FEA Model Checking Procedures

Prepared by:
Lorraine Guerin, Laura Hoffman,
Kurt Knutson, Sam Dyas

ATA Engineering, Inc.

Date:
October 2019

13290 Evening Creek Drive S, San Diego CA 92128

(858) 480-2000
www.ata-e.com
ata-engineering
@ATAEngineering

Outline

1. Pre-Meshing Philosophy
2. General Best Practice FE Model Checks
   Model Preparation Checks
3. Structural Analysis Model Checks
4. Thermal Analysis Model Checks
5. Post-Processor Checks
6. Common Solutions to Common Problems
7. Sample Checklists

Pre-Meshing Philosophy: Asking Important Questions Before You Begin Meshing Will Save Time

1. Is the CAD current and consistent with analysis objectives?
   - Confirm units: Take measurements of the CAD to verify
2. What are the simulation goals? (loads development, stress analysis, etc.)
   - This will help determine which regions are critical, what mesh type to use, etc.
3. What is the required model fidelity?
   - Stress analysis typically needs a finer mesh, especially in areas that see high loads and areas with fillets.
     - Parabolic Tet Fillet Rule: Three or more elements across fillet
   - Dynamic response used to capture modes and acceleration response are not as detailed as stress models.
     - More imperative to get stiffness and connections correct
4. What type of mesh is to be used (e.g., beam, shell, solid)?
   - Are triangular shells acceptable?
   - Will beams and shells require offsets?
5. How will you appropriately simplify the given geometry for modeling?
   - What geometry details are important? Which will cause meshing problems?
6. What do you need to do to clean up the geometry for meshing?
   - Remove sliver surfaces or unimportant features (small holes, etc.).
   - Make sure surfaces/volumes are partitioned based upon changes in properties or other important features.

Why Are All Those Questions Important?

- Avoid doing the "right" analysis with the wrong model or design.
- Don't lose the objectives behind the models, e.g., a model too big to solve or missing details in important areas.
- Share and summarize peculiar design details found in modeling processes.
- Identify and capture significant analysis assumptions as they occur.
- Example: Wrong CAD for analysis!
  - Design documentation error with geometric features of inconsistent length noticed by an analyst when producing a mesh. Design had used "reduced" dimension drawing which did not call out crucial lengths on drawing.

Model Translation Needs Systematic Checking

- Translation refers to converting CAD or FEM data from one tool to another.
- Translation is "artificially" intelligent.
- Many translation formats convert data in one form to another form (if possible).
- When converting one model for use with another code:
  - Always document the original model characteristics and results, especially for model checks or baseline results.
  - Prior to subsequent modification, perform basic simple checks and document the results in the new code.
  - Recognize that many CAD neutral formats mangle names, can skip geometry in certain forms, and do not support materials (even) if they were in the original CAD authoring tools.

Pre-Processor Model Checks Outline

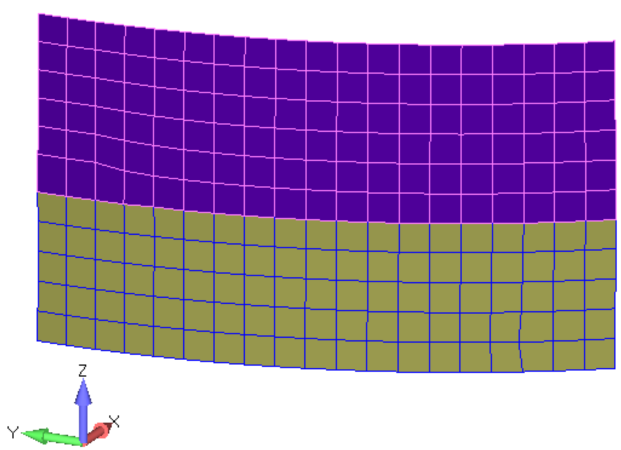
1. Rules of thumb
2. Free edges
3. Coincident nodes and elements
4. Visualization of orientation
    1. Shell element normal
    2. Material direction
    3. Bar/beam directions
    4. Element offsets
5. Element quality
6. Material properties
7. Mass
8. Summarize boundary conditions and loads
9. Self-document your model (numbering!)

[1] Meshing Rules of Thumb

- Use coordinate systems for definitions:
  - CBUSH orientation
  - Orthotropic material direction
  - Nodal output in specific degrees of freedom (DOF)
  - Load application
- Separate components by using different property IDs, even if they all use the same material ID.
- Generally speaking, choose element types that are appropriate and connect them reasonably.
- There are some common element connections that cause problems. For example you should not:
  - Connect a bar to the drilling DOFof a plate.
  - Connect a plate to an edge on a solid. Solids do not have 6 DOF at their nodes and the plate will hinge.
  - Use springs that are not quite coincident (as seen by the solver). This can cause grounding.
  - Use "stiff" springs for loads that are modeled as too stiff. This can cause grounding.
  - Use rigid elements and constraints when a simple elastic connection can be modeled instead.
- If the part being meshed will be a small part of a larger model, import the full model with the newly meshed component to check connections and positioning before running any analysis

[2] Free Edges

- The element free-edge check is a sanity check performed in the pre-processor to confirm there are no free (unconnected) element edges. A free element edge is an edge that is referenced by only one element.
- Free element edges would suggest that there are elements that are not connected. The element free-edge check is essentially just a visual, spot check.



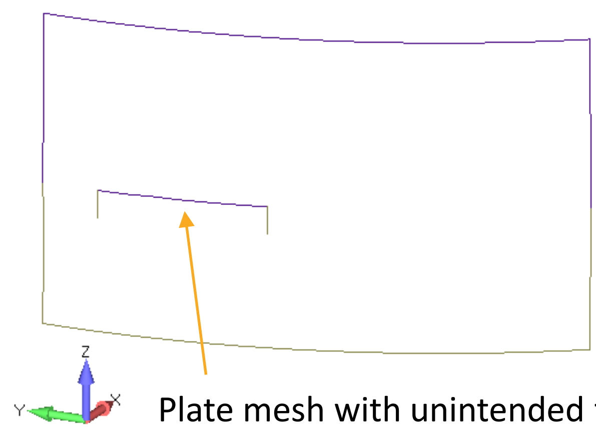
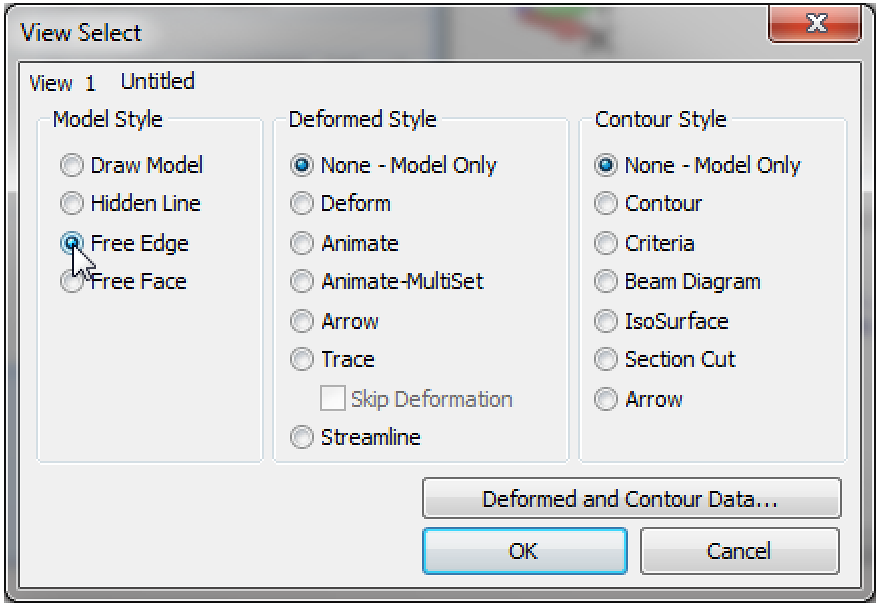


Plate mesh with unintended free edge shown

[2] Free Edges - Femap

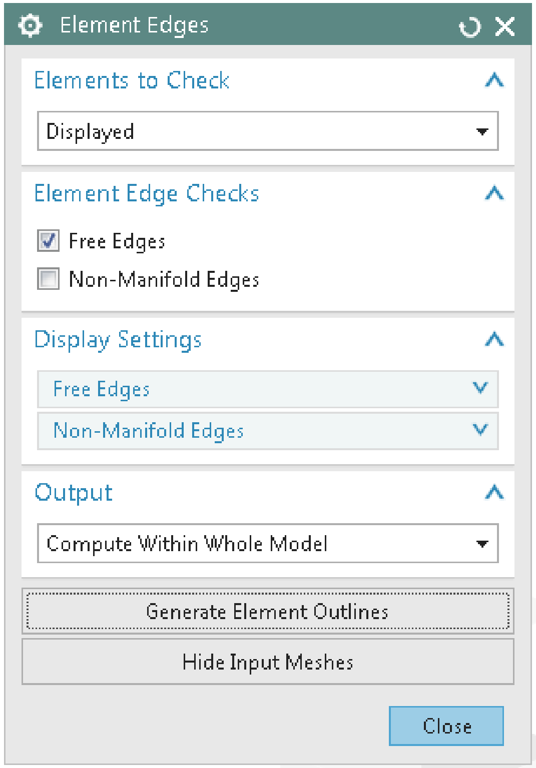
- On the Menu bar under View, choose “Select”. Alternatively press F5. In the dialogue box choose “Free Edge” under Model Style. View the model with these settings to visually inspect for free edges. Reset to “Hidden Line” to view the model again under default settings.



ATA ENGINEERING, INC.

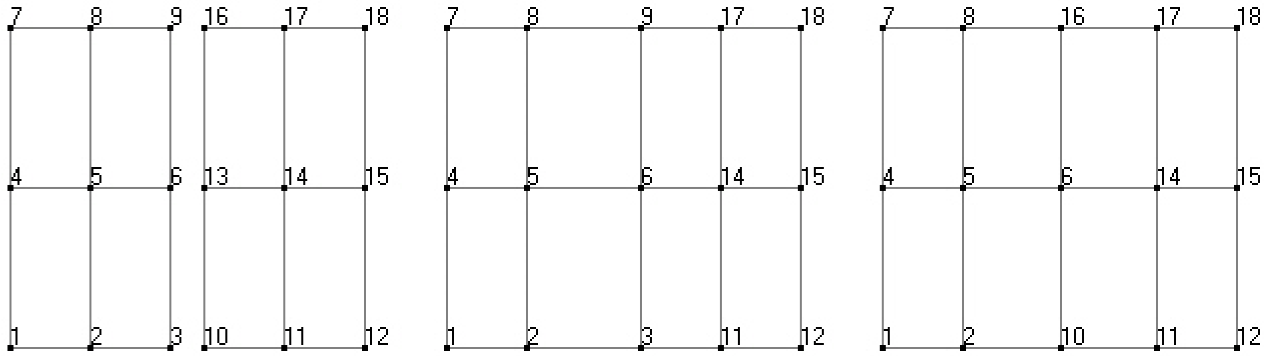
[2] Free Edges - NX

- Setting Up an Element Free-Edge Check in NX:
  - In the FEM, on the Home ribbon, click on More in the "Checks and Information" section and choose Element Edges under "Checks".
  - In the Element Edges dialog box click on the pull-down under Elements to Check and choose Displayed (this assumes that your full model is displayed in the graphics window).
  - Make sure Free Edges is checked under "Element Edge Checks" and that Compute Within Whole Model has been chosen under "Output". Click on "Generate Element Outlines" at the bottom of the Element Edges dialog box.
- Interpreting Element Free-Edge Results:
  - The element free-edge check is trivial in NX. NX will print, in the message area at the top or bottom of your screen, whether or not free edges have been found. If free edges have been found, it will highlight the element edges in the graphics window.
  - If "No free edges found" is printed in the message area, the model has passed the element free-edge check.



[3] Coincident Nodes and Elements

- The coincident node and element check is another sanity check performed in the pre-processor to confirm that all nodes are merged and no unintended duplicate nor overlapping elements exist.
- Coincident nodes often indicate locations where a mesh mating condition was not applied or became deactivated.
- There could be instances where coincident nodes would be desired for a particular mesh, so the results of the coincident node check should be reviewed carefully so that nodes that are not intended to be merged do not get merged as part of the check.
  - Example: Zero-length spring with coincident nodes
- This check can also be used to check for unconnected nodes that are not attached to any elements.



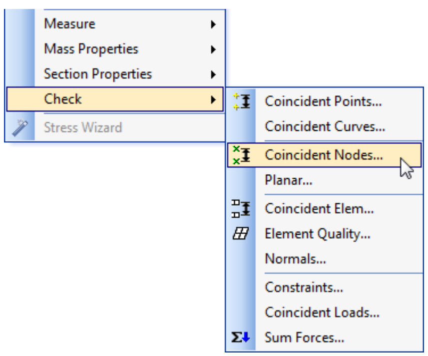
Original Model

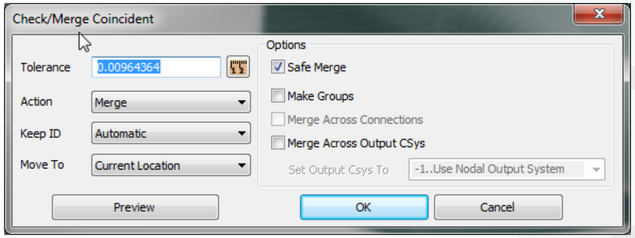
Keep ID - Lower
Move To - Higher

Keep ID - Select Node(s) to Keep (nodes 16, 6, & 10)
Move To - Midpoint

[3] Coincident Nodes and Elements - Femap

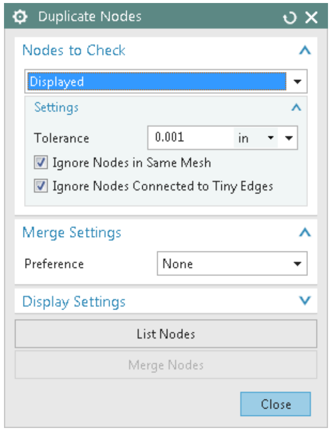
- In the Menu bar, under Tools > Check > Coincident Nodes or Coincident Elem. Select the nodes or elements of interest to include in the check. Femap determines the default node merge tolerance based on overall model size. Set the tolerance to an appropriate value based on the dimensions of the model and mesh. Choose the options for Action, Keep ID, and Move To as appropriate.
  - Safe Merge will not merge mid-side nodes to corner nodes.
  - Make Groups will make groups of nodes to merge
    - When *Action* is set to *Merge* or *Merge and List*, a single group of "Kept" nodes will be created.
    - When *Action* is set to *List* or *Detailed List*, two groups will be created, one for the nodes "To Merge", one for the nodes "To Keep".
  - By default, Femap does not merge nodes with differing output coordinate systems. Select the option "Merge Across Output CSys" to do so anyway.



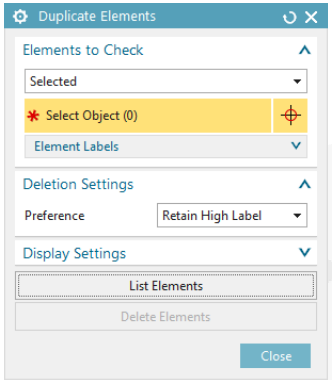


[3] Coincident Nodes and Elements - NX

- Setting Up a Coincident Node Check in NX:
  - In the FEM, on the Home ribbon, click on More in the "Checks and Information" section and choose Duplicate Nodes under "Checks". The Duplicate Nodes dialog box will appear.
  - The default search distance tolerance is 0.001 in, but an appropriate value should be used based on the dimensions of the model and mesh. You will most likely want to check the Ignore Nodes is Same Mesh and Ignore Nodes Connected to Tiny Edges boxes.
  - click on the pull-down under Nodes to Check and choose Displayed (this assumes that your full model is displayed in the graphics window). After Displayed is selected, NX immediately begins the coincident node check.
  - The Coincident Element check is in the same location. Select by element or mesh.

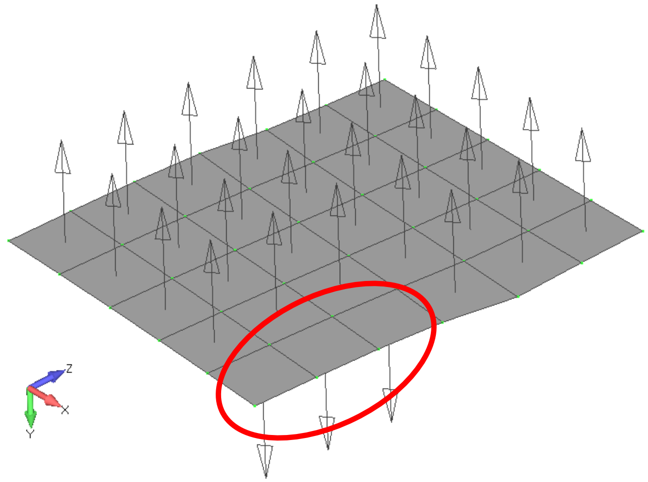


- Interpreting Results:
  - NX will print in the message area at the top or bottom of your screen whether duplicate nodes have been found. If duplicate nodes have been found, NX will highlight these nodes in the graphics window and list the number of nodes it intends to merge in the message area. If the highlighted nodes need to be merged, click Merge Nodes in the Duplicate Nodes dialog box.
  - If "No duplicate nodes found" is printed in the message area, the model has passed the coincident node check.
  - For coincident elements, clicking List Elements will print to the message area. If duplicate elements exist, you will have the option to delete them.



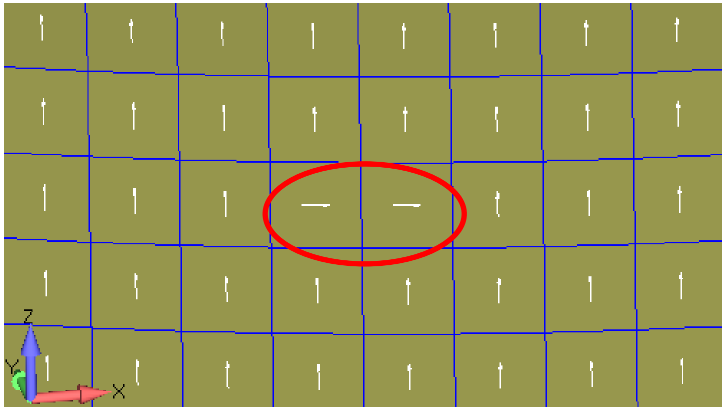
[4] Visualization of Orientation

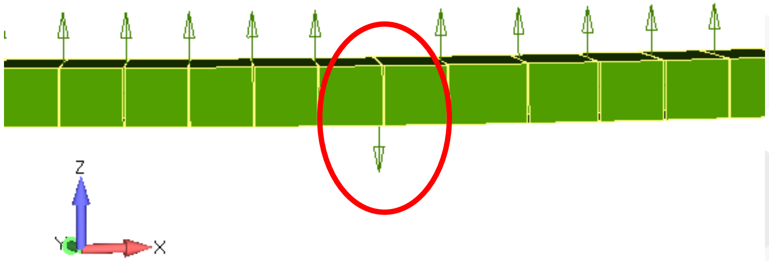
- In the pre-processor, visualize these items to verify their correct use:
  - Shell element normal
    - If normals are not consistent, something like applied pressure loads will not be properly implemented.
  - Element offsets
    - Appropriate offsets will indicate the correct bending stress.



[4] Visualization of Orientation

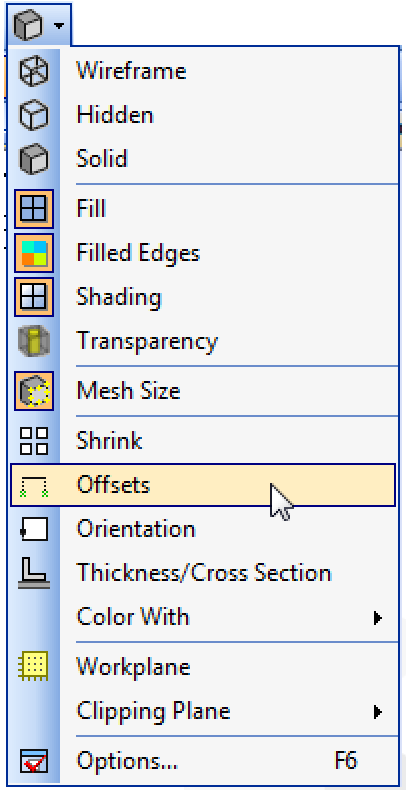
- In the pre-processor, visualize these items to verify their correct use:
  - Material orientation vectors
    - For orthotropic or laminate elements, the results depend heavily on orientation because of non-uniform stiffness.
  - Bar/beam orientation vectors
    - Orientation affects stiffness, which will affect results.
  - Bar/beam end releases





[4] Visualization of Orientation – Femap (1 of 2)

- On the Menu bar under View, choose "Options". Alternatively press F6. Under Labels, Entities and Color category.
  - Shell Element Normal
    - Select Element – Directions as the Option. Under Normal Style, select 1..Normal Vectors, and turn on by clicking the Show Direction box on the upper right.
  - Offsets
    - Select Element – Offsets/Releases as the Option. Turn on by clicking the Show Offsets box on the upper right. Alternatively this can be turned on/off using the View Select toolbar.



[4] Visualization of Orientation – Femap (2 of 2)

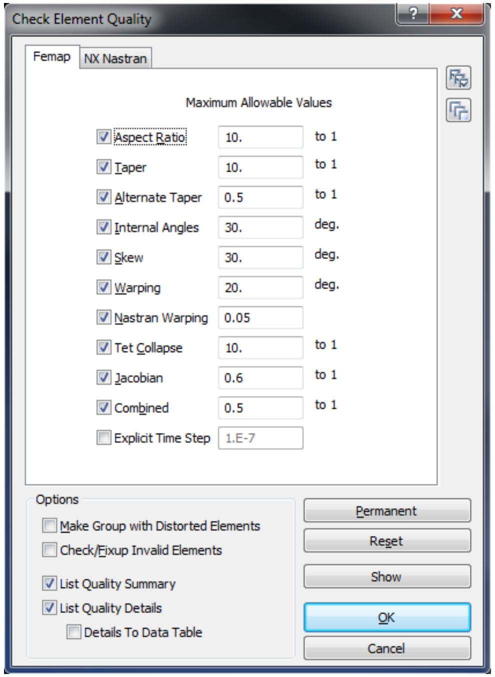
- On the Menu bar under View, choose “Options”. Alternatively press F6. Under Labels, Entities and Color category.
  - Material orientation vectors
    - Select Element – Material Directions as the Option. Turn on by clicking the Show Material Direction box on the upper right.
  - Bar/beam orientation vectors
    - Select Element – Beam Y-Axis as the Option. Turn on by clicking the Show Y Axis box on the upper right.
  - Bar/beam end releases
    - Select Element – Offsets/Releases as the Option. Turn on by clicking the Show Offsets box on the upper right. To show releases, switch Release Labels to 1..Degree of Freedom.

[4] Visualization of Orientation - NX

- Shell Element Normal
  - Right-click the Mesh Collector and select Edit Display. Turn on by setting the Type drop down menu to Vectors under 2D Element Normals.
  - Or, on the Home ribbon, click on More in the "Checks and Information" section and choose 2D Element Normals under "Checks". Select your elements or mesh, and click Display Normals.
- Offsets
  - Edit Display. Turn on by Element Thickness and Offset.
  - Or, right-click on 2D Collectors and select Plot Thickness Contours.
- Material orientation vectors
  - On the Home ribbon, click on More in the "Checks and Information" section and choose Element Material Orientation under "Checks". Select your elements or mesh, and turn on Shell Orientation. Click Display Element Material Orientation.
- Bar/beam orientation vectors and end releases
  - Right-click the Mesh Collector and select Edit Display. Turn on orientation vector and end releases under Display > Beam.

[5] Element Quality

- Specific projects may have requirements on element quality
  - In general, a best practice is to avoid any mesh-quality errors or warnings from the intended solver. If the warnings or errors cannot be avoided, document what and where they are.
- Nastran includes an element check
- Best to use pre-processor tools before running Nastran to review element quality

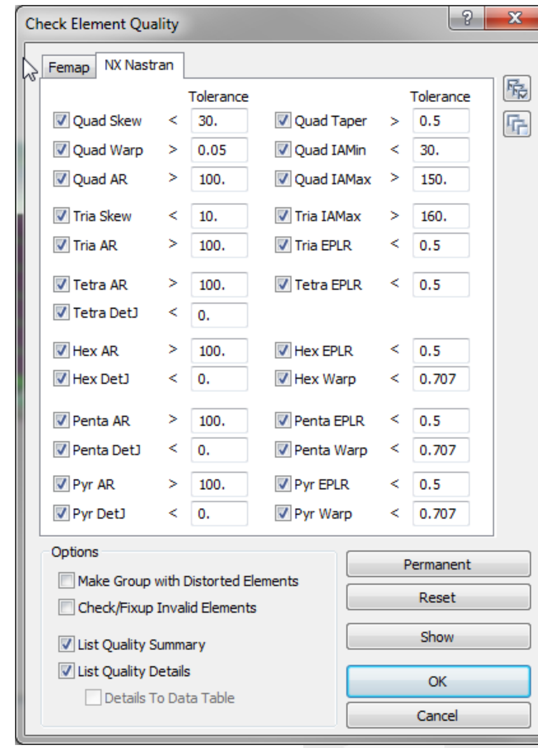


[5] Element Quality

- Use meshing pre-processor to check "automatic" and "semi-automatic" meshing quality as you create meshes
  - Aspect Ratio
  - Taper
  - Element Warping
  - Interior Angles
  - Skew
  - Jacobian
- If you have trouble with element quality reported by the solver or the pre-processor for a few specific elements, make a group of them and visually look at them. This can often give you an idea where the problem area is and give you suggestions for how to improve mesh "seeding" up front.

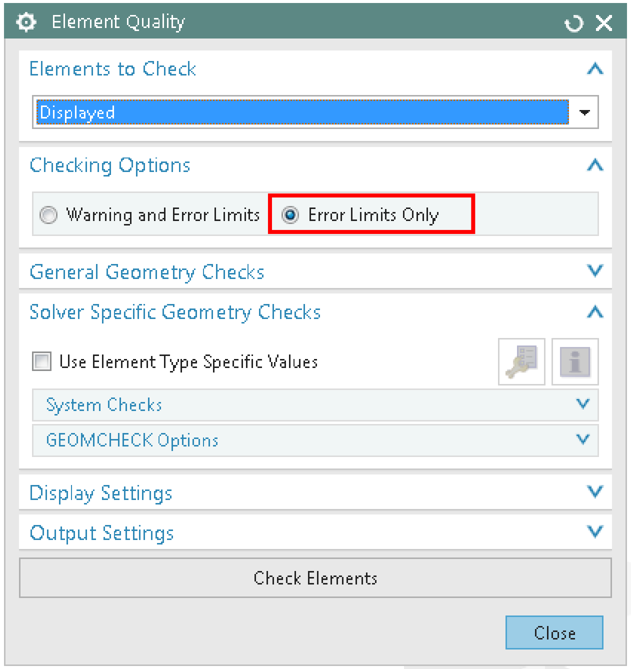
[5] Element Quality – Femap

- In the Menu bar, under Tools > Check > Element Quality. Select the elements of interest. The next dialog box has two tabs, Femap and NX Nastran. If NX Nastran is your solver, use this tab with the default checks.
  - Femap checks may be helpful if you are using a different solver. They have different default values compared to NX Nastran, and some elements may fail under those that otherwise would not under Nastran.



[5] Element Quality – NX

- Setting Up an Element Quality Check in NX:
  - In the FEM, on the Home ribbon, click on More in the "Checks and Information" section and choose Element Quality under "Checks". The Element Quality dialog box will appear.
  - Click on the pull-down under Elements to Check and choose Displayed (this assumes that your full model is displayed in the graphics window).
  - Under Checking Options, select the Error Limits Only radio button.
  - Click "Check Elements" at the bottom of the Element Quality dialog box to begin the element quality check.
- Interpreting Element Quality Results:
  - NX will print in the message area at the top or bottom of your screen whether there are elements that failed the element shape quality check. Failed elements will be appear highlighted in the graphics window.
  - If "0 failed elements" is printed in the message area, the model has passed the element quality check.



[5] General Gotcha's Related to Nodes/Elements Affecting Results or Solve Time

- CQUADs representing singly or doubly curved geometry could result in high thermal stresses.
- Very large RBE3s can cause long solve times for modal solutions.
- Nodes defined along the centerline of a cylindrical coordinate system (which references something other than basic) will have erratic behavior during modal solutions.
- Precision can affect the results of your analysis, especially if it's a large component where the area of interest is physically far from the boundary

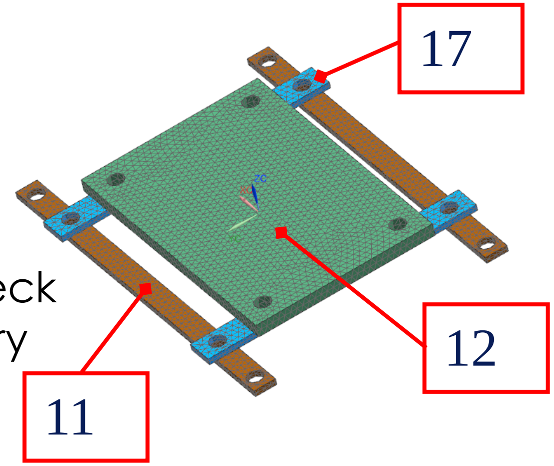
[5] Check the Mesh Geometry, First with Sanity Checks and Again Later, but Not Too Late, with Solver

- Use meshing pre-processor to check meshing quality as you create meshes.
- If creating several disparate meshes, use the solver to check each mesh on its own before connecting them together.
  - Problems will be easier to debug on a smaller scale.
- Mesh Geometry Checks:
  - Element Stretch
  - Element Distortion
  - Free Element Edges
  - Unconnected Nodes
  - Inconsistent Degrees of Freedom
- Example: Complicated Geometry "Abstracted"
  - Meshed in NX, checked in NX Nastran, before details added

[6] Material Properties Check

- This is a simple check intended to ensure that the correct material property values were entered AND that these values were entered using the correct units.
- This is meant to check that no typos occurred when entering material information.
  - Make groups, colors or displays to reflect each material and property, then systematically review each one.
  - Simple tables with images can be a great way to document models and make a systematic review.
- **Tip:** Use a spreadsheet when creating material, property names and numbers, etc. This can be a very useful way to stay organized and consistent and self document new models.

Example: Systematic material check and summary

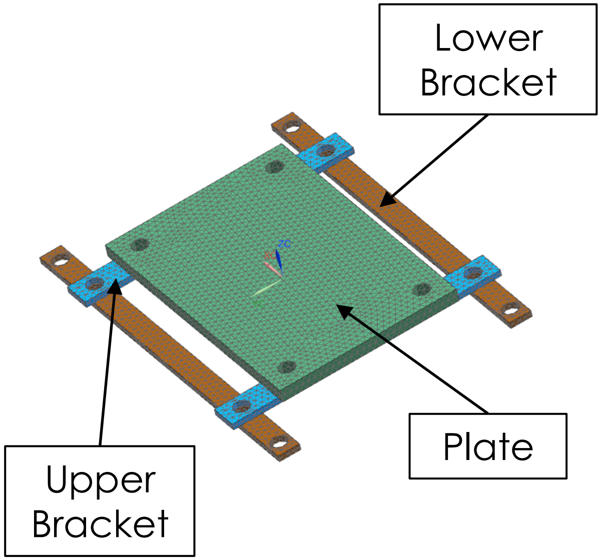


ID	Mat ID -->	Analysis Task: SI Materials 11	12	17
Property	Units	Ti-6AL-4V	Al 6061	A286
E or E1	[Pa]	1.18E+11	6.83E+10	2.01E+11
v or v12		0.31	0.33	0.30
G or G12	[Pa]	4.49E+10	2.57E+10	7.72E+10
Ftu	[Pa]	8.96E+08	2.90E+08	
Fty	[Pa]	7.58E+08	2.41E+08	
mass density	g/cm^3	4.43E+00	2.71E+00	7.95E+00

	Mat ID -->	Analysis Task: (inlbf) Materials 11	12	17
Property	Units	Ti-6AL-4V	Al 6061	A286
E or E1	[psi]	1.71E+07	9.90E+06	2.91E+07
v or v12		0.31	0.33	0.30
G or G12	[psi]	6.51E+06	3.72E+06	1.12E+07
Ftu	[psi]	1.30E+05	4.20E+04	
Fty	[psi]	1.10E+05	3.50E+04	
mass density	lb/in^3	1.60E-01	9.80E-02	2.87E-01
Primary Source		MIL HNDBK5	MIL-HNDBK5	MIL-HNDBK5

[7] Mass Check

- Compare to known design data.
- Minimal check: List the mass of the entire model. Verify against expected mass.
- Best practice: List mass by group or component. Verify against expected mass of each component.
  - In some cases, you may need to adjust the density of the material (for 3D elements) or add Non-Structural Mass (for 1D and 2D elements) to match what is listed in the project Mass Equipment List (MEL)
- Rotational inertias of modeled subsystems should also be included and summarized as available/necessary.
  - **Tip:** Strive to estimate this (I=mr^2).



Component Grouped by 3D/2D/1D Mesh (each green line is a structural mesh)			Unit Mass [lb/ea]	QTY	Subtotal Mass [lb]	Mesh Mass [lb]	Mesh Volume [in^3]	Adjusted Density [lb/in^3]
Plate						0.42051	1.04757	0.40142
PRT012345	PLATE	plate	0.41062	1	0.41062			
-	SHCS 6-32 X 0.375	plate to upper bracket	0.00247	4	0.00989			
Upper Brackets						0.25535	0.40692	0.62750
PRT012346	UPPER BRACKET	brackets, plate to lower brackets	0.12767	2	0.25535			
Lower Brackets						0.30121	0.45178	0.66672
PRT012347	LOWER BRACKET	brackets, upper brackets to mounting surface	0.14566	2	0.29132			
-	SHCS 6-32 X 0.375	lower bracket to upper bracket	0.00247	4	0.00989			

[7] Material, Property, and Mass Check – Femap or NX

- List all materials and properties in the model and double-check them
- Mass check
  - Femap
    - To list the mass of the model, under the Menu bar go to Tools > Mass Properties > Mesh Properties. Select all elements and review the message window contents.
  - NX
    - On the Home ribbon, click on More in the "Checks and Information" section and choose Solid Properties Check under "Checks". Select all elements, turn on all options, and review the message area contents.

[8] Verify And Summarize Boundary Conditions And Loads

- Prepare a visual summary of the applied loads and boundary conditions.
- Anticipate based on the requirements, experience, and engineering analysis what the results are likely to be. Perform simple hand calculations.
- Verify that all the units are correct, labeled, and consistent throughout the analysis.

[9] Self-Documenting Model: Are the Names and ID Labels in New Models or Edited Models Useful?

- Verify that the model uses reasonable numbering and useful labels to describe all properties, materials, coordinate systems, etc.
  - Even if the model is entirely native to the pre/post processor, this is good practice for model organization.
  - Are there requirements on model numbering so submodels can integrate into system models?
- For example a property name like ep_0p5mm_thk is much better than thin_shell_100.
- For best practice on any new model, don't ever use ID number lower than 10.
  - Except for nodes and elements
  - Or unless required by a project or a solver
- For modified input decks, make meaningful comments and label units.
  - Include your name, date, and company name
  - Model description and log of changes
  - May go to a partner, customer, vendor, so need to be able to trace back

ATA ENGINEERING, INC.

Final Notes

- Update and re-perform checks at key stages
- Verify models early and prior to studying or documenting analysis results

Structural Model Checks
(Presented for Nastran, Applicable for Most Solvers)

1. Unit gravity
   1. Geometry check (element quality)
   2. Weight check
   3. Grounding
   4. OLOAD (applied forces)
   5. Restraint forces (SPC forces)
2. Free-free modes
   1. Grounding
   2. Well-separated 6 rigid body modes
3. Fixed modes
   1. Modal effective mass fractions
4. Unit thermal soak
5. F06 warnings
6. Other Checks

[1] Unit Gravity Check

- The three orthogonal unit gravity loads serve as good static load cases for checking:
  - Displaced shapes (allows us to observe possible element connectivity issues)
  - Restraint force (SPCF) output
  - Representative stress contour results (your eventual stress contours will be combinations of the stress contours from your three orthogonal unit load cases; worrisome stress gradients observed in the unit load contour results should be addressed because these same gradients would occur in your final stress contours)
- The stress contours obtained from your unit loads will serve to point out stress concentration areas where possible mesh refinement is required; in general, the stress contours obtained from your unit loads will serve to validate your chosen mesh size.
- Check the OLOAD and SPCFORCE resultants in the .f06 file:
  - Total OLOAD force should be equal to weight
  - SPC force resultant should be equal and opposite to OLOAD resultant
  - F = ma!
- Compare SPC force in DOF123 to weight (multiply the mass by the appropriate gravitational acceleration constant). Percent difference should be minimal.

[1] Sample 1G Static Loads Input


```
SOL      101      $ Statics
CEND
TITLE     =GENERAL PURPOSE SPACECRAFT (GPSC)
SUBTITLE =1 G STATIC LOADS
$
SPC     = 10
$
DISP(PLOT) = ALL
$
SPCF(PLOT) = ALL
$
SUBCASE 1
  LABEL = X DIRECTION GRAVITY LOAD
  LOAD  = 10
$
SUBCASE 2
  LABEL = Y DIRECTION GRAVITY LOAD
  LOAD  = 20
$
SUBCASE 3
  LABEL = Z DIRECTION GRAVITY LOAD
  LOAD  = 30
$
BEGIN BULK
$
PARAM    WTMASS   .00259
PARAM    GRDPNT  0
$
GRAV     10       0        386.1    1.0      0.0       0.0
GRAV     20       0        386.1    0.0      1.0       0.0
GRAV     30       0        386.1    0.0      0.0       1.0
$
INCLUDE 'gpsc.blk'
INCLUDE 'gpsc.prp'
$
ENDDATA
```

Fix base

Displacement and SPCF output

Subcase for X direction load

Subcase for Y direction load

Subcase for Z direction load

Define gravity loads

ENGINEERING, INC.

[1] Sample OLOAD Resultant

0 OLOAD RESULTANT

	SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
0	1	FX	5.712465E+03	----	----	----	1.218243E+05	2.064367E+02
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	0.000000E+00	0.000000E+00	0.000000E+00	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	0.000000E+00	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	5.712465E+03	0.000000E+00	0.000000E+00	0.000000E+00	1.218243E+05	2.064367E+02
0	2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	5.712465E+03	----	-1.218243E+05	----	6.224769E-04
		FZ	----	----	0.000000E+00	0.000000E+00	0.000000E+00	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	0.000000E+00	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	5.712465E+03	0.000000E+00	-1.218243E+05	0.000000E+00	6.224769E-04
0	3	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
		FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
		FZ	----	----	5.712465E+03	-2.064367E+02	-6.224769E-04	----
		MX	----	----	----	0.000000E+00	----	----
		MY	----	----	----	----	0.000000E+00	----
		MZ	----	----	----	----	----	0.000000E+00
		TOTALS	0.000000E+00	0.000000E+00	5.712465E+03	-2.064367E+02	-6.224769E-04	0.000000E+00

- Total force should be equal to weight

[1] Sample SPC Force Resultant

SPCFORCE RESULTANT

	SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
0	1	FX	-5.712465E+03	----	----	----	0.000000E+00	-8.698672E+01
		FY	----	-1.094236E-11	----	0.000000E+00	----	-1.112148E+02
		FZ	----	----	-1.148237E-11	2.944704E-06	-7.793230E+04	----
		MX	----	----	----	-2.943455E-06	----	----
		MY	----	----	----	----	-4.389197E+04	----
		MZ	----	----	----	----	----	-8.235134E+00
		TOTALS	-5.712465E+03	-1.094236E-11	-1.148237E-11	1.248964E-09	-1.218243E+05	-2.064367E+02
0	2	FX	-9.947598E-12	----	----	----	0.000000E+00	-3.059827E-04
		FY	----	-5.712465E+03	----	0.000000E+00	----	-3.159703E-04
		FZ	----	----	-4.320100E-12	7.793100E+04	2.999597E-06	----
		MX	----	----	----	4.389327E+04	----	----
		MY	----	----	----	----	-3.000805E-06	----
		MZ	----	----	----	----	----	-5.242606E-07
		TOTALS	-9.947598E-12	-5.712465E+03	-4.320100E-12	1.218243E+05	-1.207354E-09	-6.224773E-04
0	3	FX	-6.039613E-13	----	----	----	0.000000E+00	2.660962E-05
		FY	----	-3.922196E-12	----	0.000000E+00	----	-1.318799E-05
		FZ	----	----	-5.712465E+03	2.245308E+02	6.196071E-04	----
		MX	----	----	----	-1.809408E+01	----	----
		MY	----	----	----	----	2.869821E-06	----
		MZ	----	----	----	----	----	-1.342161E-05
		TOTALS	-6.039613E-13	-3.922196E-12	-5.712465E+03	2.064367E+02	6.224769E-04	1.909939E-11

- SPC force resultant should be equal and opposite to OLOAD resultant

ATA ENGINEERING, INC.

[1] Geometry Check

- Nastran automatically performs element quality checks
- Check the NX Nastran Quick Reference Guide (QRG) for "System Element Checks" and "User Controlled Element Checks"
  - System Element Checks are fixed
- GEOMCHECK allows override of default element quality check values
- Not usually necessary to change defaults
- If Nastran runs (no FATAL errors), you can usually ignore element warnings for dynamic solutions
  - Local element stresses may be inaccurate
  - If element warnings exist and you are performing a stress analysis, review warnings for location and type to ensure proper element quality

Name	Value Type	Default	Comment
Q4_SKEW	Real≥0.0	30.0	Skew angle in degrees
Q4_TAPER	Real≥0.0	0.50	Taper ratio
Q4_WARP	Real≥0.0	0.05	Surface warping factor
Q4_IAMIN	Real≥0.0	30.0	Minimum Interior Angle in degrees
Q4_IAMAX	Real≥0.0	150.0	Maximum Interior Angle in degrees
T3_SKEW	Real≥0.0	10.0	Skew angle in degrees
T3_IAMAX	Real≥0.0	160.0	Maximum Interior Angle in degrees
TET_AR	Real≥0.0	100.0	Longest edge to shortest edge aspect ratio
TET_EPLR	Real≥0.0	0.50	Edge point length ratio
TET_DETJ	Real	0.0	| J | minimum value
TET_DETG	Real	0.0	| J | minimum value at vertex point
HEX_AR	Real≥0.0	100.0	Longest edge to shortest edge aspect ratio
HEX_EPLR	Real≥0.0	0.50	Edge point length ratio
HEX_DETJ	Real	0.0	| J | minimum value
HEX_WARP	Real≥0.0	0.707	Face warp coefficient
PEN_AR	Real≥0.0	100.0	Longest edge to shortest edge aspect ratio
PEN_EPLR	Real≥0.0	0.50	Edge point length ratio
PEN_DETJ	Real	0.0	| J | minimum value
PEN_WARP	Real≥0.0	0.707	Quadrilateral face warp coefficient
BEAM_OFF	Real≥0.0	0.15	CBEAM element offset length ratio
BAR_OFF	Real≥0.0	0.15	CBAR element offset length ratio

GEOMCHECK Specifies Geometry Check Options

Specifies tolerance values and options for optional finite element geometry tests.

Format:

GEOMCHECK test_keyword [= tol_value], [MSGLIMIT = n], [MSGTYPE = {FATAL / INFORM / WARN}],

[SUMMARY], [NONE]

ATA ENGINEERING, INC.

[1] Weight Check

- The weight check is a mass properties check that Nastran performs using the WEIGHTCHECK card; WEIGHTCHECK will give you a report on the mass properties of your part, including the mass moment of inertia and the center of mass.
- The weight-check output should be reviewed against expectations to gain confidence that all material property entries in your model are correct and that all material property values have been exported in the correct units.
- The weight check can be performed as part of the free-free modes or unit loads solution (or as part of both solutions).
- It is recommended you doing WEIGHTCHECK on all sets.
  - Rigid body mass should not be affected by reduction
  - Exception is mass fixed to ground (S-set)
- Multiply the mass by the appropriate gravitational acceleration constant in units that are consistent with the length units of your model, and check that this computed weight matches the weight of your FEM that you had measured in the pre-processor.
  - Note if your deck contains DMIG mass matrices, the mass will not match the pre-processor. Still compare to verify they look as expected.

WEIGHTCHECK Rigid Body Mass Reduction Check

At each stage of the mass matrix reduction, compute rigid body mass and compare with the rigid body mass t the g-set.

Format:

$$\text{WEIGHTCHECK}\left[\begin{array}{l}\left(\left[\begin{array}{c}\text{PRINT}\\ \text{NOPRINT}\end{array}\right], \text{PUNCH}, \text{SET} = \left(\left\{\begin{array}{c}\text{G, N, N + AUTOSPC, F, A,V}\\ \text{ALL}\end{array}\right\}\right)\right)\\ \text{GRID} = \text{gid}, \text{CGI} = \left[\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right], \left[\begin{array}{c}\text{WEIGHT}\\ \text{MASS}\end{array}\right]\Big)\end{array}\right] = \left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

Examples:

WEIGHTCHECK=YES
WEIGHTCHECK(GRID=12,SET=(G,N,A),MASS)=YES

$$\left[M_{rigid}\right] = [\Phi]^T [M][\Phi]$$

```
0                        O U T P U T   F R O M   W E I G H T   C H E C K
0                                  DEGREES OF FREEDOM SET = G
                                   REFERENCE POINT =        0
                                                M O
 *  5.712471E+03  0.000000E+00  0.000000E+00  0.000000E+00  1.218244E+05  2.064369E+02 *
 *  0.000000E+00  5.712471E+03  0.000000E+00 -1.218244E+05  0.000000E+00  6.224775E-04 *
 *  0.000000E+00  0.000000E+00  5.712471E+03 -2.064369E+02 -6.224775E-04  0.000000E+00 *
 *  0.000000E+00 -1.218244E+05 -2.064369E+02  1.140252E+07 -1.266015E-01  1.039568E-02 *
 *  1.218244E+05  0.000000E+00 -6.224775E-04 -1.266015E-01  9.685567E+06  3.085192E+05 *
 *  2.064369E+02  6.224775E-04  0.000000E+00  1.039568E-02  3.085192E+05  9.654569E+06 *
                                                 S
                          *  1.000000E+00  0.000000E+00  0.000000E+00 *
                          *  0.000000E+00  1.000000E+00  0.000000E+00 *
                          *  0.000000E+00  0.000000E+00  1.000000E+00 *
     DIRECTION
 MASS AXIS SYSTEM (S)     MASS              X-C.G.        Y-C.G.        Z-C.G.
         X            5.712471E+03      0.000000E+00 -3.613793E-02  2.132604E+01
         Y            5.712471E+03      1.089682E-07  0.000000E+00  2.132604E+01
         Z            5.712471E+03      1.089682E-07 -3.613793E-02  0.000000E+00
                                               I(S)
                          *  8.804480E+06  1.266240E-01 -2.367066E-02 *
                          *  1.266240E-01  7.087536E+06 -3.041167E+05 *
                          * -2.367066E-02 -3.041167E+05  9.654562E+06 *
                                               I(Q)
                          *  8.804480E+06                             *
                          *                7.051999E+06               *
                          *                              9.690099E+06 *
                                                 Q
                          *  1.000000E+00  0.000000E+00  0.000000E+00 *
                          *  0.000000E+00  9.932418E-01  1.160631E-01 *
                          *  0.000000E+00 -1.160631E-01  9.932418E-01 *
```

Sample weight check output in f06 file

ATA ENGINEERING, INC.

[2] Free-Free Modes

- Using a modal solution (SOL 103 Real Eigenvalues) with the model unloaded and unrestrained ("free-free" means no loads or BCs), we can check for the presence of six rigid body modes. The presence of six rigid body modes provides assurance that the model is not artificially restrained.
- Rigid body mode frequencies should be orders of magnitude below the frequency of the first flexible mode. Good separation (three to four orders of magnitude) between the sixth rigid body mode and the first flexible mode indicates that the model most likely has good numerical conditioning.
  - Sometimes hard to attain for models with very low frequency modes (e.g., slosh modes)
- In a well-conditioned model, rigid body modes should be below 0.005 Hz. This may increase for large complex models and should be considered based on proper expectations for that model and solution type.
- Include a ground check in this run (see next slide) along with weight check.
- Visualize the mode shapes.

[2] Sample Free-Free Modes Input


```
SOL     103     $ Normal modes
CEND
TITLE    =GENERAL PURPOSE SPACECRAFT (GPSC)
SUBTITLE =FREE-FREE NORMAL MODES
$
METHOD     = 70
$
DISP(PLOT) = 100
$
GROUNDCHECK(SET=(G,N),DATAREC=YES)=YES
$
WEIGHTCHECK(SET=ALL)=YES
$
BEGIN BULK
$
PARAM   WTMASS  .00259
PARAM   GRDPNT  0
$
EIGRL   70              70.0
$
$  Spacecraft bulk data
$
INCLUDE 'gpsc.blk'
INCLUDE 'gpsc.prp'
$
ENDDATA
```

Select EIGRL card

Request displacement output

GROUNDCHECK and WEIGHTCHECK

Leave F1 field blank on EIGRL card

[2] Sample Free-Free Modes Output

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	-3.317837E-09	5.760066E-05	9.167430E-06	1.000000E+00	-3.317837E-09
2	2	-2.240995E-09	4.733915E-05	7.534259E-06	1.000000E+00	-2.240995E-09
3	3	-1.251465E-09	3.537605E-05	5.630273E-06	1.000000E+00	-1.251465E-09
4	4	6.402843E-10	2.530384E-05	4.027231E-06	1.000000E+00	6.402843E-10
5	5	9.604264E-10	3.099075E-05	4.932331E-06	1.000000E+00	9.604264E-10
6	6	1.979060E-09	4.448663E-05	7.080267E-06	1.000000E+00	1.979060E-09
7	7	8.597009E+03	9.272005E+01	1.475686E+01	1.000000E+00	8.597009E+03
8	8	1.024054E+04	1.011955E+02	1.610577E+01	1.000000E+00	1.024054E+04
9	9	1.482603E+04	1.217622E+02	1.937906E+01	1.000000E+00	1.482603E+04
10	10	1.957670E+04	1.399168E+02	2.226845E+01	1.000000E+00	1.957670E+04
11	11	2.848863E+04	1.687858E+02	2.686309E+01	1.000000E+00	2.848863E+04
12	12	2.865324E+04	1.692727E+02	2.694058E+01	1.000000E+00	2.865324E+04
13	13	5.320893E+04	2.306706E+02	3.671237E+01	1.000000E+00	5.320893E+04
14	14	7.195388E+04	2.682422E+02	4.269207E+01	1.000000E+00	7.195388E+04
15	15	1.008722E+05	3.176039E+02	5.054823E+01	1.000000E+00	1.008722E+05
16	16	1.013238E+05	3.183141E+02	5.066125E+01	1.000000E+00	1.013238E+05
17	17	1.167970E+05	3.417557E+02	5.439211E+01	1.000000E+00	1.167970E+05
18	18	1.290095E+05	3.591789E+02	5.716510E+01	1.000000E+00	1.290095E+05
19	19	1.463482E+05	3.825548E+02	6.088549E+01	1.000000E+00	1.463482E+05
20	20	1.682320E+05	4.101609E+02	6.527914E+01	1.000000E+00	1.682320E+05

- Only negative eigenvalues should be small numbers
- In this case, output is greater than six orders of magnitude spread between frequencies of rigid body modes and first flexible mode
  - Hard to achieve for a large model

ENGINEERING, INC.

[2] Grounding (1/2)

- The groundcheck performs a series of rigid body translations and rotations of the structure, multiplying the stiffness matrix by the rigid body transformation matrix.
- The groundcheck will identify unintentional constraints and ill-conditioning in the stiffness matrix. This can occur from artificial internal loading in a FEM from motion, bad modeling, and/or bad element formulations. Ill-conditioning of the model stiffness matrix can lead to inaccurate results.
- A grounding check is performed using the GROUNDCHECK card in Nastran.
  - Strain energies resulting from six rigid body displacements are compared against a specified threshold.
  - Unrestrained stiffness matrix times rigid body vectors should result in zero force.
  - If the structure is connected properly and not artificially restrained the structure will "PASS" the rigid body displacement check in all six directions.
- The rigid body rotation checks are dependent upon the reference location the check is about. If no grid is specified for the reference location, the unit rotation check is made about the origin of the Nastran basic coordinate system. Verify whether the origin is appropriate to use for this check, and if not, set the grid ID in the GROUNDCHECK card using "GRID=XXXXXXXX."

GROUNDCHECK Rigid Body Motion Grounding Check

Perform grounding check analysis on stiffness matrix to expose unintentional constraints by moving the model rigidly.

Format:

$$\text{GROUNDCHECK}\left[\begin{matrix}\left(\begin{bmatrix}\text{PRINT}\\\text{NOPRINT}\end{bmatrix}, \text{PUNCH, SET} = \left(\left\{\begin{matrix}\text{G, N, N + AUTOSPC, F, A}\\\text{ALL}\end{matrix}\right\}\right)\right)\\ \text{GRID = gid, THRESH = e,DATAREC} = \begin{bmatrix}\text{YES}\\\text{NO}\end{bmatrix}, (\text{RTHRESH = r})\end{matrix}\right] = \left\{\begin{matrix}\text{YES}\\\text{NO}\end{matrix}\right\}$$

Examples:

GROUNDCHECK=YES
GROUNDCHECK(GRID=12,SET=(G,N,A),THRESH=1.E-5,DATAREC)=YES

$$[F]=[K][\Phi_{RB}]$$

$$se_i = \frac{1}{2}\{\phi_i\}^T[K]\{\phi_i\}$$

[2] Grounding (2/2)

- Enable DATAREC=YES if you wish to print the data recovery of grounding forces.
- At a minimum, check G and N sets, but all sets are available in Nastran for this check.
- By default, the strain energy threshold set to largest diagonal stiffness × 1×10-10.
  - Stiff springs can increase threshold.
  - If the threshold was automatically set by Nastran, verify the limit is less than 1.
- For most models strain energies < ~0.1, grounding forces < 1.0 N and moments < 0.5 N-m indicate a good model.

```
$*$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$*
$* CASE CONTROL
$*
$*$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$*
ECHO = NONE
GROUNDCHECK(PRINT,SET=(ALL),GRID=1481786,DATAREC=YES)=YES
```

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
    RESULTS OF RIGID BODY CHECKS OF MATRIX KGG      (G-SET)  FOLLOW:
    PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   1.000000E-02
          DIRECTION         STRAIN ENERGY        PASS/FAIL
          ---------         -------------        ---------
             1               2.000007E-05          PASS
             2               4.000005E-05          PASS
             3               1.000002E-04          PASS
             4               4.985046E+01          FAIL
             5               4.464892E+01          FAIL
             6               8.216817E+01          FAIL
    SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
      1. CELASI ELEMENTS CONNECTING TO ONLY ONE GRID POINT;
      2. CELASI ELEMENTS CONNECTING TO NON-COINCIDENT POINTS;
      3. CELASI ELEMENTS CONNECTING TO NON-COLINEAR DOF;
      4. IMPROPERLY DEFINED DMIG MATRICES;
```

```
1    GROUNDCHECK EXAMPLE                                         DECEMBER  28, 2005  NX NASTRAN 10/15/04   PAGE    12

                                                                                  DIRECTION        4
                                 G R O U N D   C H E C K   F O R C E S   ( G - S E T )

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
           1      G       0.0            0.0          -1.000000E+05   0.0            0.0            0.0
           2      G       0.0            0.0           1.000000E+05   0.0            0.0            0.0
1    GROUNDCHECK EXAMPLE                                         DECEMBER  28, 2005  NX NASTRAN 10/15/04   PAGE    16

                                                                                  DIRECTION        5
                                 G R O U N D   C H E C K   F O R C E S   ( G - S E T )

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
          44      G       0.0            0.0          -4.410524E+03  -1.344394E+03   1.326177E+03   0.0
          45      G       0.0            0.0           4.436500E+03   0.0            1.330224E+03   0.0
          48      G       0.0            0.0           4.436886E+03  -1.316576E+03   0.0            0.0
          49      G       0.0            0.0          -4.462862E+03   0.0            0.0            0.0
```

[3] Fixed Modes With MEFFMASS

- Apply appropriate boundary conditions and run modes in the frequency range of interest.
- Look at MEFFMASS (modal effective mass)
  - Measure of how much mass is moving in a mode in a given direction
  - If all modes are retained, sum of effective mass across all modes in any direction is equal to total mass in that direction
  - Modal effective mass fraction (FRACSUM option) is percentage of total mass associated with that mode
  - Total effective mass in any direction can be used as a measure of modal "completeness"
  - Can be used to select "important" or "target" modes (i.e., modes that generate large effective interface forces/moments)

MEFFMASS Modal Effective Mass Output Request

Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis.

Format:

$$\text{MEFFMASS}\left[\begin{bmatrix}\text{PRINT}\\ \text{NOPRINT}\end{bmatrix}\begin{bmatrix}\text{PUNCH}\\ \text{NOPUNCH}\end{bmatrix}\begin{bmatrix}\text{PLOT}\\ \text{NOPLOT}\end{bmatrix}, \text{GRID} = \right.$$

$$\left.\text{gid}, \begin{bmatrix}\text{SUMMARY, PARTFAC,}\\ \text{MEFFM, MEFFW,}\\ \text{FRACSUM, ALL}\end{bmatrix}\right] = \begin{Bmatrix}\text{YES}\\ \text{NO}\end{Bmatrix}$$

Examples:

MEFFMASS
MEFFMASS(GRID=12, SUMMARY, PARTFAC)

$$MEFFMASS_{ij} = \left(\left[\phi_i\right]^T \left[M\right]\left[\phi_{RBj}\right]\right)^2$$

[3] Sample MEFFMASS Input


```
SOL      103      $ Normal modes
CEND
TITLE    =GENERAL PURPOSE SPACECRAFT (GPSC)
SUBTITLE =FIXED BASE NORMAL MODES
$
METHOD = 70
$
SPC    = 10
$
DISP(PLOT) = ALL
$
MEFFMASS(FRACSUM)=YES
$
BEGIN BULK
$
PARAM   WTMASS  .00259
PARAM   GRDPNT  0
$
EIGRL   70              70.0
$
$  Spacecraft bulk data
$
INCLUDE 'gpsc.blk'
INCLUDE 'gpsc.prp'
$
ENDDATA
```

Select EIGRL card

Fix base

Displacement output

MEFFMASS request

Leave F1 field blank on EIGRL card

ATA ENGINEERING, INC.

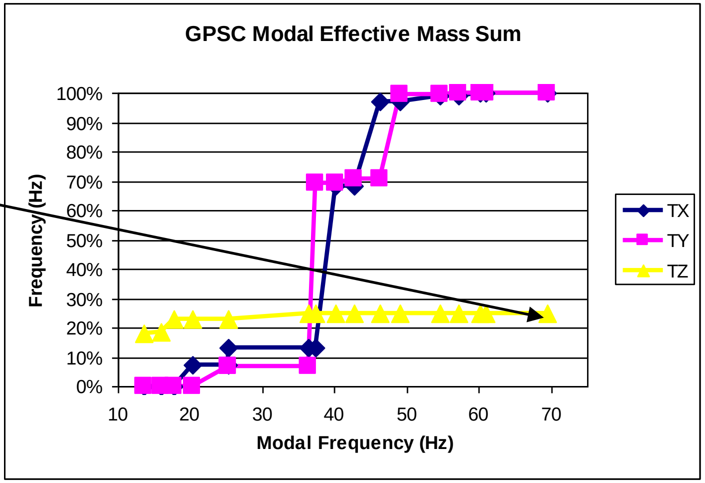
[3] Modal Effective Mass Output

- Free-Free modes
  - Check amount of rigid mass for first six modes summed across all DOF; it should be at least 95%, and ideally closer to 100%.

Mode #	Freq (Hz)	Modal Effective Mass Fraction						
		T1	T2	T3	R1	R2	R3	SUM
1	0.0009	26%	0%	13%	2%	38%	18%	99%
2	0.0007	27%	1%	17%	5%	25%	25%	100%
3	0.0005	13%	76%	1%	4%	0%	6%	100%
4	0.0008	31%	14%	1%	7%	0%	48%	100%
5	0.0009	2%	2%	26%	40%	30%	0%	100%
6	0.0011	0%	7%	42%	43%	6%	2%	99%
7	132.2	0%	0%	0%	0%	0%	0%	0%

- Fixed Modes
  - The target value for sum of modal effective mass for a DOF across the frequency range depends on the specific analysis. In general, as the sum for a given DOF increases, the analysis captures the motion in that DOF more completely.

Frequency range not sufficient to capture motion in Z

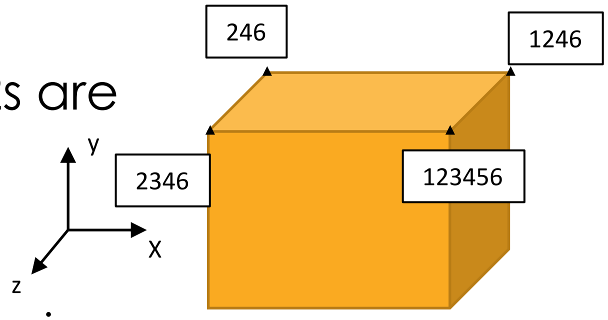


		TX		TY		TZ	
Mode	Freq (Hz)	Fraction	Sum	Fraction	Sum	Fraction	Sum
1	13.59	0.0%	0.0%	0.0%	0.0%	18.0%	18.0%
2	16.04	0.0%	0.0%	0.0%	0.0%	0.5%	18.5%
3	17.65	0.0%	0.0%	0.0%	0.0%	4.6%	23.0%
4	20.26	7.6%	7.6%	0.0%	0.0%	0.0%	23.0%
5	25.25	0.0%	7.6%	6.7%	6.7%	0.0%	23.0%
6	25.27	5.6%	13.2%	0.0%	6.7%	0.0%	23.0%
7	36.26	0.0%	13.2%	0.0%	6.8%	1.9%	25.0%
8	37.35	0.0%	13.2%	62.5%	69.2%	0.0%	25.0%
9	40.06	55.2%	68.4%	0.0%	69.2%	0.0%	25.0%
10	42.70	0.0%	68.4%	1.4%	70.6%	0.0%	25.0%
11	46.34	28.9%	97.3%	0.0%	70.6%	0.0%	25.0%
12	49.10	0.0%	97.3%	28.8%	99.4%	0.0%	25.0%
13	54.54	1.7%	98.9%	0.0%	99.4%	0.0%	25.0%
14	57.25	0.0%	98.9%	0.5%	99.8%	0.0%	25.0%
15	60.23	0.9%	99.9%	0.0%	99.8%	0.0%	25.0%
16	60.90	0.0%	99.9%	0.0%	99.9%	0.0%	25.0%
17	69.47	0.0%	99.9%	0.0%	99.9%	0.0%	25.0%

ATA ENGINEERING, INC.

[4] Unit Thermal Soak

- The unit temperature check is intended to confirm that all materials, and especially RBE2s, have a coefficient of thermal expansion (CTE).
- If the part and its RBE2s have the same CTE, then, when a unit temperature load is applied, you should observe zero stress in the part. This is what you are checking for—obtaining zero stress with a unit temperature load.
- Enable RIGID=LAGRAN in the deck if RBE2s are present
- Kinematic boundary conditions
  - One bolt RBE2 center node
  - Hold corners of solid mesh to allow free expansion: see figure



- Compare results to a hand calc $\alpha l \Delta T$
- May be necessary to convert RBE2s to stiff beams to ensure no artificial stress is added to model

[4] Sample Unit Thermal Input


```
SOL     101     $ Static, thermal load
CEND
TITLE    =GENERAL PURPOSE SPACECRAFT (GPSC)
SUBTITLE =UNIT THERMAL CHECK
$
$
SPC    = 10
$
RIGID=LAGRAN
DISP(PLOT) = ALL
STRESS(PLOT) = ALL
$
TEMP(INITIAL) = 1
TEMP(LOAD) = 10
$
BEGIN BULK
$
PARAM   WTMASS  .00259
PARAM   GRDPNT  0
$
$ Loading
$ Initial Temperature
TEMPD          1      20.
$ Unit Temperature
TEMPD         10      21.
$
$  Spacecraft bulk data, same CTE
$
INCLUDE 'gpsc.blk'
INCLUDE 'gpsc.prp'
$
ENDDATA
```

Fix base kinematically

Displacement & stress output

Temperature loading

ATA ENGINEERING, INC.

[5] F06 Warnings

- It is generally a good idea to check the Nastran warnings in the f06
  - Materials are valid
  - Element quality checks and their impact on stress results

[6] Other Checks

- Epsilon
  - Error in the load predicted from the product of stiffness matrix and solution set displacement compared to the actual input load vector: a measure of accuracy of linear solution (should be near zero).
- Separation ratio
  - The ratio of grounding force or moment in each DOF to the corresponding diagonal element of the stiffness matrix. Performed in Nastran using a DMAP. Not usually required unless requested for a specific project.
- Strain energy
  - Should be non-zero (zero strain energy implies that no load was applied).
  - Rotational strain energies almost always higher than translational.
- Max Ratio Errors
  - MAXRATIO is a measure of the numerical conditioning of a matrix.
  - Created while decomposing matrix to a triangular factor ($[A] = [L]^T[L]$).
  - Ratio of diagonal of stiffness matrix to the corresponding diagonal of triangular factor.
  - Any ratio exceeding MAXRATIO will cause solution to fail.
  - Default is 1E7:
    - MAXRATIO can be increased to 1E8 or 1E9 with a fair degree of confidence (indicates large stiffness discontinuity in the model).
    - Values larger than 1E9 should be avoided since they most likely indicate a model error.

Post-Processor Activities

- Viewing results
  - Do they make sense?
  - Is the structure behaving as intended?
  - Is the mesh density appropriate, or is further refinement needed for stress results?
- Visualize and document mode shapes
  - Free-free and fixed modes runs.
  - Look at modal stress to see what is going on with the deformation animated. Save as a GIF.

Practical Project Recommendations

- Always spend time scrutinizing the initial set of analysis results. Do they make sense? A mistake caught this early in the project cycle will save tremendous effort overall.
- Do not reduce the results to sorted tables of margins and modes until an understanding of the initial set of analysis results is obtained.
- Beware of significant result changes due to minor model modifications. Do results make sense?
- For large nonlinear analyses, perform a comparable linear analysis for comparison purposes.

Apply Systematic Methods to Verify the Analysis Model Data and Results

- With new analysis area or analysis method, check simple example first and then work incrementally, i.e., add complexity, details, or cases for the real problem.
- Used closed-form solutions to check methods whenever possible; anticipate what results and outcomes. Investigate unexpected solutions.
- Examples of checking via hand calculations:
  - Would this panel buckle under that applied load?
  - Would this cylinder elongate this much under that thermal load?

Common Solutions to Problems (1 of 2)

- Common input file errors:
  - Missing information
  - Duplicate information
  - Incorrectly entered information
- Insufficient resources:
  - Need more RAM (typical for nonlinear runs)
  - Need more disk space
    - Easiest to clear more disk space if possible
    - Otherwise, consider symmetry, local refinement, or manual remeshing
- Disconnected model:
  - In Nastran, the fatal error message is "Run terminated due to excessive pivot ratios"
  - The free-free modes checks should catch this
- MAXRATIO errors
  - MAXRATIO can be increased to 1E8 or 1E9 with a fair degree of confidence (indicates large stiffness discontinuity in the model)
  - Values larger than 1E9 should be avoided since they most likely indicate a model error
  - To ignore MAXRATIO errors, use PARAM,BAILOUT,-1
    - Use this to attempt to debug, but do not run with this!

Common Solutions to Problems (2 of 2)

- Pivot ratios too high
  - Check material and element properties, like E, thickness, moment of inertia
  - Something with high stiffness connected to something with low stiffness
    - Spring stiffness much softer than adjacent components
- Grounding errors
  - Springs (CELAS) on non-coincident GRIDs
    - Correction: Use CBUSH or make grids coincident
  - Springs (CELAS) with incompatible displacement coordinate systems
    - Correction: Use CBUSH
  - Incorrect external stiffness matrix (DMIG/OUTPUT4/etc.)
    - Some grounding is almost inevitable with DMIG cards due to truncation of significant figures
  - Incorrect MPC equations
    - Shows up in N-set check (not G-set)
  - Very poorly formed elements
  - Grounding beyond the N-set is usually not of concern
    - AUTOSPC process can introduce artificial grounding in N+AUTOSPC set
    - Restraints introduce grounding in F-set and A-set
    - Almost all user-created grounding problems identified on G- and N-set

ENGINEERING, INC.

Before Meshing

Done? (Y/N)	Major Tasks	Sub Tasks
	Confirm that your model is in the correct units. Check the units of the model file and then make some measurements on the part to convince yourself.	--
	Understand what type of simulation will be run on the FEM. (This will help determine which regions are critical, what mesh type to use, etc.). Stress analysis typically needs a finer mesh. Dynamic response (modes and acceleration response) models are often not as detailed as stress models.	Identify analysis assumptions and document
	Determine what type of mesh is to be used (e.g., beam, shell, solid).	Are triangular shells acceptable?
		Will beams require an offset?
	Determine how to appropriately simplify the given geometry for modeling. What geometry details are important Which will cause meshing problems?	--
	Clean up the geometry for meshing.	Use Free Edge checks to make sure surfaces are properly stitched together.
		Remove sliver surfaces or unimportant features (small holes, etc.).
		Make sure surfaces/volumes are partitioned based upon changes in properties or other important features.
	Determine appropriate mesh density based on given geometry, computational requirements, and desired accuracy. Discuss with your project engineer.	--

During Meshing

Confirm This Is Required (Y/N)	Done? (Y/N)	Major Tasks	Sub Tasks
		Run quality checks on all elements.	If some elements fail, determine whether these elements are in critical areas that will require remeshing for accurate results
			See your project manager for appropriate quality check thresholds
		Check Element connections and visualization	View element free edges
			Check for unexpected coincident nodes and elements
			Check for unconnected nodes
			Check shell elements for consistent element normals.
			View element offsets (do they make sense/as intended?)
			View material orientations
			View bar/beam orientations
			Document mass properties
			Check the mass and CG of your model; does it match what you expect?
			View and document boundary conditions and loads

Analysis Checks

Confirm This Is Required (Y/N)	Done? (Y/N)	Major Tasks	Sub Tasks
		Run Solver Model Checks	Unit Gravity Loads (WEIGHTCHECK, GEOMCHECK, OLOAD, SPCFORCE)
			Free-Free Modes (Grounding)
			Fixed Modes (MEFFMASS)
			Unit Thermal Soak
			Unit Displacements

Additional Checks

Done? (Y/N)	Major Tasks	Sub Tasks
	Post-Processing	View modes, thermal stresses, deformation under gravity
		Perform hand calculations to verify results make sense
	Model Organization	Number nodes and elements different components for easy post-processing
		Number BCs and Interface elements for easy post-processing
		Name entities and properties (material and physical) with meaningful names
		Create groups for post-processing

Contact Us

13290 Evening Creek Drive S
San Diego, CA 92128

(858) 480-2000

plm_sales@ata-e.com

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

@ATAEngineering

ata-engineering