

Webinar: Recovering Stress Data with Simcenter Nastran

Andy Mouron, ATA Engineering November 13th, 2019

13290 Evening Creek Drive S, Suite 250, San Diego CA 92128

(858) 480-2000



in ata-engineering

@ATAEngineering

ATA Provides High-Value Engineering Services With Expertise in Design, Analysis and Test

ATA Engineering helps to overcome product design challenges across a range of industries



ATA is a Value-Added Reseller for Siemens PLM Software

ATA offers training, free resources, and hotline support for a variety of Siemens products.



- > Siemens product lines we support include:
 - ➢ STAR-CCM+
 - ≻ Femap
 - Simcenter Nastran (formerly NX Nastran)
 - ➢ Simcenter 3D
 - ➢ NX CAD & CAM
 - ➤ Teamcenter
 - ➤ Solid Edge
- > Contact the hotline at 877-ATA-4CAE or

http://ata-plmsoftware.com/support

- > Developer of the official Simcenter Nastran training materials
- Preferred North American provider of Simcenter Nastran training
- Recognized as Smart Expert Partner with validated expertise in

Femap and STAR-CCM+





Visit Our Website for Product Information and Free Resources

www.ata-plmsoftware.com

TAC IS SOUTOR MATNER		a and an and an and		inter and	amente altre lan catar alt	-
	Support Allk's learn of experts provide com- secret, Fill in the Torm Decay and	preharolog Sectorical aupport for all of the Siemann software one of our technical hearn will be in touch shortly.	Resources are pa	ssword p	rotected, Get your pas	sword
	Attentionery you can cart as at 817	-343-4231. Heart of operation (84.m, to figure, Eastern)	Whitepopers		Macros	
	Nut Name*	, Point Stault*	Champing Works in an Assembly in No.	epr.	Nerverber Bislops II III	******
	Company ^a .	Hume Number*	Animality Lovel Part Design Using Helpart Hole	ing sing PM	Child Denert Quality	MADRO
	Software Product*		Living Synchronous moduling to Hampscate Solo	Badayin Id.	Norsenber Labels in Htt	HALTRO
	[=	. •	Reart Post-Processing with Cross Section Views	ALL POP	SIE	MENS
	Please describe the loss pre-se	«Naulog	Bala Sile Deche w H	R#		
			Presentations		Other Resources	
	Science		Design, Analysis, and Manufacturing Successively	III PM	Contemporal technics	
	A Contraction		Analysis briver delegis optimization of a House	d hundeer. MMC	Coleman Hideur	
			Residence or this Product and Hanufacturing with for Dissign and Anatom	realized (PMP)		
			Eberganian of Composite Headeling Techniques	FOF	Co-Devariat testinant	
17 ATD, Englwooring, All rights NETS: All ALL INTERNET	enserved, Web Design by Dyrapatha Mannak Alexyster Mann Lawrest Man	ы	Training Videos		Tutorials	12





Webinar: Recovering Stress Data with Simcenter Nastran

Andy Mouron, ATA Engineering November 13th, 2019

13290 Evening Creek Drive S, Suite 250, San Diego CA 92128

(858) 480-2000



in ata-engineering

@ATAEngineering

Webinar Overview: Recovering Str<u>ess Data with Sim</u>center Nastran

Purpose:

Explain the difference between nodal and elemental stress results

Motivation:

- ➤A basic understanding of how stress is computed in FEA is important when reporting stress results
- Postprocessors allow users to flip between nodal and elemental stress results seamlessly
- Understanding how the solver reports stress output is critical to knowing how to interpret stress results in the postprocessor



How is Stress Computed for an Element or Node?

Gauss/Integration Points

- In Finite Element Theory, stress results are computed at one or more points within the element (called Gauss or integration points).
- The number of Gauss points is determined by the type, shape, and order (linear vs. parabolic) of the element.
- The stresses obtained at these Gauss points inside the element are then extrapolated to the nodes when reporting nodal stress results.



<u>Note:</u> An 8-node brick element has 8 Gauss points, while a 20 node brick has 27 Gauss points.



CENTER vs. CORNER



*From Nastran Quick Reference Guide (QRG); "greyed out" entries are the Nastran defaults

Users are encouraged to request CORNER as the stress output location to allow for proper viewing of both nodal and elemental stress results.

Viewing nodal stress results using the CENTER stress output location request can lead to inaccurate stress predictions, as is demonstrated in the following example.



Stress Output Location Example

Viewing Nodal Stresses With CENTER vs. CORNER Results



Theoretical K = 4.32 (from Peterson 3rd ed., Chart 4.1) Theoretical Max Stress = 66.667 psi * 4.32 = 288.8 psi

The STRESS = CENTER output location request underpredicts the peak stress because the stress at the center of the elements is less than the extrapolated stresses at the edge of the hole.



Note: Both plots use nodal stresses.



Stress Output Request

Simcenter 3D vs. FEMAP Defaults

> The default for **Simcenter 3D** is **CENTER** > The default for **FEMAP** is **CORNER**

Modeling Object	
Construction of the second sec	
same	
abel 7	
roperties	
Description	10
ACCIENT.	
	Preview
	Enable All
	Disable All
inlution Applicability View All Solutions	
Glue Result 🔹 🕑 Enable STRESS Request	
Grid Point Force Sorting	Default
Ematic Example	PLOT -
Output Transformat Data Format	REAL -
Modal Contribution Vield Criterion	VONMESES -
Modal Effective Mas Location	CENTER
MPC Forces Composite Solid Ply Output	Default -
Plastic Strain Random Output Medium	Default -
Progressive Failure R Random Functions	Defauit -
Shell Thickness Entity Selection	^
SPC Forces Entity	ALL
Strain	Theory of
Strain Energy	
Street	
- Thermal Strain	
Temperature	
velocity +	
* Latter *	

NASTRAN Output Req	uests				23
Nodal					
Displacement	0Full Model	•	Velocity	0Full Model	-
Applied Load	0Full Model	-	Acceleration	0Full Model	-
Constraint Force	0Full Model	•	Kinetic Energy	0Full Model	-
Equation Force	0Full Model	-	Temperature	0Full Model	-
Force Balance	0Full Model	-			
Elemental					
Force	0Full Model	-	Heat Flux	0Full Model	-
Stress	0Full Model	•	Enthalpy	0Full Model	-
Total Strain	0Full Model	-	Enthalpy Rate	0Full Model	-
Elastic Strain	0Full Model	-	Temperature	0Full Model	-
Thermal Strain	0Full Model	-	Kinetic Energy	0Full Model	-
Isiber	Curvature		Energy Loss	0Full Model	-
Strain Energy	0Full Model	-	Fluid Pressure	0Full Model	-
Customization					
Element Corner Res	sults	F	Results Destination	Prev	/
Output Modes (a,b,c	THRU d)		2PostProcess Only	•	
] [Echo Model	0	к
Magnitude/Phase	Real/Imaginary			Can	icel
Relative Enforced N	Notion Results				



Changing the Default Stress Data Recovery Location in Simcenter 3D

- > The following steps will change the default setting for the STRESS data recovery location from CENTER to CORNER:
 - 1) Go to this folder, or the equivalent path on your installation and NX version: C:\Siemens\NX1851\SIMULATION\templates
 - 2) Open the file "SimNxNastranEnglish.sim" or "SimNxNastranMetric.sim" depending on the unit system of interest (you may not have write-access to either of these files in your installation directory so you might have to save a copy of the file to a temporary location such as your desktop)
 - 3) With the template file you will want to:
 - a) Create a new solution
 - b) Edit the Case Control Output Requests to your preferences
 - c) Delete the solution (your structural output requests will be saved as a modeling object, and is not deleted)
 - 4) Save the templates file and, if necessary, move it back to the templates folder in your NX installation directory
- These presets will take effect for new .sim files you create, but not existing ones.



Different Ways to Report Stress

- Nodal Peak: reports the peak nodal stress from each contributing element
- Nodal Average: reports the average of all elemental input for each node
- > Element Centroidal: reports the average of the Gauss point stresses

Stress Result Type	Calculation
Nodal Peak	max(w, x, y, z)
Nodal Average	(w + x + y + z)/4
Element Centroidal*	-

*No averaging between elements occurs with element centroidal output results; the average of each element's gauss point stresses is reported.



- A, B, C, D = average of each element's gauss point stresses (i.e., the element centroidal stress for each element)
- w, x, y, z = each element's extrapolated stress at the shared center node

The stress at this interior node has four values associated with it for a given stress tensor.



Contour Settings to Use in FEMAP

Nodal Peak

View 1 Untitled		
Contour Type		Contour Fill Mode
Match Output	Nodal Elemental	 Continuous Level Colors
Contour Group		
C Active O None / Vis	ible Group(s) 🛛 🔘 Select	
Data Selection	Data Conversion	Elemental Contour Discontinuities
🔘 All Data / Full Model	Average	No Averaging
🕐 Visible Group(s)	Max Value	Property Layer
Ontour Group	C Min Value	[☑] Material □ Color
🖉 Use Corner Data		Angle Between 20.
		OK Cancel

Element Centroidal

liew 1 Untitled		
Contour Type		Contour Fill Mode
Match Output	Nodal	Continuous
O Materi Output	Elemental	Oliver Colors
Contour Group		
Active None / Visi	ble Group(s) 💿 Select	
Data Selection	Data Conversion	Elemental Contour Discontinuities
🔘 All Data / Full Model	Average	Vo Averaging
Visible Group(s)	🔘 Max Value	Property Layer
Ontour Group	🔘 Min Value	☑ Material □ Color
Illes Corpor Data		Angle Between 20
use corrier Data		[*] Angle between

Nodal Average

/iew 1 Untitled		
Contour Type		Contour Fill Mode
Match Output	Nodal	Continuous
O Match Output	🔘 Elemental	Level Colors
Contour Group		
C Active O None / Visi	ible Group(s) 🛛 🔘 Select	
Data Selection	Data Conversion	Elemental Contour Discontinuities
🔿 All Data / Full Model	 Average 	No Averaging
Visible Group(s)	🔘 Max Value	Property Layer
O Contour Group	Min Value	Material Color
Use Corner Data		V Angle between 20.

<u>Note</u>: Nodal contouring simply averages all values at the nodes and cannot account for any discontinuities in material or geometry (hence the Element Contour Discontinuities options are not available).



Contour Settings to Use in FEMAP

Post-Processing Toolbox

The same contour settings applied using the dialog boxes on the previous slide can also be specified in the Contour "tool" in FEMAP's PostProcessing Toolbox.

Nodal Peak

Deform		
Contour		-
Style	Contour	
Results		
Output Set	3.10Gs_pZ	0
Output Vector	60031Solid Van Mises Stress	0
Additional Vector(s)		
Options		
Data Conversion	Maximum Value	
Туре	Nodal	-
Show On Groups	Full Model / Visible Groups	一面
Show As	Filled	
Levels		-
Legend	(W)	
Freebody		

Nodal Average

P	ostProcessing Toolbox		4	×
ļ	●· · · · · · · · · · · · · · · · · · ·	0 - 0 3		
	Deform			
	Contour			
	Style	Contour		
	Results			
	Output Set	3.10Gs_pZ	+0	•
	Output Vector	60031_Solid Van Mises Stress	+ 0	•
	Additional Vector(s)			
	Options			
	Data Conversion	Average		•
	Туре	Nodal		
l	Show On Groups	Full Model / Visible Groups	1	E
	Show As	Filled		
	Levels			
	Legend	V		
	Emphodu	(Iban)		
	Heebody			

🗄 Model Info 📓 Meshing 📷 PostProcessing

Element Centroidal

Deform		
Contour		-
Style	Contour	1
Results		_
Output Set	3.10Gs_pZ	•
Output Vector	60031Solid Van Mises Stress	+ 0
Additional Vector(s)	1	
Options		
Data Conversion	No Average, Centroid Only	
Data Selection	Contour Group	
 Туре 	Elemental	
Double-Sided Planar		-0.9
Show On Groups	Full Model / Visible Groups	1
Show As	Filled	1.1
Levels		
Legend	(V)	
Freebody		_
1250 (1990) (1990 (1990 (1990 (1990 (1990 (1990 (1990 (1990 (1990		
Contour		
	en en antiere en en antiere en	



Contour Settings to Use in Simcenter 3D

Nodal Avorago

Nodal Peak

Post View			ა ×
Result Display	Deformation	Legend	
Result Selection	n		^
Subcase - Static	Loads 1		Ŧ
Static Step 1			Ŧ
Result Type			^
Stress - Element	-Nodal		-
Von-Mises			•
Result Combina	ation		^
Combine At		Nodes	•
Nodal Combina	ation		^
Maximum			•
Include Visit	ole Elements Onl	y	
Include Midne	odes		
Coordinate Sys	tem		^
Coordinate Syste	m	Absolute R	ectangular 🔻
Units		lbf/in ²	•
Scale		L	1.0000
Absolute Value			
Apply dB Scalin	g		
dB Scaling			^
dB Factor		20	Ŧ
dB Reference	1.0000 kPa	1	
	Reset to D	efaults	
	O	K Apply	Cancel

Post	View			0
Result	Display Def	formation	Legend	
Result	Selection			~
Subci	ne - Static Load	1.0		14
Static	Step 1			
Result	Туре			
Stress	- Element-Nod	lul.		
Von-M	lises			
Result	Combination			^
Combi	ne At		Noder	
Node	d Combination			
Ave	iage			
0.00				
	sage Across			
23	PID Element	t Type		
10	PID 📃 Element MID 🕎 Feature	t Type Angle	45.0000	
10	PID 📄 Elemen MID 🕎 Feature MID 🗐 Feature	t Type Angle	45.0000	
	PID Element MID Feature clude Voible El-	t Type Angle ements Or Jements	45.0000	
	PID C Element MID V Feature Include Voible Electude Internal E Include Midmodes	t Type « Angle ements Or Jements	45.0000) Ny	
E In E In E Incl Coord	PID E Element MID V Feature Icluste Voible El- Icluste Internal E Isole Midmodes	t Type « Angle ements Or Jernents	45.0000) Ny	
In a coord	PID III Element MID III Feature Iclude Visible El- Iclude Internal B Iude Midhodes Inate System nate System	t Type « Angle ements Or Jements	45.0000) by Absolute Rec	tangular •
E In E In E Incl Coord Coord	PED Classes PED Classes MED (2) Feature rolude Visible El- clude Internal E lude Mishodes inate System nate System	t Type • Angle •ments Or	45.0000) Ny Absolute Rec	*
E In E In E Incl Coord Coord	PID C Element MID IV Feature clude Vaithe El- clude Internal E lude Midhodes inate System nate System	t Type e Angle ementa Or Jements	45.0000) Ny Absolute Rec Bol/In ⁴	tangular •
Coord Coord hits cale	PID I Element MID II Feature clude Vaittle El- clude Internal E lude Midnodes inatte System nate System	t Type e Angle emerta Or Jeroents	45.0000) Ny Absolute Rec Bol/In ⁴	tangular • 1.0000
In Incl Coord Ints cale Abso	PID C Element MID IV Feature clude Vaitale Electude Unitale Electude internal E lude Midnodes intale System nate System lute Value y dB Scaling	t Type « Angle emerta Or Jeroents	45.0000) Ny Absolute Rec Bol/in ⁴	tangular • 1.0000
In I	PID C Element MID IV Feature clude Vaitale Electure clude Midnodes inate System nate System lute Value y dB Scaling Jing	t Type « Angle ementa Or Jeroents	45.0000) Ny Absolute Rec Bol/In ⁴	tangular •
E In In In In In In In In In In	PID C Element MID IV Feature Induke Valitie El- Induke Midmodes Induke System nate System fute Value y dB Scaling ding	t Type e Angle ements Or	45.0000) hy Abiolute Rec Bol/in ⁴	tangular • 1.0000
In I	PID C Element MID IV Feature Induke Valide Element Induke Midnodes Induke System Nate System Nate System Nate Value y dB Scaling ding	t Type e Angle ements Or Jernents	45.0000) hy Absolute Rec Bol/In ⁴	tangular • 1.0000
India Coord India	PID III Element MID III Feature Induke Valitie Ele Induke Midmodes Induke System Nate System Nate System Nate Value y dB Scaling ding	t Type e Angle ementa Or Jersents	45.0000) hy Absolute Rec Befin ⁴	tangular • 1.0000

<u>Note</u>: "Average Across" settings will be mesh dependent; uncheck the box to turn off averaging across feature.

Element Centroidal

Post \	/iew				<u>ა</u> ა	
Result	Display	Deformation	Legend			
Result Selection						
Subcase - Static Loads 1 👻						
Static Step 1					Ψ.	
Result	Туре				۸	
Stress	- Element-	Nodal			•	
Von-N	lises				•	
Result	Combina	tion			^	
Combi	ne At		Elemer	nts	•	
Eleme	ent Criterio	n			^	
Cent	roid				•	
Incl	ude Midno	odes				
Coordi	nate Syst	tem			^	
Coordinate System			Absolu	Absolute Rectangular 🔻		
Units			lbf/ir	1 ²	•	
Scale				1.00	000	
Absol	ute Value					
Apply	dB Scalin	9				
dB 2ca	ling				^	
dB Fact	or		20		*	
dB Refe	rence	1.0000 kPa				
		Reset to D	efaults			
				nnly Can	cel	
				Can		

<u>Note</u>: Could also use "Stress-Elemental" result type with Result Combination set to "None" for element centroidal results.



When Would I Use Nodal vs. Elemental <u>Stress Results?</u>

➤ General rules:

- I) Nodal peak is always more conservative than nodal average.
- 2) Element centroidal is less conservative but may be required if you have a specific allowable correlated to this stress type (e.g., element-size correlated allowables).
- When assessing fatigue, nodal results are often used. For strength assessment, nodal or elemental results could be used depending on the required fidelity of the analysis.



<u>Side Note 1</u>: Comparing Different Stress Result Types is One Means of Evaluating Convergence



- The difference between nodal peak and nodal average stress decreases as the mesh size is reduced, the result of the elements' Gauss points moving closer to the nodes.
- As a result, comparing nodal peak with nodal averaged results can be used as a convergence criterion.



Von Mises Stress at Middle of Clamped Edge

<u>Side Note 2</u>: Nodal Average Stress Calculation in Simcenter 3D vs. FEMAP

- The nodal average stress calculation is handled differently in Simcenter 3D and FEMAP:
 - Simcenter 3D: averages the component stresses at each node before computing the von Mises stress at the node
 - FEMAP: first computes von Mises stress from the component stresses then averages those von Mises stress values at each node
- The order of operations differs which is important to note if trying to compare Simcenter 3D nodal average stress results with FEMAP nodal average stress results



Questions?

Submit questions in the chat or unmute yourself now

Zoom Application



Web Interface

Contact Us



13290 Evening Creek Drive S Suite 250, San Diego, CA 92128

(858) 480-2000

plm_sales@ata-e.com

www.ata-e.com www.ata-plmsoftware.com

@ATAEngineering

ata-engineering

