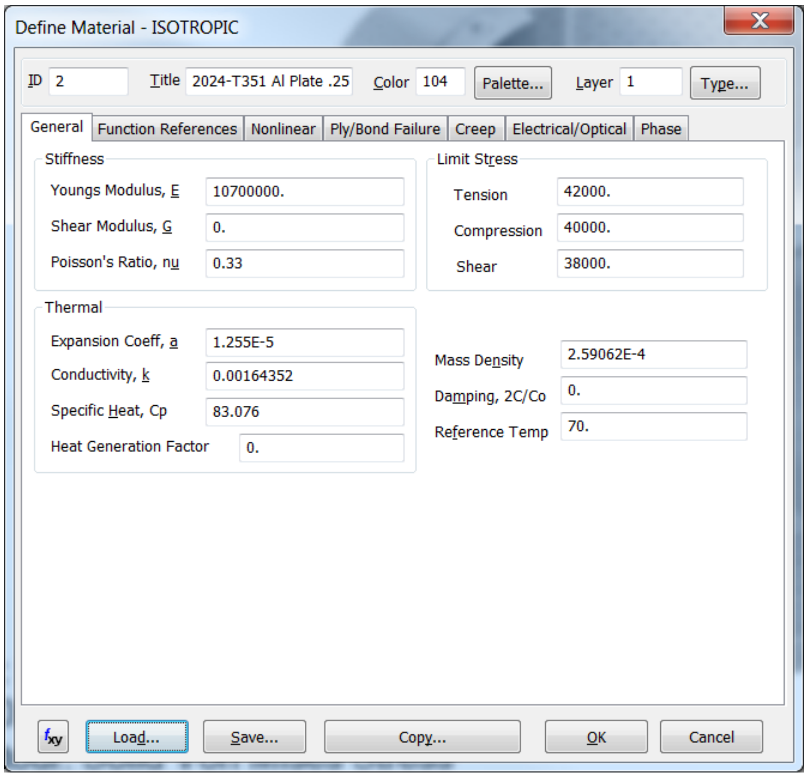
The table below provides the unit system for the analysis of an aluminum structure:

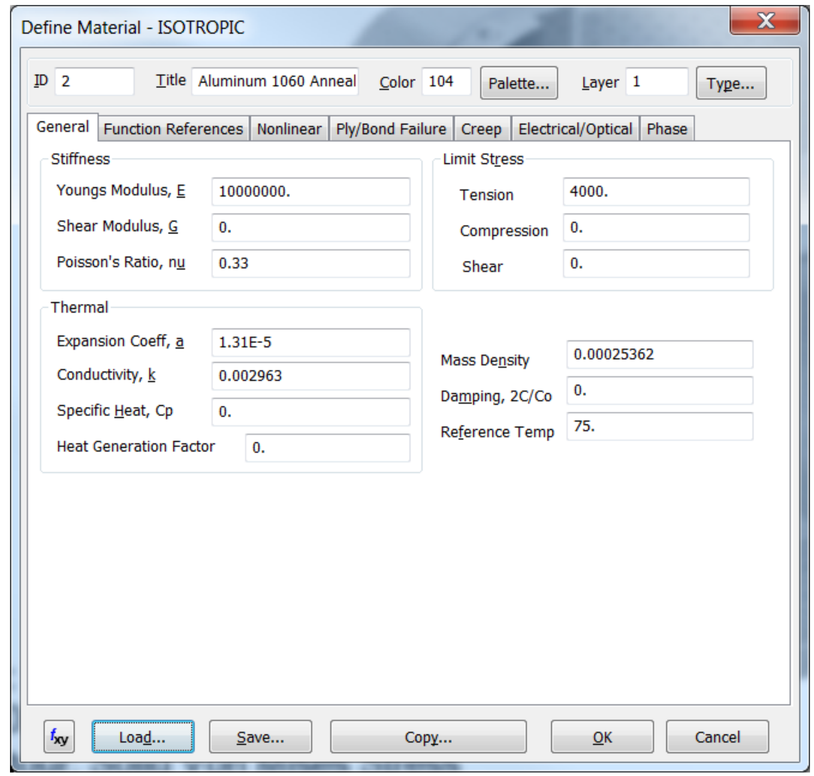
SI (N, mm, ton, s, C)	E MPa	γ	Mass Density ton/mm^3	k (N·mm)/(s·mm·C)	C_p (N·mm/(ton·C)	CTE 1/°C
	70e+03	0.33	2.77e-09	170	896e+06	2.3e-05
English (lbf, in, snail, F)	E PSI	γ	Mass Density lbf·sec^2/in	k lb_f·in/in·s·°F	C_p lb_f·in/snail·°F	CTE 1/°F
	10e+06	0.33	2.53e-04	21.2	772e03	1.3Ee-05

4.2 An Example of Transient thermal-Stress analysis with API Processing

One of the simple hurdles of doing a transient thermal-stress analysis is just understanding how the units fit together. If one has to depend upon the default unit setup within Femap and NX Nastran as a guide, then you might be challenged since the default setting has the thermal energy unit in BTU's.

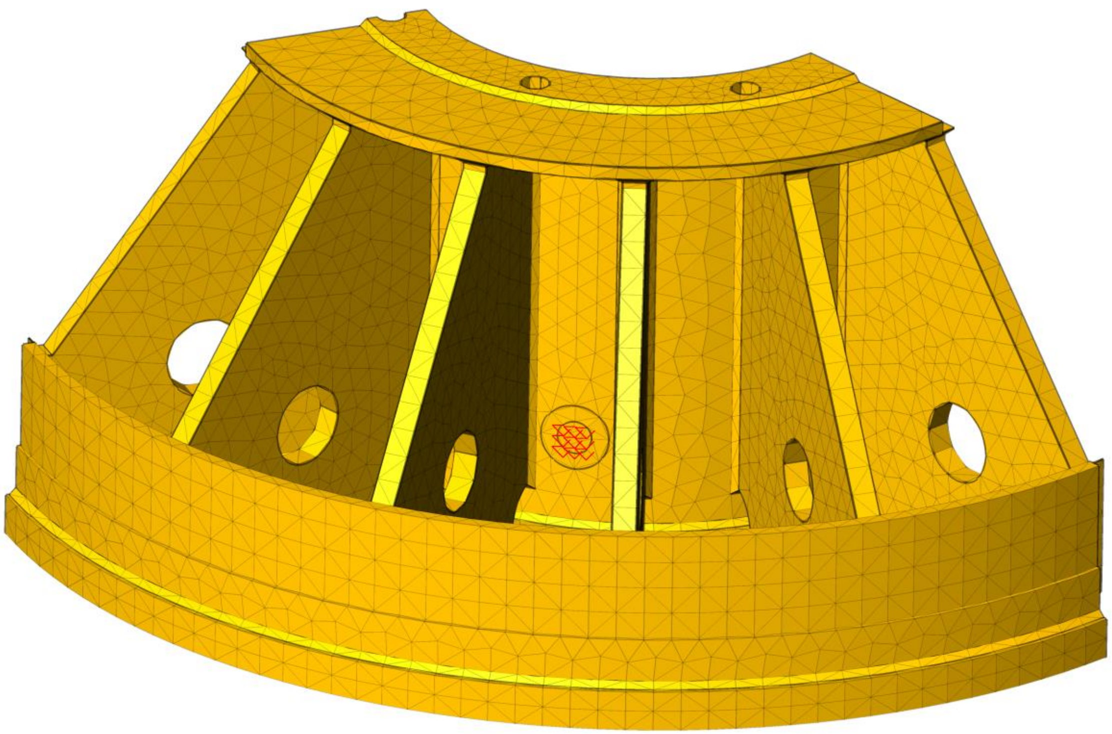
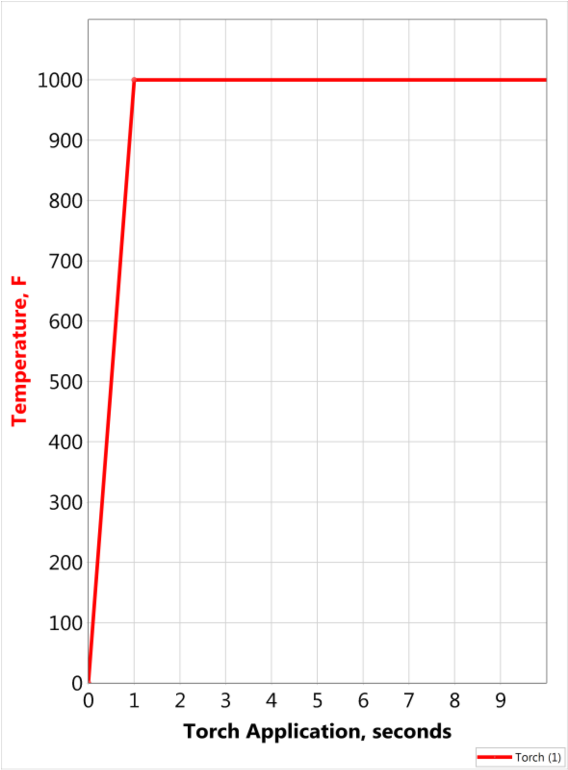
Default (material.esp) Aluminum (~English)	mat_eng_in-lbf-psi-degF-BTU.esp Aluminum





Let's back-up and decipher these units. First off, please note that the energy system is in BTU and not in our base units. One BTU is equal to 1054.35 W or 9,331.4 lbf·in/s. As for the Specific Heat value reported in the default set, it is also likewise in BTUs and when multiplied by the BTU conversion, one has 774,500 lbf·in/snail·°F. From an engineering viewpoint, the results are the same whether one uses BTU or lbf·in/s as the energy unit, one just has to be consistent through the whole process whether steady-state (only k) or transient (k and c_p).

Here's our transient thermal example – a chopped down thrust collar block:

Cut-Down Section of Thrust Collar / Torch Heating	Temperature Ramp Function
	

Our analysis setup for a thermal-stress analysis is given below:

Checklist

Thermal

k and C_p values defined

Time Function

Thermal Load

- ***Initial***
- ***Transient***

Analysis: transient heat transfer

Stress

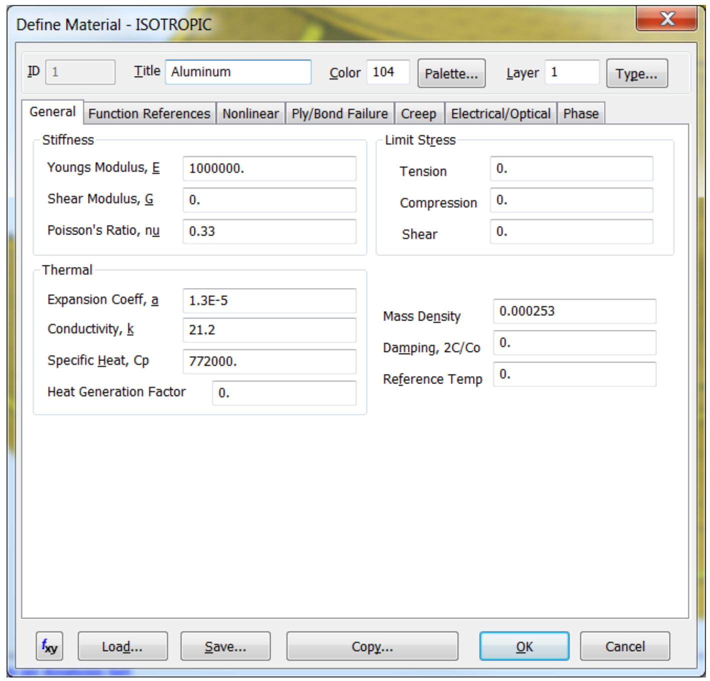
E, nu, mass density and CTE

Temperature Load

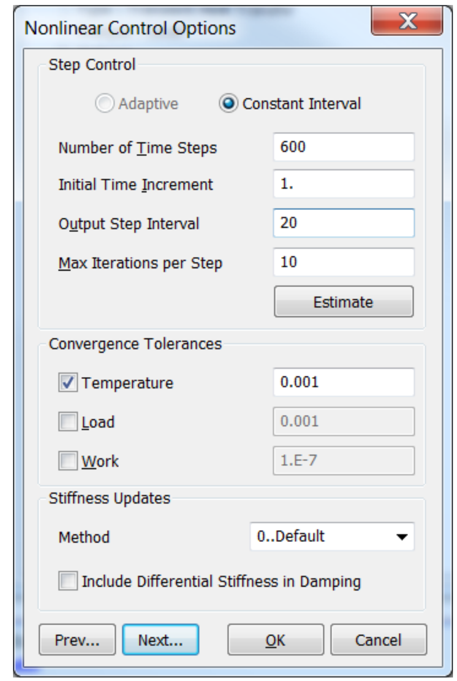
Constraint Set

Analysis: static

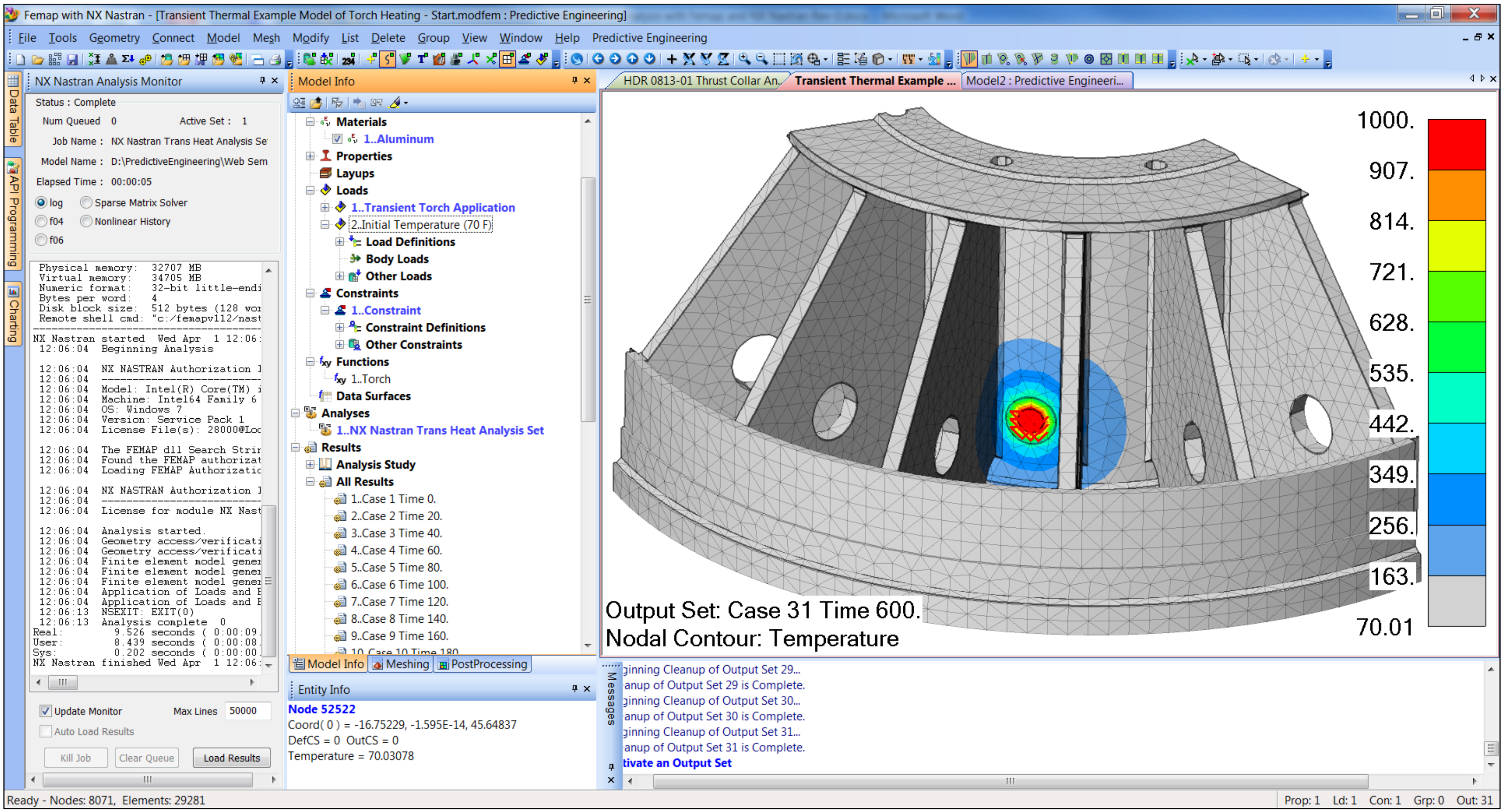
Material Specification for Thermal-Stress Simulation



Transient Heat Transfer



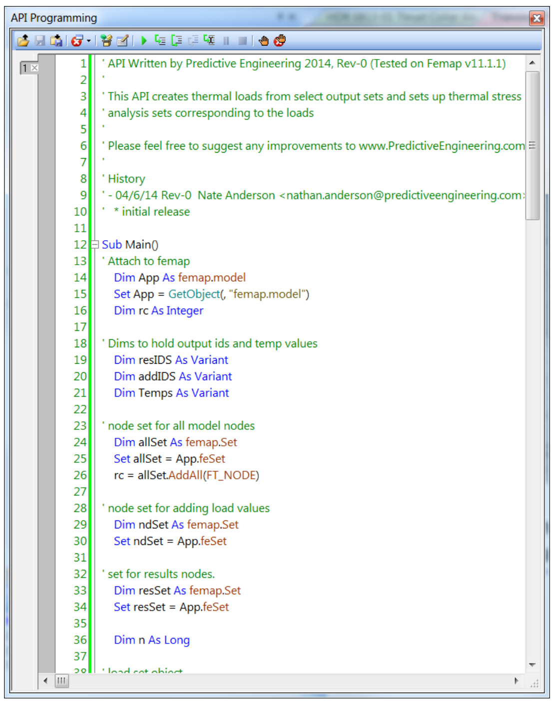
The transient thermal analysis creates 30 results sets and can be animated using one of the new options within Femapv11.2.0. The analysis uses an initial temperature of 70 F.

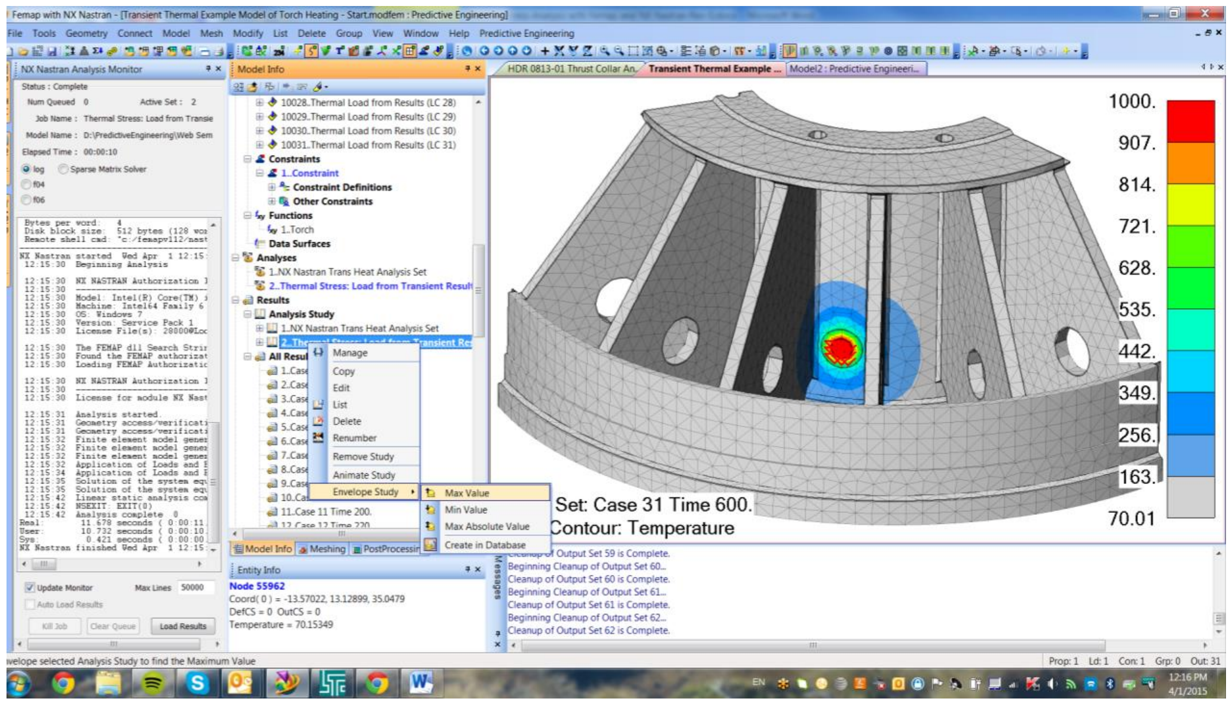


One dilemma with transient thermal analyses is to determine which thermal load set might create the highest thermal-stress in the system. There are many tribal methodologies that may be advocated or one can use the brute force approach and just perform a stress-analysis on sufficiently fine set of thermal results. We will show the later since it provides the most robust way to assure a reviewer that the worst-case behavior has been captured.

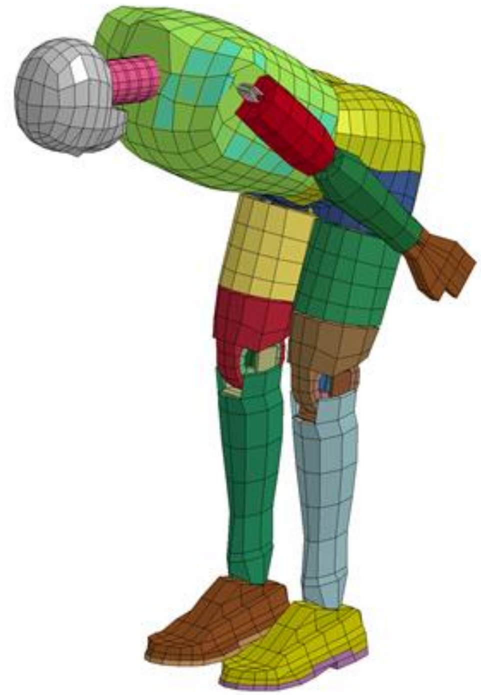
Femap Application Programing Interface (API) Automation Program for Thermal-Stress Analysis

The API requests thermal results and a constrain set. Load cases are then created and a new Analysis Set is created. When analyzed, 30 result sets are created and can be Enveloped for Max Value.





Thank You



Predictive Engineering is located in Portland, OR

PredictiveEngineering.com