

Thermal-Stress Analysis

A Technical Seminar for Femap and NX Nastran Users

George Laird, PhD, PE, Principal Mechanical Engineer





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
 - o Constant Delta T
 - o 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
 - o 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

Predictive Engineering

FINITE ELEMENT 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



Solar Panel DC to AC Power Converter

Composite Curing Mandrel



Thermal Differential Expansion ASME Tube Sheet Pressure Vessel 500kW Turbine Generator



Thermal-Stress Fracture Hydroelectric Generator Thrust Collar









Thermal-Fatigue Analysis of Active Optic Cable **Thermal-Deflection Optical Telescope Plasma Tube with Brazed Inserts Thermal-Stress Analysis of Water Cooled Thermal-Shock Stress Analysis of Pulp** High-Voltage Thermal-Shock Deflection Simulation Furnace Grate **Refiner Plate for the Paper Industry** Output Set Andritz 2300 RPM with Bolt Preloan eformed(0.00912): Total Translation emental Contour: Solid Max Prin Stress



Thermal-Stress in Kevlar-Carbon Laminate Composite



Thermal-Stress in High-Temperature Hydrogen Reactor



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:

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

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 thermalstresses.

Create Body Loa	ds			Define Material - ISOTROPIC
Load Set 1	Temperature 500 C	Coord Sys	0Basic Rectangular	ID 2 Itle Alumina Color 104 Palette Layer 1 Type
Translational Ac	cel / Gravity (length/time/time)	Rotational Ac	eleration (radians/time/time)	General Function References Nonlinear Ply/Bond Failure Creep Electrical/Optical Phase
Active	Time/Freq Dependence		Time/Freq Dependence	- Stiffness
Ах 0.	0None 🔻 🖡	Anx 0.	0None 💌 🖡	Youngs Modulus, E 40000000. Tension 0.
Ay 0.	0None	Ary 0.	0None	Shear Modulus, <u>G</u> 0. Compression 0.
A7 0.	0None	Arz 0.	0None	Poisson's Ratio, n <u>u</u> 0.2 Shear 0.
				Thermal
Rotational Veloc	ity (revolutions/time)	Center of Rot	ations	Expansion Coeff, a 8.1E-6 Mass Density
Active			Specify of Pick Location	Conductivity, <u>k</u> 0. Damping 2C/Co 0.
Wx 0.	0None 🔻 🖡		χ 0.	Specific Heat, Cp 0.
Wy 0.	0None 🔻 🖡		Υ 0.	Heat Generation Factor 0,
Wz 0.	0None 🔻 🖡		Ζ 0.	
Varying Translat	tional Acceleration (length/time/time)	Thermal		
Active	Acceleration vs. Location	Act <u>i</u> ve [Default Temperature T 500.	
Ах 0.	0None 🔻 🖡			
Ay 0.	Axis to Vary Along 0X 💌			
A7 0			Rotating Around Vector	
			OK Cancel	fxy Load ⊆ave Copy ΟΚ Cancel

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

X

Figure 3: Thermal-stress shrink fit using linear contact



Output Set: NX NASTRAN Case 1 Deformed(0.213): Total Translation

Elemental Contour: Solid Von Mises Stress

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.



1062. 973.9 885.4 796.9 708.4 619.9 531.4 442.9 354.4 265.9 177.4

88.87

0.375



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).

eral Function Refere	ences Nonlinear Ply/Bon	d Failure Creep Electri	cal/Optical Phase	
tiffness		Limit Stress		
/oungs Modulus, <u>E</u>	200000.	Tension	0.	
Shear Modulus, <u>G</u>	0.	Compression	0.	
²oisson's Ratio, n <u>u</u>	0.29	Shear	0.	
hermal				
Expansion Coeff, <u>a</u>	1.69E-5		0	
Conductivity, <u>k</u> 0. Specific <u>H</u> eat, Cp 0.		Mass De <u>n</u> sity	0	
		Damping, 2C/Co	0.	
leat Generation Facto	or 0.	Reference Temp	-200.	

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

	elI	Untitled		Coord	Sys	0E	Basic Rectangular	-	
Transl	ational A	ccel / Gravity (length/time/tim	ie)	Rotati	onal Ad	celer	ation (radians/time/ti	me)	
<u>A</u> c	tive	Time/Freq Depende	nce				Time/Freq Depe	ndence	
Ax [0.	0None	▼ f _{xy}	Arx	0.		0None	-	f _{ag}
Ay [0.	0None	▼ f _{xy}	Ary	0.		0None	•	f _x
Az [0.	0None	▼ f _{yy}	Arz	0.		0None	•	f _x
Wy [Wz [0. 0.	0None	• f _{x0}			Y Z	0.		
Varyin	g Transla tive	ational Acceleration (length/ti Acceleration vs. Loca	me/time)	Therm	nal :tįve	Defau	ult <u>T</u> emperature	т 0.	
Ax [0.	0None	▼ f _x	/					
	0	Avis to Vory Alana	X T						

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.

Define RIGID Element - Enter Nodes or Select with Cursor	X
JD 4645 Color 26624 Palette Layer 1 Property RBE1 RBE2 RBE3 (Interpolation)	Type Independent O Node +965 New Node At Center
Thermal Expansion Single RBE2. Coefficient 0. Material Convert	<u>O</u> K Cancel



Define RIGID Element - Enter Nodes or Select with Cursor JD 4645 Color 26624 Palette Layer Property	✓ I Type
RBE1 RBE2 RBE3 (Interpolation) Dependent Nodes 876 DOF 878 879 V TX V RX 879 V TY Ry 3" 881 882 V TZ V Rz Belete 884 885 *	Independent Node 4965 New Node At Center
Thermal Expansion Coefficient 6.6E-6 Material Convert	ОК Сапсеі



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



Output Set: HDR-0813-01: Thermal Profile After 5 min of Torch Heating (Exp. Calibrated) Nodal Contour: Temperature



-2.989E-13

2015

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:

$$q = k \cdot L \cdot (\Delta T)$$
 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 $(N \cdot m)/(s \cdot m \cdot °C)$ and c_p in units of $(N \cdot m)/(kg \cdot °C)$, while in English units we have q as $lbf \cdot in/s$ and k as $lb_{f} \cdot in/in \cdot s \cdot °F$ and c_p as $lb_{f} \cdot in/snail \cdot °F$.

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/m²⋅°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 E)	E PSI	γ	Mass Density Ibf·sec ² /in	k Ib _f ∙in∕in∙s∙°F	C _p Ib _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)

	024-1	351 AI Fidle	.25 <u>C</u> olor	104	Pal	ette	Layer 1		Ту <u>р</u> е
Seneral Function Refer	ences	Nonlinear	Ply/Bond Fai	lure Cr	еер	Electri	cal/Optical	Phase	
Stiffness					Stre	SS			
Youngs Modulus, <u>E</u> 10700000.			Te	nsior	ı	42000.			
Shear Modulus, <u>G</u> 0.			Co	mpre	ession	40000.			
Poisson's Ratio, n <u>u</u> 0.33			Sł	near		38000.			
Thermal									
Expansion Coeff, <u>a</u>	1.25	5E-5							
Conductivity, <u>k</u>		0.00164352			Mass De <u>n</u> sity Da <u>m</u> ping, 2C/Co		0.		
									Heat Generation Fact

mat_eng_in-lbf-psi-degF-BTU.esp Aluminum

Define Material - ISOTR	OPIC	16.2	X					
ID 2 Title /	luminum 1060 Anneal <u>C</u> ole	or 104 Palette	Layer 1 Type					
General Function Refer	ences Nonlinear Ply/Bond F	Failure Creep Electri	cal/Optical Phase					
Stiffness								
Youngs Modulus, <u>E</u>	1000000.	Tension	4000.					
Shear Modulus, <u>G</u>	0.	Compression	0.					
Poisson's Ratio, n <u>u</u>	0.33	Shear	0.					
Thermal								
Expansion Coeff, <u>a</u>	1.31E-5	Mara Daraita	0.00025362					
Conductivity, <u>k</u>	0.002963	Mass De <u>n</u> sity	0					
Specific <u>H</u> eat, Cp	0.	Da <u>m</u> ping, 2C/Co	0.					
Heat Generation Factor	or 0.	Reference Temp	/5.					
f _{xy} Loa <u>d</u>	<u>S</u> ave	Сору	<u>Q</u> K Cancel					

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:







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

Checklist

Material Specification for Thermal-Stress Simulation

Transient Heat Transfer

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

Define Material - ISOTR	OPIC		X	No	onlinear Control Options	X
ID 1 Ittle A	luminum <u>C</u> olor	104 Palette	Layer 1 Type		Step Control	
General Function Refere	ences Nonlinear Ply/Bond Fa	ilure Creep Electri	ical/Optical Phase		Adaptive OCO	nstant Interval
Stiffness		Limit Stress			Number of Time Steps	600
Youngs Modulus, <u>E</u>	1000000.	Tension	0.		Taitial Time Incompany	1
Shear Modulus, <u>G</u>	0.	Compression	0.		Initial Time Increment	1.
Poisson's Ratio, n <u>u</u>	0.33	Shear	0.		O <u>u</u> tput Step Interval	20
Thermal					Max Iterations per Step	10
Expansion Coeff, <u>a</u>	1.3E-5	Maga Dansiba	0.000253			Estimate
Conductivity, k	21.2	Mass De <u>n</u> sity	0		Convergence Teleronees	
Specific <u>H</u> eat, Cp	772000.	Damping, 2C/Co			Convergence Tolerances	
Heat Generation Facto	or O.	Reference Temp	0.		Temperature	0.001
					Load	0.001
					<u>W</u> ork	1.E-7
					Stiffness Updates	
					Method 0	Default 👻
					Include Differential Stiffne	ss in Damping
f _{xy} Loa <u>d</u>	Save Co	opy	OK Cancel	(Prev Next	<u>O</u> K Cancel

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 thermalstress 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.

API Programming	Femap with NX Nastran - [Transient Thermal Examp	ple Model of Torch Heating - Start.modfem : Predictive Enginee	ring)	0 ×_
	File Tools Geometry Connect Model Mesh	Modify List Delete Group View Window Help Pr	edictive Engineering	_ 5 ×
		NA 2 - SVT ST X E 4 8 0 0	000+XYZ44IM6-E40-W-M. MARRAN = * 000 + XYZ 4 000	
1 API Written by Predictive Engineering 2014, Rev-0 (Tested on Femap v11.1.1)	NX Nastran Analysis Monitor 🛛 🕈 🗙	Model Info # ×	HDR 0813-01 Thrust Collar An, Transient Thermal Example Model2 : Predictive Engineeri	4 Þ 🗙
2 3 'This API creates thermal loads from select output sets and sets up thermal stress	Status : Complete	23 ● 示 (+ 示 タ・		1000
4 'analysis sets corresponding to the loads	Job Name : Thermal Stress: Load from Transie	 a ◆ 10028_memail coad nonin results (cC 28) a ◆ 10029_Thermal Load from Results (LC 29) 		1000.
6 'Please feel free to suggest any improvements to www.PredictiveEngineering.com ≡	Model Name : D:\PredictiveEngineering\Web Sem Elapsed Time : 00:00:10	 10030.Thermal Load from Results (LC 30) 10031.Thermal Load from Results (LC 31) 	000	907.
7 '	log 💿 Sparse Matrix Solver	E Constraints		
9 '- 04/6/14 Rev-0 Nate Anderson <nathan.anderson@predictiveengineering.com< th=""><th>0 f04 0 f06</th><th>A:= Constraint Definitions Other Constraints</th><th></th><th>814.</th></nathan.anderson@predictiveengineering.com<>	0 f04 0 f06	A:= Constraint Definitions Other Constraints		814.
10 ' * initial release	Bytes per word: 4	G fay Functions		721
12 Sub Main()	Reacte shell cad: "c:/fesapv112/nast	Total Surfaces		721.
13 'Attach to femap 14 Dim Ann Ac feman model	NA Mastran started Ved Apr 1 12:15: 12:15:30 Beginning Analysis	3 Analyses 3 1.NX Nastran Trans Heat Analysis Set		628.
15 Set App = GetObject(, "femap.model")	12:15:30 NX NASTRAN Authorization 1 12:15:30 Hodel: Intel(R) Core(TM) i	Supervised Stress: Load from Transient Result		A see
16 Dim rc As Integer 17	12:15:30 Bachine: Intel64 Faally 6 12:15:30 05: Windows 7 12:15:30 Version: Service Pack 1 12:15:20 License File(a): 2000007cc	Interfection of the second secon		535.
18 'Dims to hold output ids and temp values	12:15:30 The FEMAP dll Search Strir 12:15:30 Found the FEMAP authorizat	All Resul		442.
20 Dim addIDS As Variant	12:15:30 Loading FEMAP Authorizatic 12:15:30 NX NASTRAN Authorization 1	al L.Case Copy		
21 Dim Temps As Variant	12:15:30 12:15:30 License for adule NX Mast	al 3.Case 🔛 List		349.
23 ' node set for all model nodes	12 15:31 Analysis started 12 15:31 Geometry access/verificati 12 15:31 Geometry access/verificati	al 5.Case Delete		256
24 Dim allSet As femap.Set 25 Set allSet = App feSet	12:15:32 Finite element model gener 12:15:32 Finite element model gener 12:15:32 Finite element model gener	al 6.Case Remove Study		230.1
26 rc = allSetAddAll(FT_NODE)	12 15 32 Application of Loads and E 12 15 34 Application of Loads and E 12 15 35 Solution of the system eqt	el 8.Case Animate Study		163.
27 28 ' node set for adding load values	12 15 35 Solution of the system equ 12 15 42 Linear static analysis cos 12 15 42 NSEXIT: EXIT(0)	In.Cal Envelope Study Max Value Min Value Min Value	Set: Case 31 Time 600.	70.04
29 Dim ndSet As femap.Set	Real: 11.678 seconds (0.00:11. User: 10.732 seconds (0.00:10.	Max Absol	Inte Value Contour: Temperature	/0.01
30 Set ndSet = App.feSet 31	Sys: 0.421 seconds (0.00.00. NX Nastran finished Ved Apr 1 12:15 +	1 Model Info Meshing PostProcessin Create in D	Database Output Set 59 is Complete.	
32 'set for results nodes.	•	Entity Info # ×	Beginning Cleanup of Output Set 60	
33 Dim resSet As femap.Set	Update Monitor Max Lines 50000	Node 55962	Beginning Cleanup of Output Set 61	
34 Set resSet = App.feSet	Auto Load Results	Coord(0) = -13.57022, 13.12899, 35.0479 DefCS = 0 OutCS = 0	Cleanup of Output Set 61 is Complete.	
35	Kill Job Clean Queue Load Results	Temperature = 70.15349	Beginning Cleanup of Output Set 62 Cleanup of Output Set 62 is Complete	
36 Dim n As Long	·		X 4 III	
37	welope selected Analysis Study to find the Maximur	m Value	Prop: 1 Lo	d: 1 Con: 1 Grp: 0 Out: 31
V 201 'load out object	🚱 💿 📋 🛜 💽	💁 🎐 🖵 🐬 💌 🔜	n 🔅 n 😌 🦉 🖓 🕒 🗎 👘 🖉 🖉 👘 👘 🖉	8 📾 🔫 12:16 PM 4/1/2015



Thank You

Predictive Engineering is located in Portland, OR





PredictiveEngineering.com