

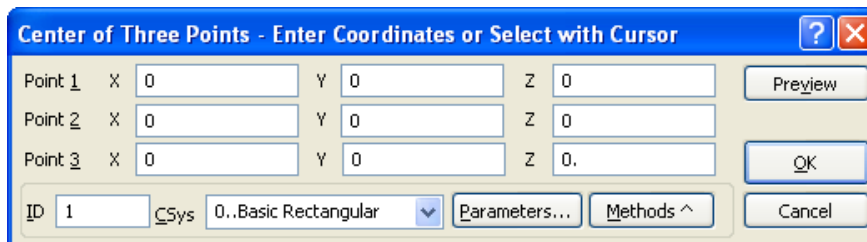
# What's New for version 10.3

## User Interface

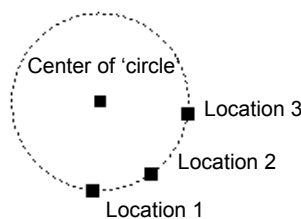
General, Entity Select, Menu, Toolbars, Model Info tree, Data Table, Entity Editor, Data Surface Editor, Meshing Toolbox, PostProcessing Toolbox

### General

- Added *Filter Title* and *Clear Title Filter* icon buttons to the *Load Set*, *Constraint Set*, *Group*, *Layer*, *View*, *Solid*, and *Freebody Manager* dialog boxes.
- Only tabs of entity types which currently exist in the model will be displayed in the *View*, *Visibility* dialog box.
- User created Toolbars will now transfer between versions of FEMAP.
- Pressing *Ctrl+M* while in a dialog box field asking for a length will display the *Select Curve to Measure* dialog box, which will return the selected curves length.
- Added the *Locate Center* to the Methods for specifying the a coordinate.



The *Locate Center* method requires three specified locations which are not colinear to determine a “circle”. The “center” location is then determined by finding the center point of the “circle”. A geometric circular curve is NOT created.



### Entity Select

- Added “on Property” and “on CSys” methods when selecting Coordinate Systems.

### Menu

- Added *Tools*, *Toolbars*, *Aeroelasticity* command. See *Toolbars* section.
- Added *Model*, *Aeroelasticity...* commands (Panel/Body, Property, Spline, and Control Surface) to create the various entities used in Static Aeroelastic analysis and Aerodynamic Flutter analysis. See *Aeroelasticity* section.
- Added *Mesh*, *Geometry Preparation* command. See *Meshing* section.
- Added commands to *Modify*, *Edit...*, *Modify*, *Color...*, *Modify*, *Layer...*, and *Modify* *Renumber...* menus for the Aeroelasticity entities (*Aero Panel/Body*, *Aero Property*, *Aero Spline*, and *Aero Control Surface*).

- Added *Modify, Update Other, Aero Interference Group* command. Allows modification of *IGID* on any number of selected *Aero Panel/Body* entities at the same time.
- Added *List, Output, Force Balance to Data Table* and *List, Output, Force Balance Interface Load to Data Table* commands. Also, updated *List, Output, Force Balance* and *List, Output, Force Balance Interface Load* to use *Freebody* entities. See *Freebody* tool section.
- Added commands to *Delete, Model...* menu to delete the Aeroelasticity entities (*Aero Panel/Body, Aero Property, Aero Spline, and Aero Control Surface*)
- Added *Delete, Output, Freebody* command to delete any number of selected *Freebody* entities.
- Added *Group, Coord Sys, on Property* and *Group, Coord Sys, on CSys* commands to add additional methods to add Coordinate Systems to groups.
- Added *View, Align By, Surface* and *View, Align By, Normal to Plane* commands to align the active view to either the normal of a selected planar surface or the normal of a specified plane, respectively.

### Toolbars

- Added *Aeroelasticity* Toolbar. Contains overall visibility controls (*Draw Entity* check box) of the *Aero Panel, Aero Mesh, Aero Spline, and Aero Control Surfaces* options in the *Labels, Entities and Color* section of the *View, Options* command.
- Added *Mesh Geometry Preparation* icon to *Mesh* Toolbar. See *Meshing* section.

### Model Info tree

- Added *Aero Model* branch and underlying branches for *Panels/Bodies, Properties, Splines, and Control Surfaces*, which allow for creation, copying, editing, listing, and deleting of the various aeroelasticity entities. The color and layer may also be changed.
- Added Visibility check boxes (on/off) for *Aero Model - Panels/Bodies, Splines, and Control Surfaces*.
- Added Compare command to context-sensitive menu for Results. Provides that same functionality as the *Model, Output, Compare* command for the selected sets.

### Data Table

- Added a “Skew” column when using the “Add Element Checks” command.

### Entity Editor

- Added “Skew” field to *Element Quality* section when an element is loaded in the Editor.

### Data Surface Editor

- Added “Mapping Tolerance” to the “Options” of the Output Map Data Surface.

When a “Target” location is projected onto the “Source” data surface and the distance to a discrete data point is less than the tolerance, the “Source” value of the “coincident” location is directly mapped to the “Target” without interpolation. If multiple nodes fall within this tolerance, then the first one encountered numerically will be directly mapped. Default value is the “Merge Tolerance” of the “Target” model.

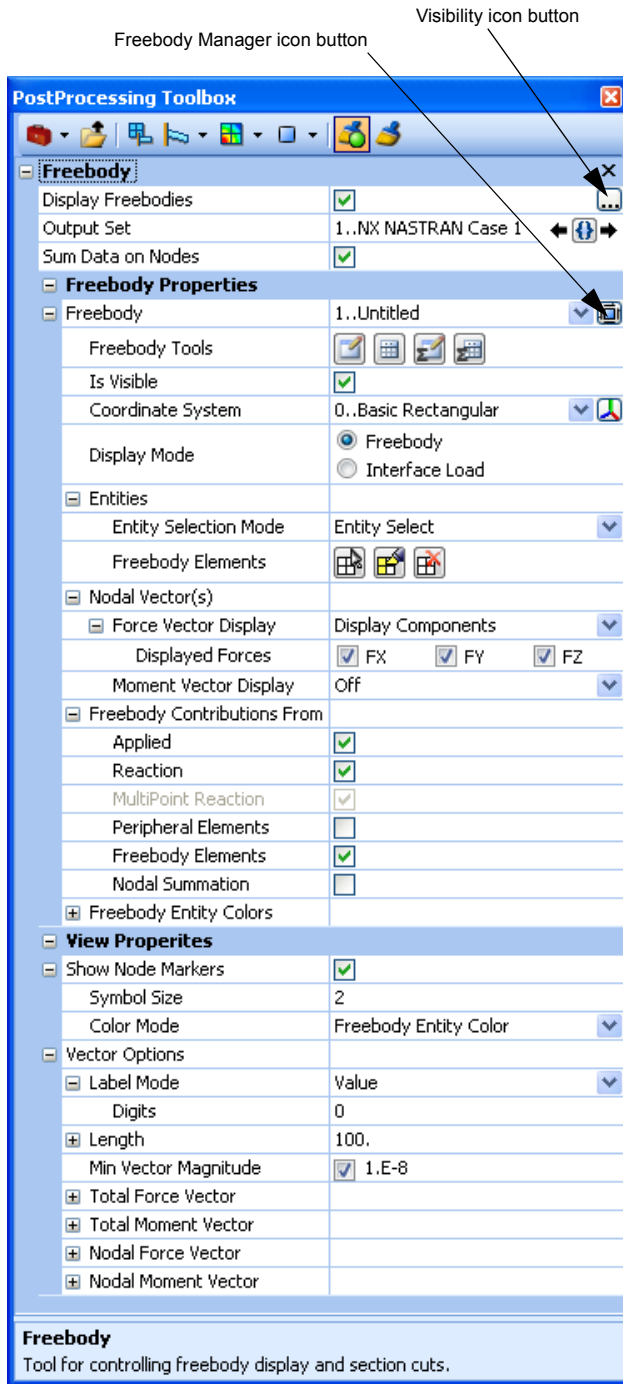
### Meshing Toolbox

- Added *Add Surface Mesh Point* check box to *Feature Removal* tool (*Feature Type* = “Loops” only). Will create a point at the “center” of the “loop”, then use that point as a “mesh point” on the surface. See Section 5.1.2.9, “Mesh, Mesh Control, Mesh Points on Surface...” for more information.
- Performance improvements to *Propagate by Mapped Approach* option in *Mesh Sizing* tool. Also, if no “mesh sizing exists on a curve, now the number of nodes attached is used for the initial mesh sizing.

### PostProcessing Toolbox

- Added *Freebody tool* to all facets of Freebody display post-processing.

The *Freebody* tool is the gateway to using freebody diagrams for post-processing. The freebody display can be performed at any time, whether you are showing a deformed and contour plot, or a simple undeformed plot. The “type” of freebody display, the output set and contributions used in the calculations, and many view options for freebody entities are all controlled via this tool. In order to use the *Freebody* tool fully, the “Grid Point Force” and “Grid Point Moment” results must have been recovered from Nastran. This is done in FEMAP by selecting the “Force Balance” option in the *Nastran Output Requests* dialog box found in the *Analysis Set Manager*. See Section 4.10.1.5, "Output Requests" for more information.



A *Freebody* entity must be created before any additional options may be specified. To do this, use the *Freebody Manager*, which is accessed by pressing the *Add Freebody* icon button next to the drop-down list next to *Freebody* in the *Freebody Properties* section. Multiple *Freebody* entities may be created.

Once *Freebody* entities have been created, each may be made visible or hidden individually in all views using the *Is Visible* check box in the *Freebody Properties* section or the check boxes in the *Freebody* tab of the *Visibility* dialog box (see Section 6.1.4, "View, Visibility..."). The “...” icon button next to *Display Freebodies* will give direct access to the *Visibility* dialog box with the *Freebody* tab selected.

**Options - Freebody tool**

The *Freebody* tool is divided into 3 sections. The top of the *Freebody* tool contains 3 options which affect all Freebody entities in a *View*. The options in the *Freebody Properties* section changes based on which *Freebody* entity is selected with the *Freebody* drop-down list. Options in *View Properties* section change depending on which *View* is currently active in the model.

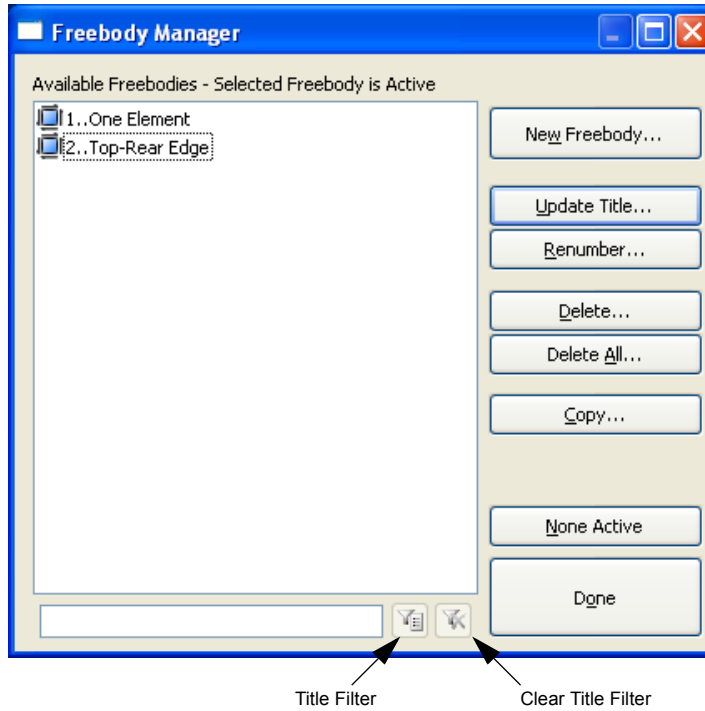
The three options at the top of the *Freebody* tool are used to control the overall visibility of all *Freebody* entities (*Display Freebodies*), which *Output Set* will be used to create the freebody display, and if data should be summed at nodes (*Sum Data on Nodes*). The arrow icons can be used to go to the *Next* or *Previous* output set or the *Select Output Set* icon button can be used to access the *Select Output Set* dialog box. See *Select Output Set* and *Select Output Vector* dialog boxes section for more information. When *Sum Data On Nodes* is on, the grid point force and moment data from all element corners attached to that node will be summed at each node. When off, the individual grid point forces and moments will be displayed at each element corner along with the element ID next to the value in parentheses.

**Freebody Properties**

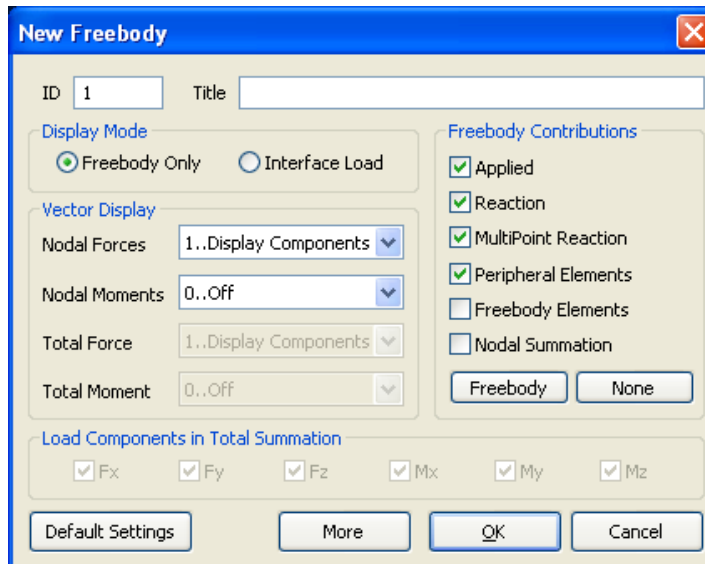
**Freebody** - This drop-down is used to select which options are currently available for use in the *Freebody Properties* section. To create a new *Freebody*

entity or edit an existing one, click the *Add Freebody* icon button to access the *Freebody Manager*.

- **Freebody Manager** - Used to create, edit, renumber, copy, and delete *Freebody* entities.



**New Freebody** - When clicked, the *New Freebody* dialog box will appear.



In this dialog box, specify an *ID* and *Title* (optional) along with some “top-level” options for the new *Freebody* entity, such as *Display Mode*, *Vector Display*, *Freebody Contributions*, and *Load Components in Total Summation*. These options will be described later in this section

**Update Title** - Highlight a *Freebody* entity in *Available Freebodies* list, then click this button to enter a new *Title*.

**Renumber** - Highlight a *Freebody* entity in *Available Freebodies* list, then click this button to change the *ID*.

**Delete** - Highlight a *Freebody* entity in *Available Freebodies* list, then click this button to delete it from the model.

**Delete All** - Deletes all *Freebody* entities in the model.

**Copy** - Highlight a *Freebody* entity in *Available Freebodies* list, then click this button to make a copy.

**None Active** - When clicked, there is no longer an “Active” *Freebody* entity.

**Default Settings** - When clicked, the following options are set:

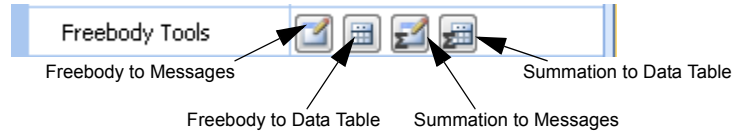
**Display Mode:** “Freebody Only”

**Vector Display:** “Nodal Forces” displayed as Components, “Nodal Moments” Off

**Freebody Contributions:** “Applied”, “Reaction”, “MultiPoint Reaction”, and “Peripheral Elements” On, “Freebody Elements” and “Nodal Summation” Off.

**More** - Click this button to create another new *Freebody* entity.

**Freebody Tools** - This section contains four icon buttons used for sending the data used in the calculations to create the freebody display to the *Messages* window or the *Data Table*.



- List Freebody to Messages Window** - Lists all contributions used to create the display of the *Freebody* entity currently selected in the *Freebody* tool to the *Messages* window. *ID* is the node ID where the *Nodal Force* and *Nodal Moment* vectors are being calculated and *Source* is the Element ID which is providing the force and moment contributions.

GRID POINT FORCE BALANCE - CSys 0							
OUTPUT SET: 1							
Node ID	Source	T1	T2	T3	R1	R2	R3
44	ELEM 311	-44.024826	4.02154732	-23.3571415	0.041980416	0.021537298	-0.44794273
44	ELEM 1112	8.39359379	-3.21601176	-130.090652	30.5459824	-10.3943205	2.38664889
44	ELEM 310	-85.4044724	13.0145521	-244.782349	-7.336308E-3	0.038634751	0.45457935
44	ELEM 302	91.0497284	-10.3473005	3.62258649	0.099446349	0.036858566	-0.082332134
44	ELEM 303	45.4979591	-6.00606871	198.784973	0.031655654	0.017965702	0.23545831
44	ELEM 1111	-15.511982	2.53328228	195.822571	-30.711729	10.2793236	-2.54641151
44	**TOTALS**	5.968559E-13	-3.26938E-12	1.136868E-11	-2.80188E-12	6.079581E-13	-1.58207E-13
45	ELEM 311	-43.9991722	4.98038197	-164.437866	0.015834594	0.0086759878	-0.54798007
45	ELEM 303	36.9290886	-3.49657011	60.1852837	0.022098033	0.0028936476	0.13732958
45	ELEM 304	25.045639	-2.34045839	160.72998	0.0017170185	-2.316987E-3	-0.18500957
45	ELEM 312	-17.9755554	0.85664654	-56.4773903	-0.039649647	-9.252648E-3	0.59566009
45	**TOTALS**	6.139089E-12	-2.31037E-13	9.094947E-13	7.579354E-14	-3.44169E-15	1.598721E-14
53	ELEM 311	49.6366692	-6.03098059	164.597015	0.017782852	0.0096285511	0.59913862
53	ELEM 318	-87.0495377	12.3153238	-209.684158	-0.028069573	0.040856786	0.25275946
53	ELEM 1112	-8.39359379	3.21601176	130.090652	-11.2208157	4.95793104	-1.00537276
53	ELEM 310	95.2380829	-10.4246721	-38.7390747	0.042675663	0.041355856	-0.32442579
53	ELEM 319	-52.7989273	4.42405462	15.777564	-0.13382339	0.035268549	-0.65703124
53	ELEM 1113	3.36730599	-3.49973702	-62.0419884	11.3222504	-5.08504057	1.13493168
53	**TOTALS**	-3.38218E-12	4.192202E-13	-1.96962E-11	3.363698E-13	-4.16556E-13	-8.52651E-14
54	ELEM 311	38.3873253	-2.97094893	23.1980057	-0.034723017	0.0034986499	0.52211398
54	ELEM 319	-40.5667152	4.06097937	-127.731537	0.061882673	0.0065702517	-0.55789727
54	ELEM 312	27.1891899	-1.7938149	125.118645	-0.067419954	0.0055851308	-0.64411157
54	ELEM 320	-25.0098	0.70378435	-20.5851192	0.040560294	-0.015654033	0.67989486
54	**TOTALS**	-2.13163E-12	2.842171E-14	9.663381E-13	-8.22606E-14	-1.77636E-15	7.21645E-15

- **List Freebody to Data Table** - Reports all contributions used to create the display of the *Freebody* entity currently selected in the *Freebody* tool to the *Data Table*. The *ID* is the node ID where the *Nodal Force* and *Nodal Moment* vectors are being calculated and *Source* is the Element ID which is providing the force and moment contributions

Output ...	CSys ID	Source	T1	T2	T3	R1	R2	R3
Freebody ID: 1								
ID: 44								
1	0	ELEM 311	-44.02483	4.021547	-23.35714	0.04198042	0.0215373	-0.4479427
1	0	ELEM 1112	8.393594	-3.216012	-130.0907	30.54598	-10.39432	2.386649
1	0	ELEM 310	-85.40447	13.01455	-244.7823	-0.007336308	0.03863475	0.4545794
1	0	ELEM 302	91.04973	-10.3473	3.622586	0.09944635	0.03685857	-0.08233213
1	0	ELEM 303	45.49796	-6.006069	198.785	0.03165565	0.0179657	0.2354583
1	0	ELEM 1111	-15.51198	2.533282	195.8226	-30.71173	10.27932	-2.546412
1	0	**TOTALS**	5.968559e-013	-3.269385e-...	1.136868e-011	-2.801884e-...	6.079581e-013	-1.582068e-...
ID: 45								
1	0	ELEM 311	-43.99917	4.980382	-164.4379	0.01583459	0.008675988	-0.5479801
1	0	ELEM 303	36.92909	-3.49657	60.18528	0.02209803	0.002893648	0.1373296
1	0	ELEM 304	25.04564	-2.340458	160.73	0.001717019	-0.002316987	-0.1850096
1	0	ELEM 312	-17.97556	0.8566465	-56.47739	-0.03964965	-0.009252648	0.5956601
1	0	**TOTALS**	6.139089e-012	-2.310374e-...	9.094947e-013	7.579354e-014	-3.441691e-...	1.598721e-014
ID: 53								

- **List Freebody Summation to Messages Window** (*Display Mode* set to “Interface Load” only) - Lists all contributions used to create the display of the *Total Summation Vector* for the *Freebody* entity currently selected in the *Freebody* tool to the *Messages* window. The “Header” above the listing contains information about the “Components included in summation”, “Contributions included in the summation”, and “location” of the summation. The (*F*) and (*P*) designators in the listings indicate contributions from *Freebody Elements (F)* and contributions from *Peripheral Elements (P)*. The *d1*, *d2*, and *d3* fields represent the distance from the X, Y, and Z location of the node (*Node ID*) to the location where the summation is taking place

Node ID	Source	T1	T2	T3	R1	R2	R3	d1	d2	d3
Peripheral Elements [x] Freebody Elements [x] Total Summation [ ]										
Summation about: 20.584, 11.8498, 77.4781										
44	ELEM 311 (F)	-44.024826	4.02154732	-23.3571415	0.041980416	0.021537298	-0.44794273	1.95088825	-0.12790675	-2.5021915
44	ELEM 1112 (P)	-44.024826	4.02154732	-23.3571415	13.0921993	155.747253	1.76657438	1.95088825	-0.12790675	-2.5021915
44	ELEM 310 (P)	-85.4044724	13.0145521	-244.782349	-7.336308E-3	0.038634751	0.45457935	1.95088825	-0.12790675	-2.5021915
44	ELEM 302 (P)	91.0497284	-10.3473005	3.62258649	0.099446349	0.036858566	-0.082332134	1.95088825	-0.12790675	-2.5021915
44	ELEM 303 (P)	45.4979591	-6.00606871	198.784973	0.031655654	0.017965702	0.23545831	1.95088825	-0.12790675	-2.5021915
44	ELEM 1111 (P)	-15.511982	2.53328228	195.822571	-30.711729	10.2793236	-2.54641151	1.95088825	-0.12790675	-2.5021915
45	ELEM 311 (F)	-43.9991722	4.98038197	-164.437866	0.015834594	0.008675987	-0.54798007	-1.63471575	0.15667625	-2.4987515
45	ELEM 303 (P)	36.9290886	-3.49657011	60.1852837	0.022098033	0.0028936476	0.13732958	-1.63471575	0.15667625	-2.4987515
45	ELEM 304 (P)	25.045639	-2.34045839	160.72998	0.0017170185	-2.316987E-3	-0.18500957	-1.63471575	0.15667625	-2.4987515
45	ELEM 312 (P)	-17.9755554	0.85664654	-56.4773903	-0.039649647	-9.252648E-3	0.59566009	-1.63471575	0.15667625	-2.4987515
53	ELEM 311 (F)	49.6366692	-6.03098059	164.597015	0.017782852	0.0096285511	0.59913862	1.58643425	-0.15282975	2.4988225
53	ELEM 318 (P)	-87.0495377	12.3153238	-209.684158	-0.028069573	0.040856786	0.25275946	1.58643425	-0.15282975	2.4988225
53	ELEM 1112 (P)	-8.39359379	3.21601176	130.090652	-11.2208157	4.95793104	-0.0573276	1.58643425	-0.15282975	2.4988225
53	ELEM 310 (P)	-85.4044724	13.0145521	-244.782349	-7.336308E-3	0.038634751	0.45457935	1.58643425	-0.15282975	2.4988225
53	ELEM 319 (P)	-52.7989273	4.42405462	15.777564	-0.13882339	0.035268549	-0.65703124	1.58643425	-0.15282975	2.4988225
53	ELEM 1113 (P)	3.36730599	-3.49973702	-62.0419884	11.3222504	-5.08504057	1.13493168	1.58643425	-0.15282975	2.4988225
54	ELEM 311 (F)	38.3873253	-2.97094893	23.1980057	0.294723017	0.0034986499	0.52211398	-1.90260675	0.12406025	2.5021205
54	ELEM 319 (P)	-40.5667152	4.06097937	-127.731537	0.061582673	0.0065702517	-0.55789727	-1.90260675	0.12406025	2.5021205
54	ELEM 312 (P)	27.1891899	-1.7938149	125.118645	-25.9458847	-944.519104	-3.25162697	-1.90260675	0.12406025	2.5021205
54	ELEM 320 (P)	-25.0098	0.70378435	-20.5851192	0.040560294	-0.015654033	0.67989486	-1.90260675	0.12406025	2.5021205
***** TOTAL SUMMATION *****										
2.3841858E-7, 5.9604645E-7, -1.335144E-5, -3.814697E-6, 2.2888184E-5, 2.3841858E-7										

- **List Freebody Summation to Data Table** (*Display Mode* set to “Interface Load” only) - Reports all the same information as *List Freebody Summation to Messages Window*, but sends it to the *Data Table*. One difference is that the “Header” information is still sent to the *Messages* window, as there is no logical place to report this information in the *Data Table*.

**Is Visible** - When On, the *Freebody* entity currently in the *Freebody* drop-down will be visible in the graphics window in all views. Display of *Freebody* entities may also be controlled via the *Freebody* tab of the *Visibility* dialog box.

**Coordinate System** - Drop-down list specifies which coordinate system should be used to display the freebody vectors. You can create a new coordinate system by using the *New Coord Sys* icon button.

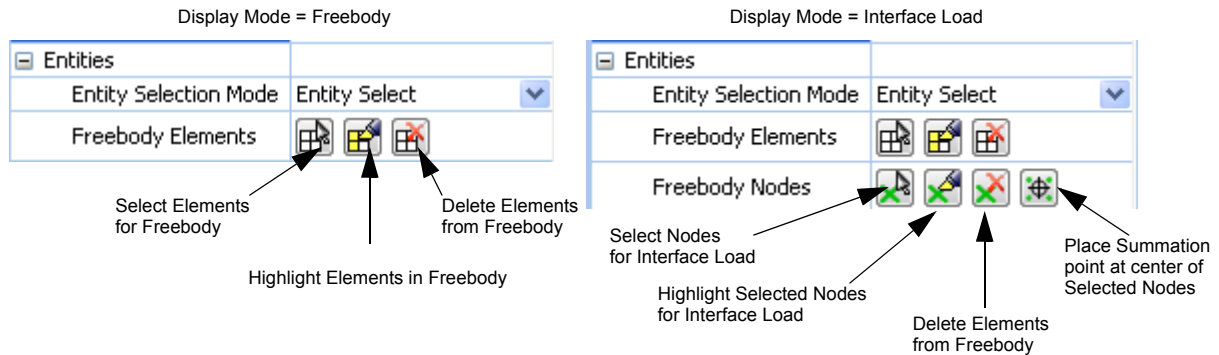
**Display Mode** - Each *Freebody* entity can be displayed in two different modes, *Freebody* or *Interface Load*.

- **Freebody** - Only *Freebody Elements* may be selected in the *Entities* section and only the vectors in the *Nodal Vector(s)* section can be displayed and controlled.
- **Interface Load** - Both *Freebody Nodes* and *Freebody Elements* must be selected in the *Entities* section and vectors in both the *Nodal Vector(s)* and the *Total Summation Vector* sections can be displayed and controlled. Additionally, a *Location* must be selected when using this option.

**Note:** Only entities which can be displayed and controlled by the selected *Display Type* will be available in the *Freebody Entity Colors* section, while setting the *View Properties* for all the different freebody vector types and nodes markers is available at all times.

**Entities** - Allows you to specify which *Freebody Elements* (*Display Mode* = “Freebody”) or *Freebody Nodes* and *Freebody Elements* (*Display Mode* = “Interface Load”) are used by a *Freebody* entity. Based on the *Entity Selection Mode*, elements and nodes may be selected for the *Freebody* entity directly or by using a pre-defined group.

- **Entity Selection Mode** - When set to *Entity Select*, elements and nodes are selected, highlighted in the graphics window, or deleted from the *Freebody* entity using the icon buttons below. An additional icon button exists for placing the summation location at the center of the selected nodes.



When set to *Group Select*, elements and nodes are determined by selecting a group from the *Group* drop-down list. If *Group* is set to “-1..Active”, then the elements will be retrieved from the Active group in the model. The *Group Manager* dialog box may also be accessed by the icon button next to the *Group* drop-down (see Section 6.4.3.1, “Group, Create/Manage...” for more information).

**Total Summation Vector** (*Display Mode* set to “Interface Load” only) - Allows you to specify the *Location* of the *Total Force Vector* and *Total Moment Vector*, along with how these vectors are displayed and what components will be summed to create these vectors.

- **Location** - Allows you to specify the location of summation for the *Total Summation Vector*. Click the icon button next to location to pick a location from the graphics window. Additionally, the individual coordinates may be entered or edited below the *Location*, when expanded.

Total Summation Vector		
Location	[20.58404, 11.84982, 77.47806]	
Coordinate System	0..Basic Rectangular	
X	20.58404	
Y	11.84982	
Z	77.47806	
Force Vector Display	Display Components	
Displayed Forces	<input checked="" type="checkbox"/> FX	<input checked="" type="checkbox"/> FY <input checked="" type="checkbox"/> FZ
Moment Vector Display	Display Resultant	
Displayed Moments	<input checked="" type="checkbox"/> MX	<input checked="" type="checkbox"/> MY <input checked="" type="checkbox"/> MZ
Summed Components		
Forces	<input checked="" type="checkbox"/> X	<input checked="" type="checkbox"/> Y <input checked="" type="checkbox"/> Z
Moments	<input checked="" type="checkbox"/> X	<input checked="" type="checkbox"/> Y <input checked="" type="checkbox"/> Z

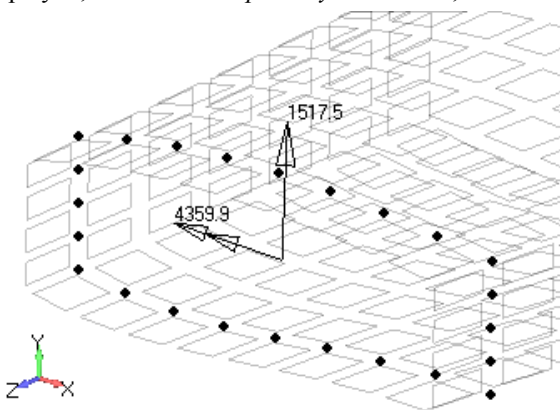
When nodes are selected in the *Entities* section, the user will be prompted to answer the following question:

Auto-locate total summation vector at center of freebody nodes (“X-coordinate”, “Y-coordinate”, “Z-coordinate” in coordinate system “ID of *Coordinate System* specified in *Freebody Properties*”)?

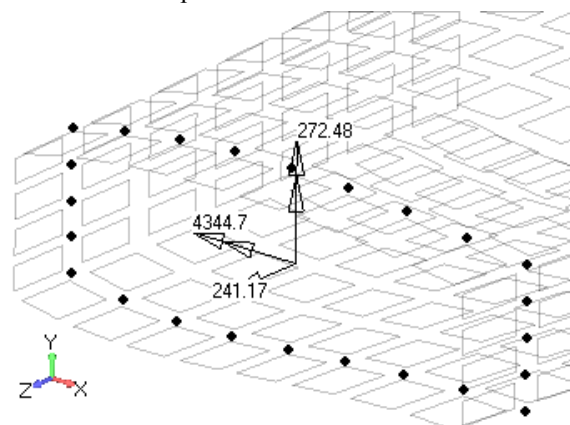
If you click *Yes*, the *Location* will be specified at the center of the selected nodes. If you click *No*, the *Location* will be at (0.0, 0.0, 0.0) or the *Location* last used by the *Freebody* entity currently in the *Freebody* tool.

- Force Vector Display** - This option controls how the “Force vector” (single arrow head) of the *Total Summation Vector* will be displayed. When set to “Off”, the force vector will be not be displayed. When set to “Display Components”, the force vector will be displayed in X, Y, and/or Z Components (individual components may be toggled on/off using the *FX*, *FY*, and *FZ* check boxes for *Displayed Forces*). When set to “Display Resultant”, the force vector will be displayed as a single resultant vector based on the components currently “on” in *Displayed Forces*.
- Moment Vector Display** - This option controls how the “Moment vector” (double arrow head) of the *Total Summation Vector* will be displayed. When set to “Off”, the moment vector will be not be displayed. When set to “Display Components”, the moment vector will be displayed in X, Y, and/or Z Components (individual components may be toggled on/off using the *MX*, *MY*, and *MZ* check boxes for *Displayed Moments*). When set to “Display Resultant”, the moment vector will be displayed as a single resultant vector based on the components currently “on” in *Displayed Moments*.
- Summed Components** - This option controls which *Force* and *Moment* components will be used to calculate the *Total Summation Vector*. Turning individual *Force* components on/off is also very likely to affect the *Moment* values, so keep that in mind.

Following figures show the *Total Summation Vector*. *Freebody Node Markers* are “On”, *Node Vector(s)* not displayed, *Element Transparency* set to 75%, and *Element Shrink View* Option is “On”.



Display Mode = Interface Load  
Total Summation Vector Force and Moment set to “Display Resultant”



Display Mode = Interface Load  
Total Summation Vector Force set to “Off” and Moment set to “Display Components”

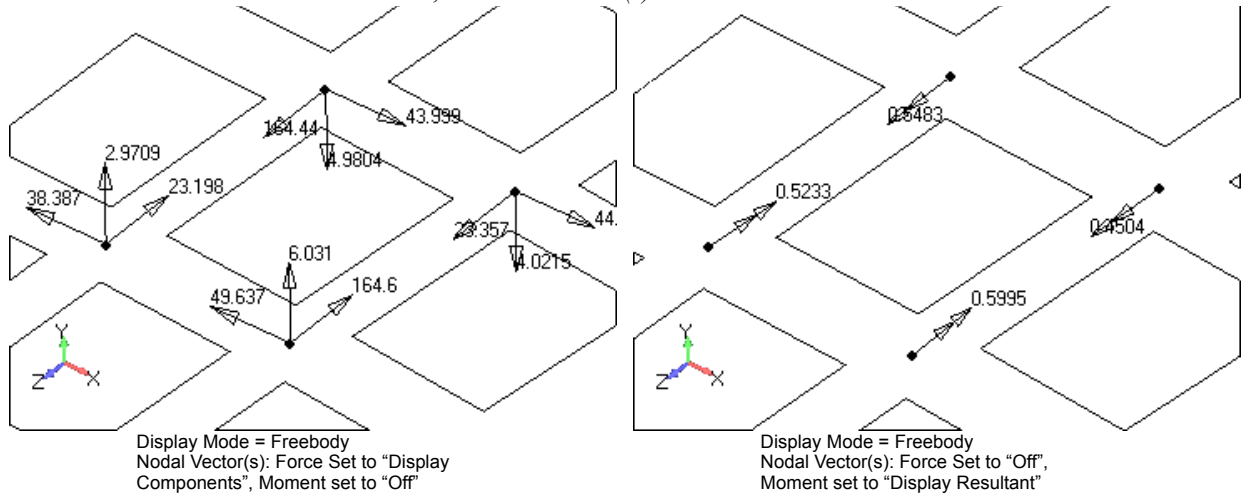
**Nodal Vector(s)** - Allows you to control how the Force and Moment vectors are displayed at each node (*Sum Data on Nodes* in *View Properties* section “On”) or each “element corner” (*Sum Data on Nodes* “Off”).

- Force Vector Display** - This option controls how the “Force vectors” (single arrow head) are displayed. When set to “Off”, the force vectors will be not be displayed. When set to “Display Components”, the force vector at each node/element corner will be displayed in X, Y, and/or Z Components (individual components may be tog-

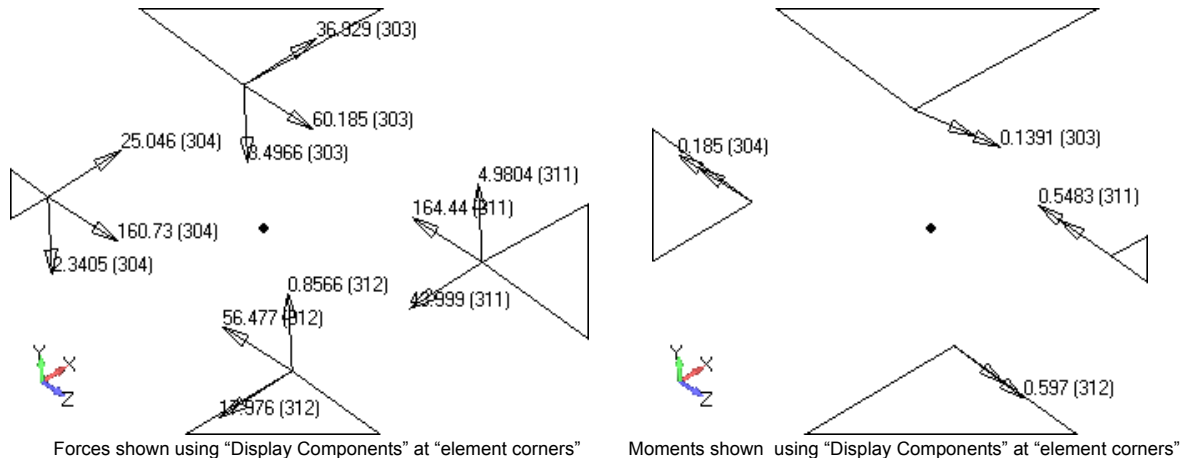
gled on/off using the  $FX$ ,  $FY$ , and  $FZ$  check boxes for *Displayed Forces*). When set to “Display Resultant”, the force vector at each node/element corner will be displayed as a single resultant vector based on the components currently “on” in *Displayed Forces*.

- Moment Vector Display** - This option controls how the “Moment vectors” (double arrow head) are displayed. When set to “Off”, the moment vectors will be not be displayed. When set to “Display Components”, the moment vector at each node/element corner will be displayed in X, Y, and/or Z Components (individual components may be toggled on/off using the  $MX$ ,  $MY$ , and  $MZ$  check boxes for *Displayed Moments*). When set to “Display Resultant”, the moment vector at each node/element corner will be displayed as a single resultant vector based on the components currently “on” in *Displayed Moments*..

When *Sum Data on Nodes* is “On”, the *Nodal Vector(s)* will be at each node:



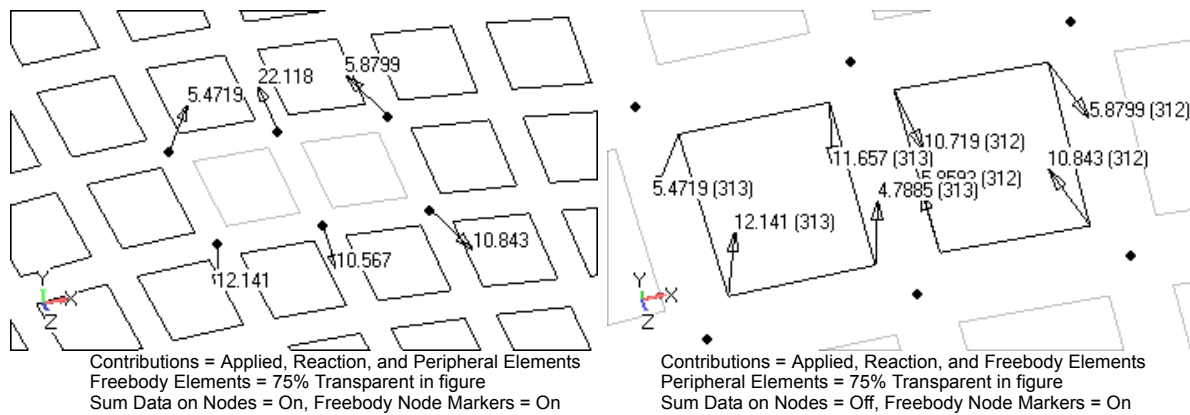
When *Sum Data on Nodes* is “Off”, the *Nodal Vector(s)* at each element corner will include the Element ID



**Freebody Contributions From** - Allows you to control the calculation of the *Freebody* entity by choosing which contributions should be included. Available contributions are from *Applied* Loads, from *Reaction* Forces and Moments at single point constraints and/or constraint equations, from the selected elements (*Freebody Elements*), and from the elements surrounding the *Freebody Elements* (*Peripheral Elements*). Toggling various options on/off can drastically alter the values and appearance of a *Freebody* entity, so be sure to have the proper contributions included for your particular needs.

- Applied** - When On, includes contributions from all loads applied to the model used to produce the results in the selected *Output Set*.
- Reaction** - When On, includes contributions from all reaction forces and moments at single point constraints in the model used to produce the results in the selected *Output Set*.
- MultiPoint Reaction** - When On, includes contributions from all reaction forces and moments from constraint equations, rigid elements, and interpolation elements in the model used to produce the results in the selected *Output Set*.

- **Peripheral Elements** - When On, includes grid point force and moment contributions from the selected *Output Set* for the elements surrounding the *Freebody Elements* selected in *Entities* section.
- **Freebody Elements** - When On, includes grid point force and moment contributions from the selected *Output Set* for the elements selected in *Entities* section.
- **Nodal Summation** - When On, includes force and moment contributions from nodal summation. Typically, these are very small numbers, unless there is a “non-balanced” force or moment in the model.



**Freebody Entity Colors** - Allows you to specify colors for *Node Marker(s)*, *Total Force Vector*, *Total Moment Vector*, *Nodal Force Vector(s)*, and/or *Nodal Moment Vector(s)* for each *Freebody* entity. Click the icon button to select a color from the *Color Palette*. These colors will only be used when the “Color Mode” for any of these items is set to “Freebody Entity Color” in the *View Properties* section of the *Freebody* tool or via the *Freebody...* options in the *View Options* dialog box, *PostProcessing* category (See Section 8.3.25, "Freebody options").

### View Properties

The *View Properties* control the visibility, style, color, and labeling for *Freebody* display. Each view in the model can have different options set in the section. When a different view is activated, the values from that view will fill the *View Properties* section.

**Show Node Markers** - controls the visibility, symbol size, and color of the “node markers” for *Freebody* entities. Having the node markers visible is a good way to visually inspect the nodes or element corners being used in the freebody calculations. The *Symbol Size* can be entered directly or increased/decreased using the “slider bar”. When *Color Mode* is set to “Freebody Entity Color”, the node markers will use the color specified for *Freebody Node Marker(s)* in the *Freebody Properties* section.

**Vector Options** - controls the *Label Mode*, *Length*, and *Label Format* of the *Freebody* vectors. *Label Mode* allows you to display *No Labels*, the *Value* of each freebody vector, or the value using exponents. For *Label Format*, the number of digits may be entered directly or increased/decreased using the “slider bar”. This will chance the number of significant digits being displayed. When *Label Format* is set to “0”, this is an “automatic mode” and FEMAP will determine the number of significant digits to display.

When *Adjust Length* is “off”, the length of each freebody vector “type” is controlled by a combination of the entered *Length* value and the *Factor* value entered for the *Freebody Total Force*, *Freebody Total Moment*, *Freebody Nodal Force*, and *Freebody Nodal Moment* view options.

When *Adjust Length* is “on”, the length of the freebody vectors will be adjusted based on the vector’s value (i.e., larger values = longer vectors). The *Units/Length* value is an additional parameter used to control the length of the vectors when in this mode. Essentially, the *Units/Length* value is used in the following manner:

If *Units/Length* value is 250, then a freebody vector value of 500 would be shown using a length of “2\*Factor” on the screen. For the same freebody vector value of 500, entering a *Units/Length* value of 100 would display the vector using a length of “5\*Factor” on the screen.

**Min Vector Magnitude** - allows you to set a tolerance below which the vectors are not displayed. Using the default value of 1.0E-8, this option will basically remove vectors from the display that are not zero just due to numerical round-off. The value can also be used as a cut-off value, so if it is set to 10, only vector values above 10 will be displayed.

**Total Force Vector/Total Moment Vector** - controls the *Vector Style*, *Color Mode*, and *Factor* for the *Total Summation* Force and Moment vectors. The *Total Summation* vectors are only visible when the *Display Mode* of a *Freebody* entity is set to “Interface Load”.

When *Vector Style* is set to *Arrow* or *Center Arrow*, the vectors will be displayed as lines. When set to *Solid Arrow* or *Center Solid Arrow*, the vectors will be “thicker, filled-in solids”. *Factor* is an additional scale factor which can be entered to change the size of the selected vector type.

When *Color Mode* is set to *Freebody Entity Color*, the “Freebody Entity Colors” specified for each *Freebody* entity in the *Freebody* tool is used. This allows multiple *Freebody* entities to be displayed at one time using unique colors for clarity. *RGB Color* uses Red to display the X component, Green for the Y component, and Blue for the Z component of each vector.

**Nodal Force Vector/Nodal Moment Vector** - offers the same options as *Freebody Total Force/Freebody Total Moment*, but these options control the *Nodal Vector(s)*. One difference is in *Color Mode*, where an additional option, *Source Color* exists. When set to *Source Color*, this selected vector type uses the color of the “source” elements, the color of the load for *Applied* loads, and/or the color of the constraint for *Reaction* forces and moments. When the *Sum Data on Nodes* option is “on” and *Source Color* is selected, the *View Color* will be used.

## Geometry

- Enhanced *Geometry*, *Solid*, *Embed* to allow embedding of multiple solids into the base solid all at once.

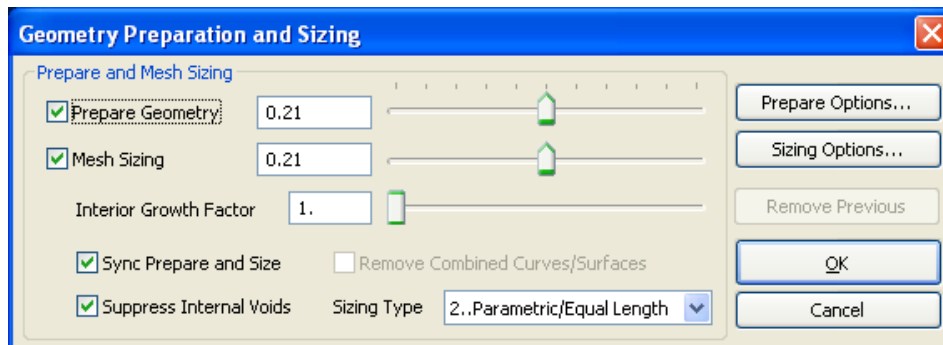
## Meshing

- Enhanced “Suppress Short Edges” option in *Mesh*, *Mesh Sizing*, *Size on Surface* and *Mesh*, *Mesh Sizing*, *Size on Solid* to be a percentage of Mesh Size instead of a percentage of “average curve length” on selected geometry.
- Added *Mesh*, *Geometry Preparation* command

This command uses a set of parameters to find situations in geometry which typically result in poor element quality, then uses a combination of automatic curve/surface splitting, creation of Combined Curves/Boundary Surfaces, and feature suppression to likely improve mesh quality. In addition, this command will “prepare” some parts to a degree which will allow FEMAP to successfully mesh the part.

**Note:** If FEMAP is successful when meshing a solid with acceptable mesh quality for your application, then using “Mesh, Geometry Preparation” is probably not necessary. Also, please be aware when using this process, it is quite common for certain small features to be ignored or removed completely.

In most cases, this automatic process will be all that is needed to produce a good quality mesh. However, even if it cannot fully automatically produce an acceptable mesh, it will still provide a good starting point for using the other interactive geometry cleanup tools, and greatly reduce the amount of work required.



**Note:** It is recommended to use the “Mesh, Geometry Preparation” command BEFORE manually creating additional Combined Curves /Boundary Surfaces for meshing purposes.

Surfaces and Curves which have loads or boundary conditions applied will be ignored.

By default, the command goes through two steps, *Prepare Geometry* and *Mesh Sizing*. You can choose to skip either step by simply un-checking the box next to *Prepare Geometry* or *Mesh Sizing*. The value for size shown for

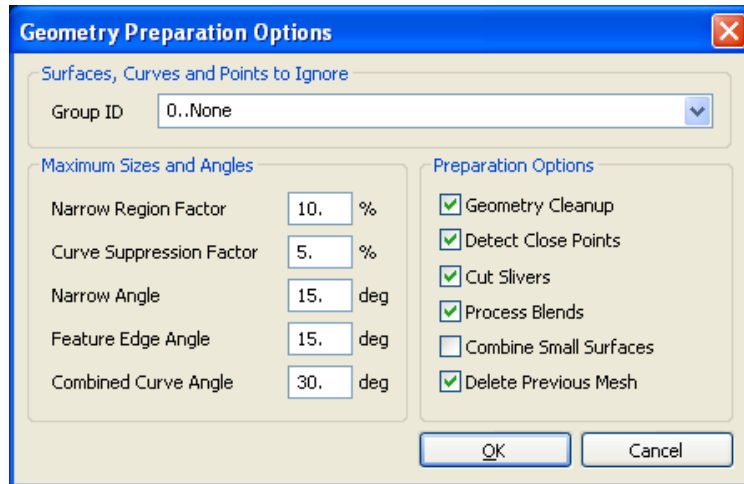
both *Prepare Geometry* and *Mesh Sizing* is the “Default Mesh Size” calculated by FEMAP (uses the same algorithm as "Mesh, Geometry, Solids").

### Prepare Geometry

The value for *Prepare Geometry* is simply used as a baseline value for the various *Prepare Options*. Therefore, it is typically a good idea to change the *Prepare Geometry* value instead of the individual *Prepare Options* values.

### Prepare Options button

Opens the *Geometry Preparation Options* dialog box. In general, the "Prepare Geometry" process has been developed to function most effectively using the default values in the "Maximum Sizes and Angles" section and all of the "Preparation Options" set to "on", except "Combine Small Surfaces". These values should only be changed and/or options turned off if you run into a problem.

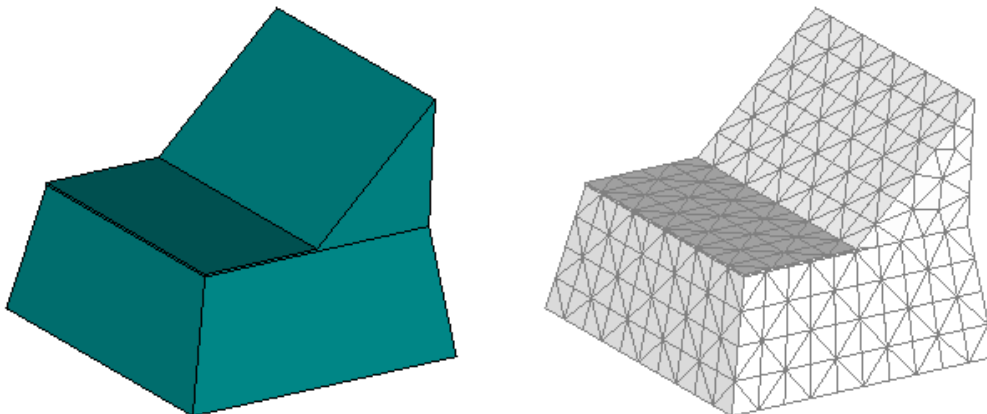


**Surfaces, Curves and Points to Ignore** - allows you to choose a group containing Surfaces, Curves, and/or Points to exclude from the "Prepare Geometry" process.

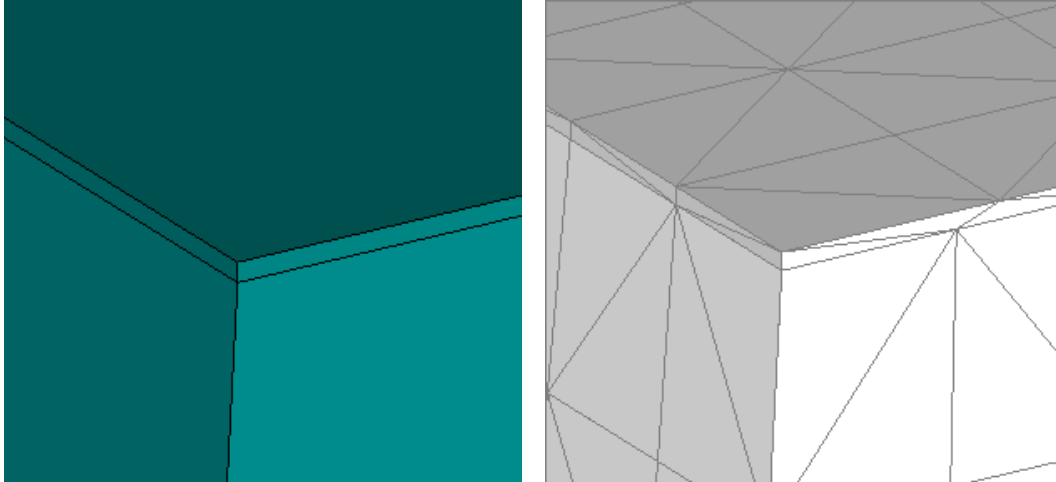
**Maximum Sizes and Angles** - allows you to specify “percentage of prepare size” and angle tolerances to help control the “Prepare Geometry” process. There are 5 values to set:

- *Narrow Region Factor* (default = 10%) - If distance between two locations on a region of a surface is less than n% of "Prepare Size", the surface will be split. The locations where distance is checked are automatically determined by faceting the curves based on a percentage of "Prepare Size" (the faceting percentage cannot be changed by the user).

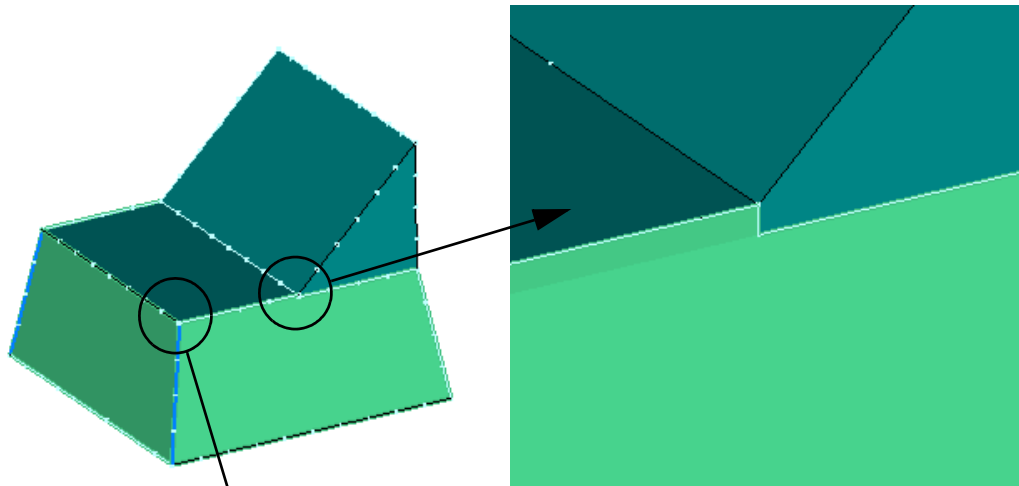
For example, this simple part has a “narrow region”. Without going through the “Prepare Geometry” process, the worst elements in the resulting mesh have a “Tet Collapse Ratio” of 16.437 and a “Jacobian” of 0.8386167..



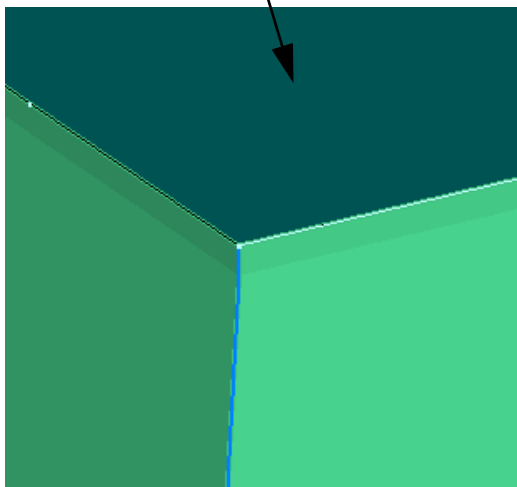
Zoomed-in view of “narrow region” at the corner of the part:



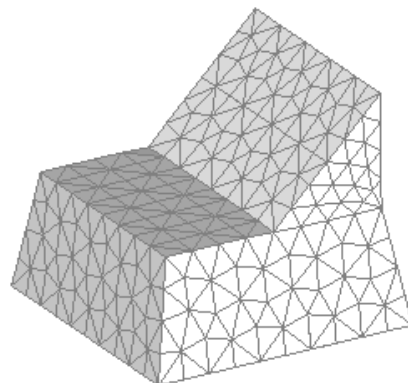
After the “Prepare Geometry” process using the defaults, the “narrow region” has been split from the original surface, then combined with surfaces from the “base”. Also, two short curves at the split locations have been suppressed. Finally, 2 Combined Curves have been created to allow larger elements in an area that used to be restricted by the “narrow region” Worst elements now have “Tet Collapse Ratio” of 5.67 and “Jacobian” of 0.694



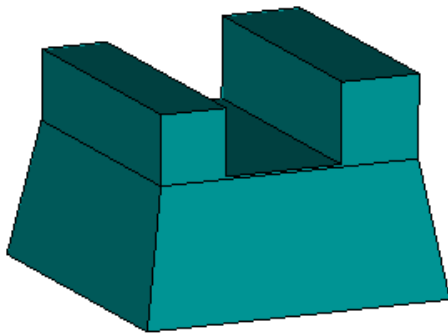
Close-up of “narrow region” split  
“Split Curve” is suppressed



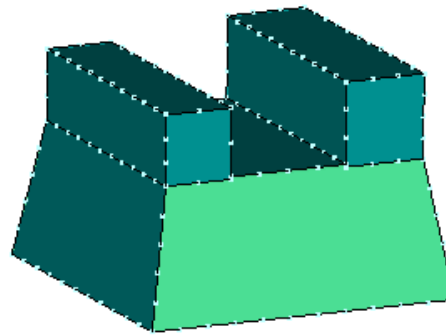
Close-up of Combined Curve and  
Boundary Surfaces at Corner



A surface which has a “narrow region” that connects two other larger regions is also a good candidate for splitting, then combination to other surfaces. A surface may be split multiple times if needed to isolate the “narrow region”.



Before “Prepare Geometry” process  
When meshed, Worst Tet Collapse = 15.72  
Worst Jacobian = 0.793



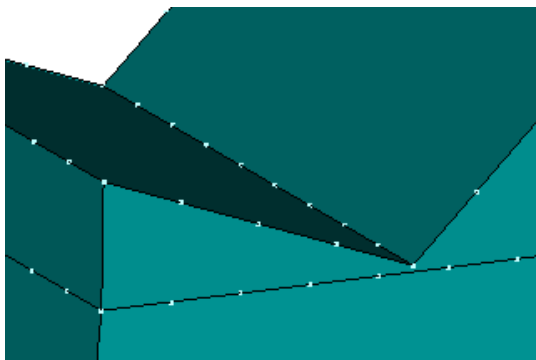
After “Prepare Geometry” process  
When meshed, Worst Tet Collapse = 4.642  
Worst Jacobian = 0.458

- *Curve Suppression Factor* (default = 5%) - If curve is less than n% of "Prepare Size", it will be suppressed. Also, if all curves on a surface are less than n% of "Prepare Size", the surface will also be suppressed and the surface "collapsed to a single point".
- *Narrow Angle* (default = 15 degrees) - If a surface has a narrow region, but the tangent vectors of the bounding curves at the locations where the "narrowness" occurs are not within this value, then the split will not occur. See description of "Detect Close Points" in the "Preparation Options" section for some exceptions.
- *Feature Edge Angle* (default = 15 degrees) - If angle of a feature is more than this value, then the "Prepare Geometry" process will look for other surfaces which are not above this threshold to combine with surfaces which will benefit from being combined. If no other suitable surface can be located, then surfaces which are over this value may still be combined when needed.
- *Combined Curve Angle* (default = 30 degrees) - If angle is larger than this value, curves will not be combined. Unlike combining curves via the Meshing Toolbox, which has the option to create boundary surfaces while creating combined curves, this command only deals with combined curves. This is because the surfaces to combine have already been determined earlier in the "Prepare Geometry" process.

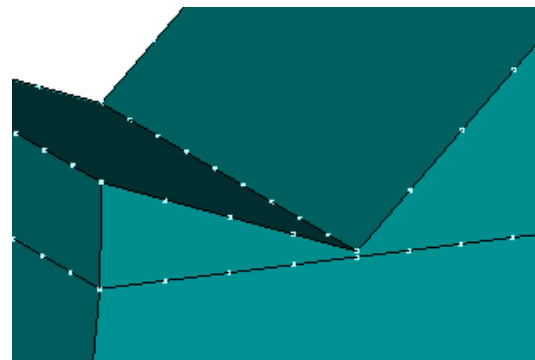
**Preparation Options** - allows you to toggle 6 different options of the “Prepare Geometry” process on/off.

- *Geometry Cleanup* - When on, applies a subset of options found in the "Geometry, Solid, Cleanup" command to attempt cleanup of any numerical issues which may exist in the geometry. Many times, these types of issues arise during translation of the geometry.
- *Detect Close Points* - When on, detects when a point between two bounding curves of a surface is very close to a location on a third bounding curve on the surface (i.e., "knife edge"), then splits the surface at these locations and suppresses the "split curve". Using the default values for "Narrow Region Factor" and "Narrow Angle", this case would be ignored.

For example, the angles of the curves at the “narrow region” location on the part below are not within the “Narrow Angle” tolerance value. If “Detect Close Points” is “off”, this portion of the geometry will not be “prepared”..



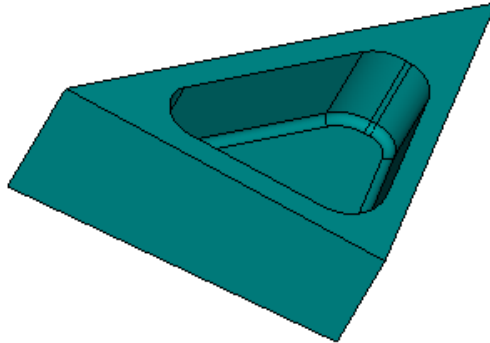
“Detect Close Points” Set to “Off”  
Nothing split or suppressed



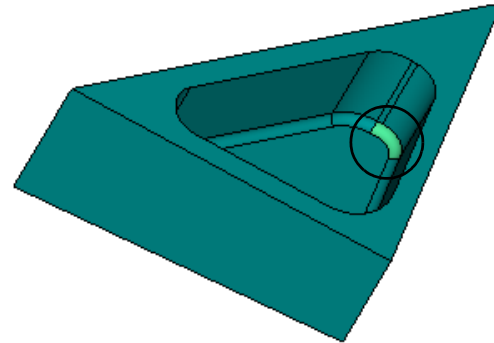
“Detect Close Points” Set to “On”  
Surface split and “split line” suppressed

- *Cut Slivers* - When on, will review all surfaces considered "slivers" and determine if they should be "cut" again to allow for more effective combining with adjacent surfaces.
- *Process Blends* - When on, attempts to combine small fillet surfaces in a "fillet chain" to larger surfaces in the "fillet chain" to create Boundary Surfaces in hopes of creating a better surface mesh.

For example, this simple part has a "fillet chain" with a small surface near larger surfaces:



Before "Prepare Geometry" process



After "Prepare Geometry" process  
Smaller fillet surface combined to  
larger surface in "Fillet chain"

- *Combine Small Surfaces* - In many cases, suppressing very small surfaces entirely is a better option, therefore this option is off by default. When on, attempts to combine very small surfaces to surrounding surfaces instead of suppressing them.
- *Delete Previous Mesh* - When on, deletes any existing surface and/or solid mesh currently on the solid which was selected for the "Prepare Geometry" process.

### Mesh Sizing and Sizing Options button

The value for *Mesh Sizing* and the options found when the *Sizing Options* button is pressed are mostly the same as options found in the "Mesh, Mesh Control, Size on Solid" command (see Section 5.1.2.4, "Mesh, Mesh Control, Size on Surface..."). The one exception is that *Max Size of Small Feature* is entered as a percentage of the *Mesh Sizing* value entered in the *Geometry Preparation and Sizing* dialog box instead of being entered as an actual value.

### Interior Growth Factor

Same as *Growth Factor* in the "Surfaced Interior Mesh Growth" section of the "Mesh, Mesh Control, Surface" and "Mesh, Mesh Control, Solid" (see Section 5.1.2.4, "Mesh, Mesh Control, Size on Surface..."). Value (1.0 by default) may be changed using the slider bar or by manually typing in a value (must be between 1.0 and 10.0).

### Sync Prepare and Size

When on (default), the values for *Prepare Geometry* and *Mesh Sizing* will change at the same time to the same value when the slider is moved left or right or the value is entered manually into either field.

### Suppress Internal Voids

When on (default), suppress any volumes which are completely contained within the solid (for example, a cube with an internal sphere).

**Note:** There is no "limiting size" on an internal void, so if you have a mostly hollow structure (i.e., pressure vessel or fully enclosed tank), and this option is on, the entire "internal void" will be suppressed.

### Remove Combined Curves/Surfaces

When on (default after "Mesh, Geometry Preparation" command has been used once), will remove Combined Curves/Boundary Surfaces on the geometry currently selected before starting the "Prepare Geometry" process.

### Sizing Type

Same as "Sizing Type" of the "Mesh, Mesh Control, Surface" and "Mesh, Mesh Control, Solid" commands (see Section 5.1.2.4, "Mesh, Mesh Control, Size on Surface..."). "2..Parametric/Equal Length" is the default.

### Remove Previous button

Removes all Combined Curves/Boundary Surfaces, along with any "surface splits" created by the most recent use of the "Mesh, Geometry Preparation" command on the selected geometry. Exits the command after completion.

- Added *Improve Collapsed Tets* option to the *Solid Automeshing Options* dialog box of the *Mesh, Geometry, Solid* command, which is accessed by click the *Options* button.

When this option is “on” (default), the mesher will locate elements with a “Tet Collapse Ratio” higher than the specified value (default is 100), then attempt to improve the mesh quality by moving “internal nodes” to new locations. Once the nodes have been moved, the new “triangular seed mesh” is sent through the tet mesher again.

- Renamed the *Length Based Sizing* option in the *Mesh, Mesh Control, Size on Surface* and *Mesh, Mesh Control, Size on Solid* commands to *Sizing Type* and added the “2..Parametric/Equal Length” option, which is also now the default.

When this option is set to “0..Parametric”, all sizing along curves is done in the parametric space of the curves. In many cases this is desirable resulting in a finer mesh in areas of high curvature. In some cases however - with unstitched geometry, or geometry that has curves with unusual parameterization - “1..Equal Length” spacing along the curves will yield much better results. Especially when dealing with unstitched geometry, “equal length” spacing will produce meshes with matching nodal locations far more reliably than “parametric” spacing. The default is “2..Parametric/Equal Length”, which sizes all curves using the “Parametric” option, then determines an “average distance” between each of the “mesh locations” on each curve. If the distance between any of the mesh locations is more than 1% different than the “average distance”, then that curve is resized using “Equal Length” sizing.

- Improved the *Surface Interior Mesh Growth* option in the *Mesh, Mesh Control, Size on Surface* and *Mesh, Mesh Control, Size on Solid* commands to allow mapped meshing on surface where it was applied. Previously, mapped meshing was not available on these surfaces.
- Improved *Mesh, Mesh Control, Custom Size Along Curve* command to remove the limitation on number of custom points which can be assigned.

## Elements

- Updated the *Spring/Damper* element to use the *Type*, either *CBUSH* or *Other (NASTRAN CROD/CVISC)*, specified on the *Property* referenced by the element to determine if a CBUSH or a combination of CROD and/or CVISC elements will be exported to Nastran. Formally, this was done by setting the element formulation. Also, the *Define Spring/Damper Element* dialog box will now change to show the appropriate inputs based on the *Type* of the referenced *Property*. Finally, *CBUSH* elements will now use a circular symbol for display, while *Other (NASTRAN CROD/CVISC)* elements will use a rectangular symbol.

## Materials

- Added Mullins Effect (MATHEM) and Viscoelastic Effect (MATHEV) support for NX Nastran Hyperelastic materials for SOL 601/701 in *Other Types*. The additional options are accessed using the *Next* button when defining Mooney-Rivlin, Hyperfoam, Ogden, Arruda-Boyce, or Sussman-Bathe types.
- Added Viscoelastic Material (MATVE) in *Other Types* for NX Nastran SOL 601.
- Added NITONAL material type in *Other Types* for NEi Nastran..

## Properties

- Added *Mean Dilatational Formulation* option to *Plane Strain* Property. This option is for NX Nastran only and is for properties which do not reference a hyperelastic material for Plane Strain or Plane Stress Elements. The formulation of the elements also must be set to “1..CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8” (Plane Strain) or “2..CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8” (Plane Stress) in order to export this property type. The “Mean Dilatational Formulation” switch on the property may be used for nearly incompressible materials, but is ignored for SOL 601. Also, Nonstructural mass/are is ignored for SOL 601.
- Added *Type* in *Spring/Damper* Property to define if the elements referencing this Property are CBUSH elements or a combination of CROD and/or CVISC elements when exporting to Nastran.
- Added support for NEi Nastran Failure Theories, Max Stress (STRESS), NASA LaRC (LAERC02), Puck PCP (PUCK), and Multicontinuum (MCT), on *Laminate* Property.

## Aeroelasticity - New for 10.3!

The commands under the *Model, Aeroelasticity* menu are used to create entities required to perform Static Aeroelastic analysis (SOL 144) and Aerodynamic Flutter analysis (SOL 145) with Nastran solvers. An underlying finite element model is also needed to properly run an aeroelastic analysis. Typically, this underlying “structural model” consists of only beam and/or shell elements.

There are 4 different types of aeroelastic entities supported for Nastran:

- Aero Panel/Body
- Aero Property
- Aero Splines
- Aero Control Surfaces

The various “Aero entities” interact with one another in several ways. Every *Aero Panel/Body* is required to have an appropriate *Aero Property* assigned. Several *Aero Panels/Bodies* may reference the same *Aero Property*.

Next, each *Aero Spline* must reference an *Aero Panel/Body* and a group of “structural” nodes in the model. The *Aero Spline* entities connect the “aeroelastic model” to the underlying “structural model”. Any number of “aerodynamic boxes” (Aero Mesh) may be selected from the referenced *Aero Panel/Body*.

Finally, each *Aero Control Surface* needs to reference at least one “aerodynamic box” (Aero Mesh) on an *Aero Panel/Body* set to “Aero Panel”.

Once all the Aero entities have been defined, additional options for Static Aeroelasticity and Aerodynamic Flutter will need to be set using the Analysis Set Manager.

### Model, Aeroelasticity, Panel/Body...



...creates an Aero Panel or Aero Body (Slender Body and/or Interference Body). The dialog box changes depending on what is specified for *Aero Body Type*. When *Aero Body Type* is set to “0..Aero Panel (CAERO1)”, then FEMAP is making an “Aero Panel”, which will be written to Nastran as a CAERO1 entry. When *Aero Body Type* is set to “1..Aero Body (CAERO2)”, then FEMAP is making a “Slender/Interference Body”, which will be written to Nastran as a CAERO2 entry. Each *Aero Body Type* contains different inputs, will be discussed in greater detail later.

The *ID*, *Title*, *Color*, *Layer*, and *Property* fields are common to both *Aero Body Types*, as well as the *Orientation CSys* and *IGID* fields in the *Options* section.

**Note:** The ID value for *Aero Panel* will increment by 1000 automatically. This is due to the fact that each *Aero Panel/Body* has a *Mesh Control* section which defines the “Aero Mesh” (*Number Chord \* Number Span* for an “Aero Panel”, *Number of Body Elements* for “Aero Slender Body”) and each “Aero Element” must have a unique ID. FEMAP numbers the “Aero Mesh” using the *Aero Panel/Body* ID as a prefix. For example, an “Aero Panel” with ID of 2000 has *Number Chord* set to 10 and *Number Span* set to 5 for a total of 50 “Aero Elements”. They are numbered 2000 to 2049 for this Aero Panel.

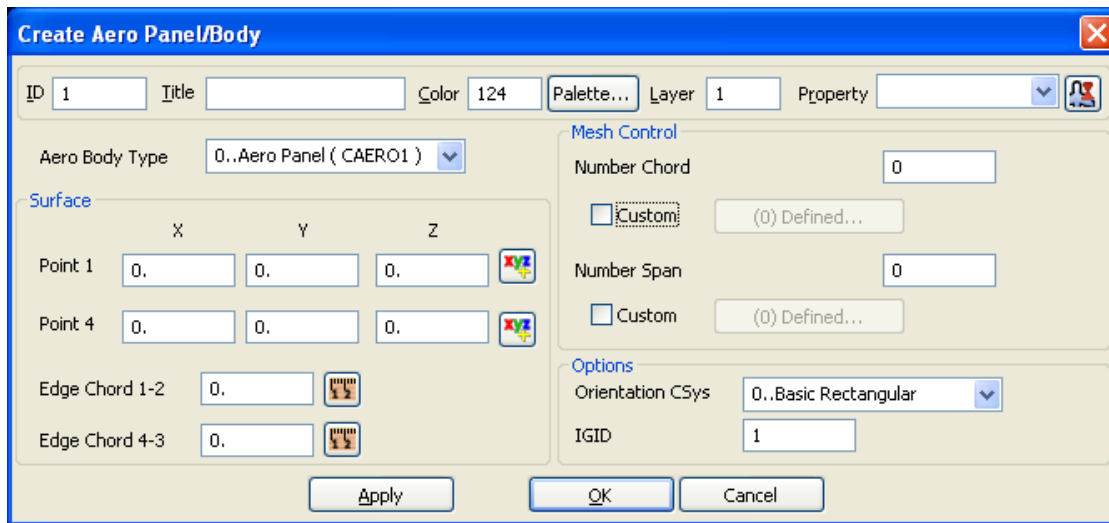
Select an existing *Aero Property* from the *Property* drop-down. The *Type* on the *Aero Property* must correspond to the *Aero Body Type* on *Aero Panel/Body* (i.e., *Type* must be “Aero Body (PAERO2)” on the *Aero Property* used by an *Aero Panel/Body* with *Aero Body Type* set to “1..Aero Body (CAERO2)”). If an *Aero Property* does not currently exist, click the *Create Aero Property* icon button to create one “on-the fly”.

*Orientation CSys* is used to orient the locations of *Point 1* and *Point 4* (Aero Panel Only) and is written to the CP field of the CAEROi entry, while *IGID* designates the “Interference Group ID” and writes out the IGID field to CAEROi entry (aerodynamic elements with different IGIDs are uncoupled).

**Note:** To change the IGID value on multiple *Aero Panel/Body* entities all at once, use the *Modify, Update Other, Aero Interference Group* command.

## Aero Body Type = "0..Aero Panel (CAERO1)"

This *Aero Body Type* will create an "Aero Panel". The values represent two "leading edge" locations and the length of two "side chords". The number of divisions for "chord" and "span" are also entered to define the "Aero Mesh". Typically, the panel will have 4 corners, but can have 3 by setting the length of one "side chord" to 0.0.



### Surface

**Point 1** - XYZ values of the first "leading edge" location in the *Orientation CSys*. Enter values directly as text, click in *X*, *Y*, or *Z* field and select a location from the graphics window, or use the *Specify Location* icon button. Writes values to the X1, Y1, and Z1 fields on the CAERO1 entry.

**Point 4** - XYZ values of the other "leading edge" location in the *Orientation CSys*. Same options as *Point 1*, except writes values to the X4, Y4, and Z4 fields on the CAERO1 entry.

**Edge Chord 1-2** - Specifies the "side chord length" from "Point 1" to "Point 2" in the X-direction of the *Orientation CSys*. Writes value to X12 field of CAERO1 entry.

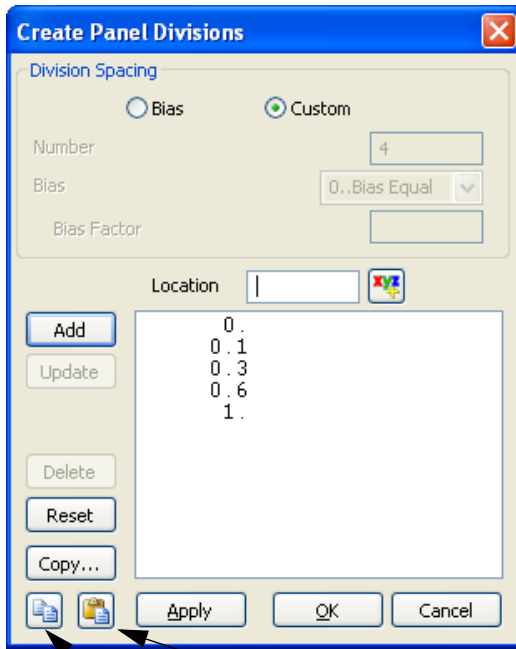
**Edge Chord 4-3** - Specifies the "side chord length" from "Point 4" to "Point 3" in the X-direction of the *Orientation CSys*. Writes value to X43 field of CAERO1 entry.

### Mesh Control

**Number Chord** - Specifies the number of evenly spaced divisions used to represent the "Aero Mesh" (Aero Boxes) from "Point 1" to "Point 2" ("Point 4" to "Point 3") on the Aero Panel. Writes value to NCHORD field on the CAERO1 entry.

**Number Span** - Specifies the number of evenly spaced divisions used to represent the "Aero Mesh" (Aero Boxes) from "Point 1" to "Point 4" ("Point 2" to "Point 3") on the Aero Panel. Writes value to NSPAN field on the CAERO1 entry.

**Custom option** - Alternatively, to specify a custom set of "division points" for the "Chord" or "Span", turn on the *Custom* option, then click the *(0) Defined* button to open the *Create Panel Divisions* dialog box.



Copy to Clipboard      Paste from Clipboard

When *Division Spacing* is set to “Custom”, enter text values directly into the *Location* field or click the *Specify Location* icon button to select from the graphics window. Values MUST be between 0.0 and 1.0 and the list MUST include 0.0 and 1.0 to create a valid aero mesh.

Click the *Add* button to add the current value in *Location* to the list of values.

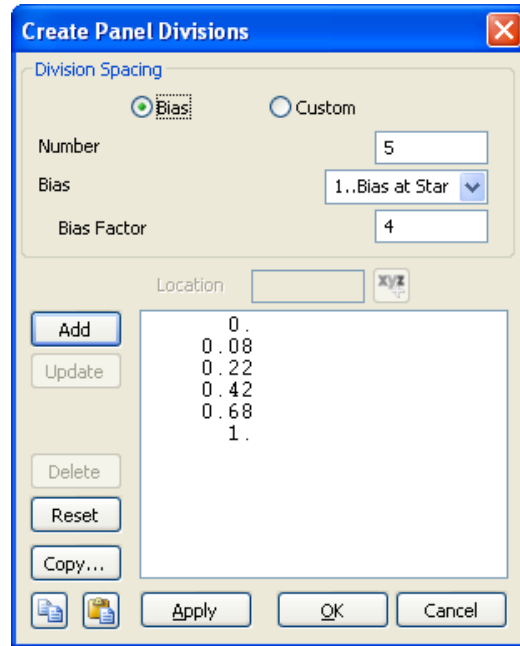
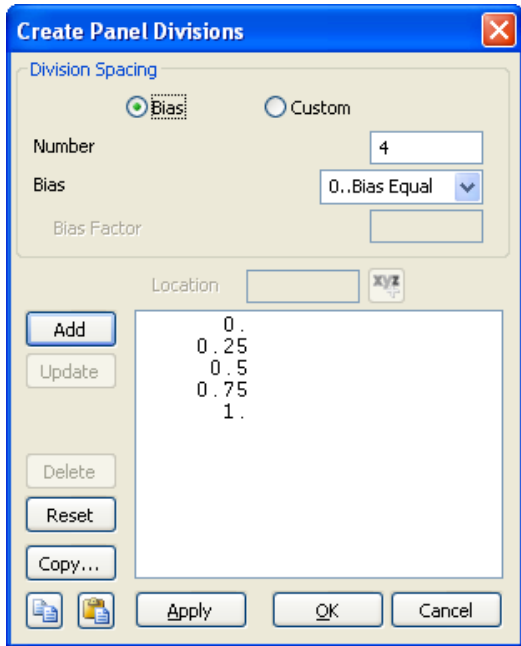
Once a value is in the list, it can be highlighted and the location will be shown in the graphics window. Click *Update* button to change a highlighted value to the value currently in the *Location* field or click *Delete* button to remove the value from the list. The *Reset* button can be used to clear all values from the list.

The *Copy* button can be used to copy the “custom” panel division list from another *Aero Panel/Body* in the current model.

The *Copy to Clipboard* and *Paste from Clipboard* icon buttons can be used to copy/paste the current list of values to/from the clipboard.

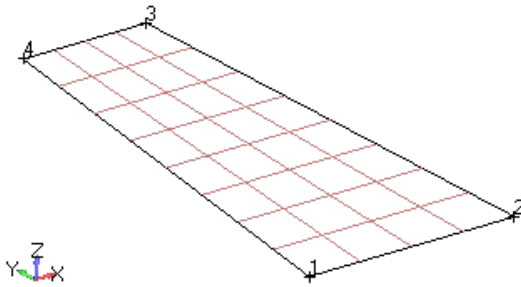
The *Apply* button will show the current divisions on the Aero Panel in the graphics window.

When *Division Spacing* is set to “Bias”, enter a *Number*, choose a type of *Bias* (“Bias Equal”, “Bias at Start”, “Bias at End”, “Bias at Center”, or “Bias at Both Ends”) and enter a *Bias Factor* (if needed). Once these parameters have been specified, click the *Add* button in the listing section to add values.

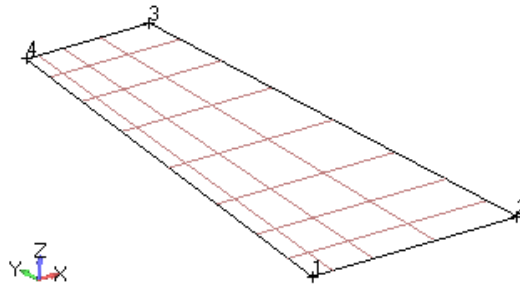


When “Custom” is used for *Number Chord*, an AEFACCT entry will be written to Nastran and the ID of the AEFACCT will be referenced by the LCHORD field on the CAERO1. When “Custom” is used for *Number Span*, the AEFACCT is referenced by the LSPAN field of the CAERO1.

Some example Aero Panels - *Point 1* at (0.0, 0.0, 0.0), *Point 4* at (2.0, 10.0, 0.0), *Edge Chord 1-2* = 5, *Edge Chord 4-3* = 3, *Orientation CSys* = Basic Rectangular:



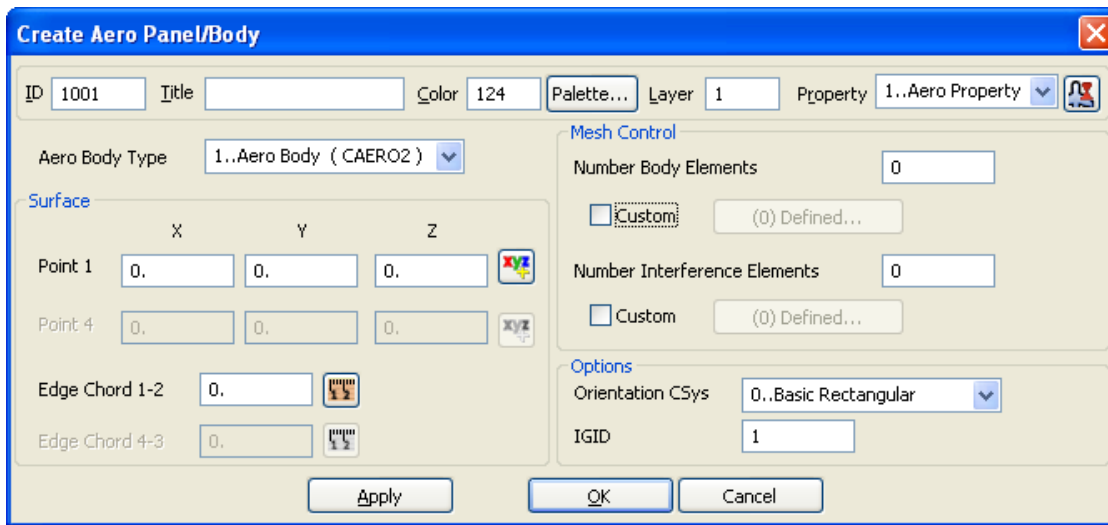
Number Chord = 4, Number Span = 8



Number Chord (Custom) = 4, Bias at Start, BF = 4  
Number Span (Custom) = 8, Bias at Both Ends, BF = 2

### Aero Body Type = "1..Aero Body (CAERO2)"

This *Aero Body Type* will create an "Aero Slender/Interference Body". The values required are a location for the start of the body and the length of the body. The number of divisions for "Slender Body" is also entered to define the "Aero Mesh". Additionally, a value for the number "Interference Body" divisions needs to be entered.



Only the divisions along the length of the "Slender/Interference Body" are specified using this dialog box. The values for the "Slender Body Radius", "Interference Body Radius", and the "Theta Arrays" are defined using the *Aero Property* with *Type* set to "Aero Body (PAERO2)".

### Surface

**Point 1** - XYZ values of the first start of the Slender/Interference Body in the *Orientation CSys*. Enter values directly as text, click in *X*, *Y*, or *Z* field and select a location from the graphics window, or use the *Specify Location* icon button. Writes values to the X1, Y1, and Z1 fields on the CAERO2 entry.

**Edge Chord 1-2** - Specifies the "side chord length" from "Point 1" to "Point 2" in the X-direction of the *Orientation CSys*. Writes value to X12 field of CAERO2 entry.

### Mesh Control

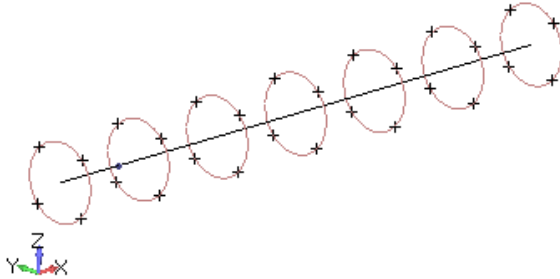
**Number Body Elements** - Specifies the number of evenly spaced divisions used to represent the "Aero Mesh" (Aero Boxes) on the "Slender Body" from "Point 1" to "Point 2" on the "Slender Body". Writes value to NSB field on the CAERO2 entry.

**Number Interference Elements** - Specifies the number of evenly spaced divisions used to represent the "Interference Body" from "Point 1" to "Point 2". Writes value to NINT field on the CAERO2 entry.

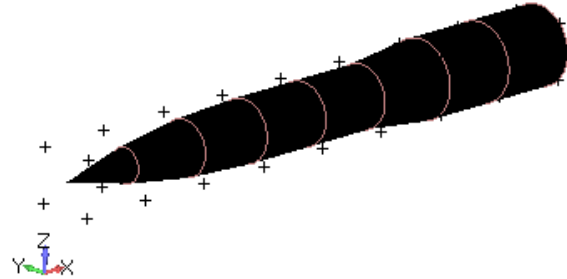
**Custom option** - Alternatively, to specify a custom set of "division points" along the length of the "Slender Body" or "Interference Body", turn on the *Custom* option, then click the *(0) Defined* button to open the *Create Panel Divi-*

sions dialog box. For more information on using the *Create Panel Divisions* dialog box, see the “Custom option” portion of the *Aero Body Type* = “0..Aero Panel (CAERO1)” section above.

When “Custom” is used for *Number Body Elements*, an AEFACCT entry will be written to Nastran and the ID of the AEFACCT will be referenced by the LSB field on the CAERO2. When “Custom” is used for *Number Interference Elements*, the AEFACCT is referenced by the LINT field of the CAERO2.



Number of Body Elements = 6  
 Number of Interference Elements = 6  
 Reference Radius on Aero Property = 2.5  
 Shown in Wireframe Display Mode



Number of Body Elements = 8  
 Number of Interference Elements = 8  
 Reference Radius on Aero Property = 2.5  
 Slender Body Division Radius list on Aero Property  
 0, 1.111, 1.778, 2, 2, 2.5, 2.5, 2.5

## Model, Aeroelasticity, Property...



...creates an Aero Property for an Aero Panel or an Aero Body (Slender Body and/or Interference Body). The dialog box changes depending on what is specified for *Type*. When *Type* is set to “Aero Panel (PAERO1)”, then FEMAP is making a “Aero Panel” property, which will be written to Nastran as a PAERO1 entry. Other than *ID*, *Title*, *Color*, and *Layer*, there is nothing else to enter for an “Aero Panel” property.

The dialog box titled "Create Aero Property" has the following fields and options:

- ID: 1000
- Title: Aero Property
- Color: 110
- Palette... button
- Layer: 1
- Type:  Aero Panel (PAERO1)  Aero Body (PAERO2)

When *Type* is set to “Aero Body (PAERO2)”, then FEMAP is making a “Slender/Interference Body” property, which will be written to Nastran as a PAERO2 entry. Along with the *ID*, *Title*, *Color*, and *Layer* fields, there are several other values which may be entered and effect the display and behavior of all *Aero Body* entities which reference a particular *Aero Property*. These additional options are described in greater detail below.

### Common

**Reference Radius** - Is the reference half-width of “Slender Body” and the half-width of the constant width “Interference Tube. Writes the WIDTH entry to the PAERO2 entry.

**Aspect Ratio (h/w)** - Aspect Ratio of interference tube (height/width). Writes the AR field to the PAERO2 entry.

### Slender Body Properties

**Orientation** - Specifies the type of motion allowed for bodies. The selected direction (Z, Y, or ZY) is in the specified “aerodynamic coordinate system” for the analysis. Writes “Z”, “Y”, or “ZY” to the ORIENT field of the PAERO2 entry.

**Note:** In FEMAP, the “aerodynamic coordinate system” is defined using the *Analysis Set Manager* (“Model, Analysis” command). When *Analysis Type* is set to “25..Static Aeroelasticity”, the aerodynamic coordinate system is specified by the *Aerodynamic CSys* drop-down in the *NASTRAN Aerodynamic Data (AEROS)* dialog box. When *Analysis Type* is set to “26..Aerodynamic Flutter”, it is specified by the *Aerodynamic CSys* drop-down in the *NASTRAN Aerodynamic Data (AEROx, MKAEROx)* dialog box.

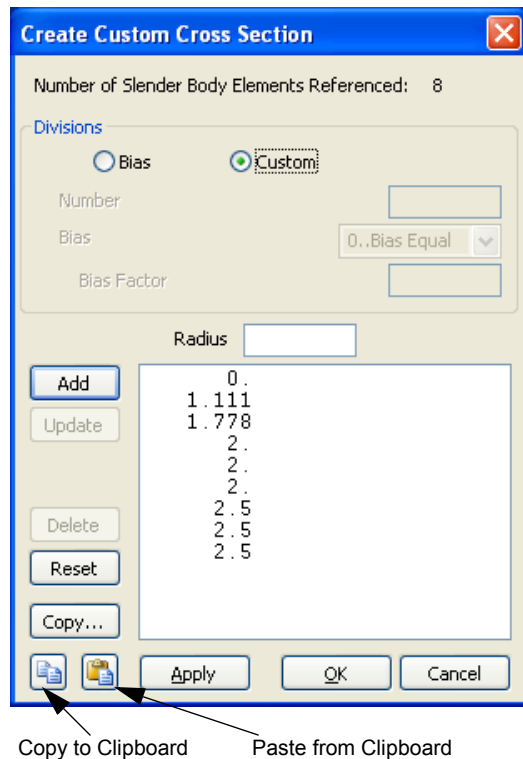
**Slender Body Division Radius** - When on, allows you to enter a list of slender body half-widths at the “end points” of the slender body “Aero Elements”. When off, the half-width of the entire slender body is specified by the *Reference Radius* value in the *Common* section. Click the *Custom List...* button to enter values in the *Create Custom Cross Section* dialog box. See *Create Custom Cross Section dialog box* section below for more details.

**Note:** The number of Radius values entered for the *Aero Property* MUST correspond to the number of divisions specified *Number Body Elements* (constant or custom) on the Aero Body. Therefore, if there are 8 constant divisions, you need to enter 9 Radius values (1 value for the “start” of the aero body, 7 for each “division location”, and 1 value for the “end”).

**Interference Body Division Radius** - When on, allows you to enter a list of slender body half-widths at the “end points” of the interference body “Aero Elements”. Click the *Custom List...* button to enter values in the *Create Custom Cross Section* dialog box. See *Create Custom Cross Section dialog box* section below for more details.

**Note:** The number of Radius values entered for the *Aero Property* MUST correspond to the number of divisions specified *Number Interference Elements* (constant or custom) on the Aero Body. Therefore, if there are 8 constant divisions, you need to enter 9 Radius values (1 value for the “start” of the aero body, 7 for each “division location”, and 1 value for the “end”).

### Create Custom Cross Section dialog box



Used to enter list of custom *Radius* (half-width) values for the slender body and interference body.

When *Divisions* is set to “Custom”, enter text values directly into the *Radius* field. Values must be above 0.0.

Click the *Add* button to add the current value in *Radius* to the list of values. To add a value to a specific place in the list, highlight a value, enter the new value, then click *Add* and the value will be added above the highlighted line.

Once a value is in the list, it can be highlighted. Click *Update* button to change a highlighted value to the value currently in the *Radius* field or click *Delete* button to remove the value from the list. The *Reset* button can be used to clear all values from the list.

The *Copy* button can be used to copy the “custom” divisions from another *Aero Property* in the current model.

The *Copy to Clipboard* and *Paste from Clipboard* icon buttons can be used to copy/paste the current list of values to/from the clipboard.

The *Apply* button will show the current radius values at each division on the “Aero Body” in the graphics window.

When *Divisions* is set to “Bias”, enter a *Number*, choose a type of *Bias* (“Bias Equal”, “Bias at Start”, “Bias at End”, “Bias at Center”, or “Bias at Both Ends”) and enter a *Bias Factor* (if needed). Once these parameters have been specified, enter a *Radius* value, then click the *Add* button in the listing section to add values from 0.0 to the *Radius* value based on the type of bias selected.

### Interference Body Theta Array 1 and Interference Body Theta Array 2

**Divisions** - use the *Define Div...* button to open the *Create Body Theta Locations* dialog box, where you can then enter a list of “theta divisions” for the interference body. The *Create Body Theta Locations* dialog box is very similar to the *Create Custom Cross Section* dialog box described above. The only difference is that you are entering *Angle* values instead of *Radius* values. The *Angle* values must be between 0 and 360 degrees.

The *Divisions* set in the *Interference Body Theta Array 1* will be written to an AEFAC entry in Nastran which is referenced by the LTH1 field of the PAERO2 entry. The *Divisions* set in the *Interference Body Theta Array 2* will be written to an AEFAC entry in which is referenced by the LTH2 field of the PAERO2 entry.

The portion of the *Interference Body Theta Array 1* section where you can enter 3 different *Interference Element 1* and *Interference Element 2* “ranges of aero body elements” is used to define TH1i (first aero element) and THN1 (last aero element) entries on the PAERO2 entry. Up to 3 ranges can be specified. All aero body elements specified in these ranges will use the *Divisions* of *Interference Body Theta Array 1*, while all other aero body elements referencing this *Aero Property* will use the *Divisions* of *Interference Body Theta Array 2*.

See figures in *Aero Body Type = “1..Aero Body (CAERO2)”* portion of Section 4.5.1, “Model, Aeroelasticity, Panel/Body...” for examples of various Slender Body and Interference Body options specified on the *Aero Property*.

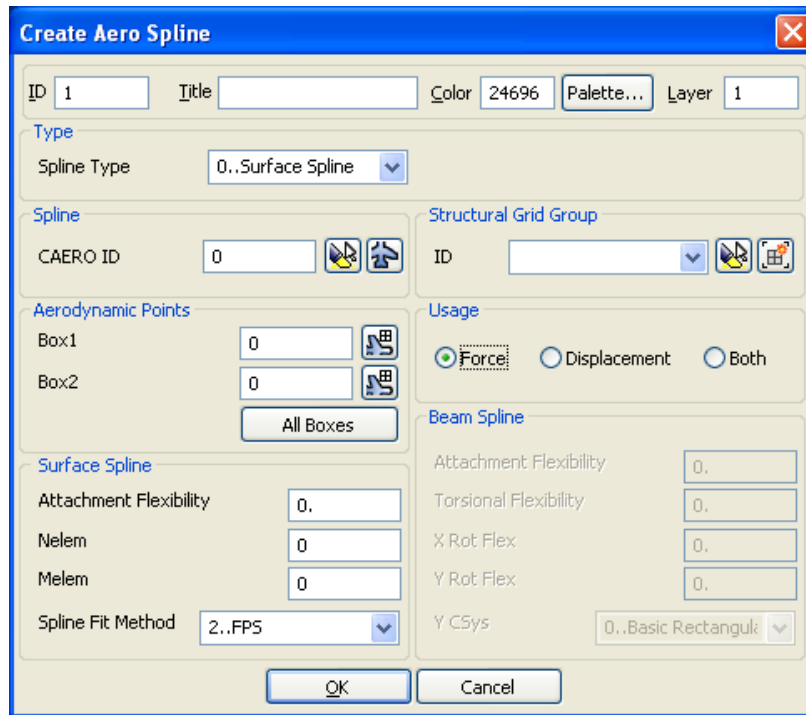
### Model, Aeroelasticity, Spline...



...creates an *Aero Spline*, which “connects” an *Aero Panel/Body* entity to nodes on the underlying “structural model”. This is done by interpolating motion (displacement) and/or forces from the aeroelastic analysis.

There are two “spline types”, Surface Spline and Beam Spline. Regardless of *Spline Type*, each *Aero Spline* must reference an existing *Aero Panel/Body* and must reference a FEMAP Group containing nodes on the structural

model. Also, at least 2 “aerodynamic points” (aero elements/aero boxes) from the referenced *Aero Panel/Body* must be selected.



The *ID*, *Title*, *Color*, and *Layer* fields are common to both *Spline Types*.

**Type**

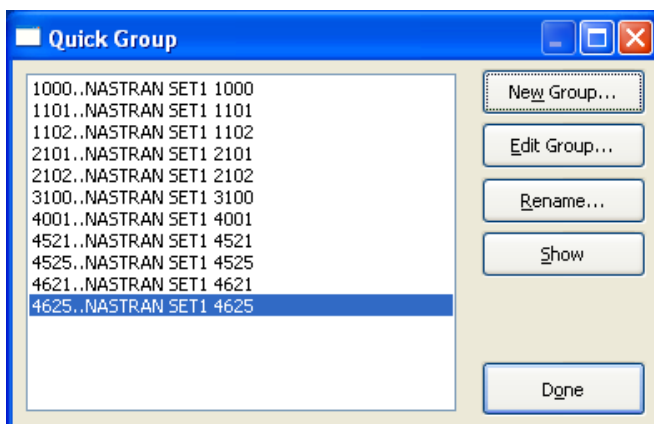
**Spline Type** - choose between “0..Surface Spline” and “1..Beam Spline”. When using “0..Surface Spline” the *Aero Spline* will be written as a SPLINE1 entry to Nastran and additional entries for the SPLINE1 may be specified in the *Surface Spline section*. A “1..Beam Spline” will be written as SPLINE2 and additional entries for SPLINE2 may be specified in the *Beam Spline section*.

**Spline**

**CAERO ID** - used to enter the ID of an existing *Aero Panel/Body* entity. The ID may be entered in manually or an *Aero Panel/Body* may be chosen from the graphics window. The *Show When Selected* icon button will highlight the specified *Aero Panel/Body* in the graphics window, while the *Select Aero Panel* icon button will allow you to choose an *Aero Panel/Body* from a list. This value will be written to the CAERO field on the SPLINEi entry.

**Structural Grid Group**

**ID** - used to specify the ID of a Group in FEMAP containing nodes on the structural model. The *Show When Selected* icon button will highlight nodes in the group in the graphics window. The *Quick Group* icon button will open the Quick Group dialog box, which can be used to create a new Group or edit an existing one.



In the *Quick Group* dialog box, click *New Group* to create a new group.

Highlight the new group or an existing one, then click *Edit Group* to *Add*, *Remove*, or *Exclude* nodes to/from the group. Since these groups only need to contain nodes, the only thing which can be selected using this dialog box is nodes.

You can rename any group by highlighting it in the list, then clicking *Rename*.

To “Show” the highlighted group in the graphics window, click *Show*. When done looking at the Group, press *Hide*. Click *Done* to exit the *Quick Group* dialog box.

The selected group will be written to as a SET1 entry to Nastran which is referenced by the SETG field of the SPLINEi entry.

### Aerodynamic Points

**Box1** - enter the ID or select an aero element (aero box) from the screen to be the first aero element in a “range of aero elements” where motions (displacements) will be interpolated. Click the *Select Aero Mesh for Aero Spline* icon button to bring up a dialog box which may make graphical selection of the aero element easier. This value will be written to the BOX1 field on the SPLINE1 and to the ID1 field of the SPLINE2 entry.

**Box2** - similar to *Box1*, but is last aero element in a “range of aero elements” where motions (displacements) will be interpolated. This value will be written to the BOX2 field on the SPLINE1 entry and to the ID2 field of the SPLINE2 entry .

**All Boxes button** - when chosen, places the aero element with the lowest ID on the referenced *Aero Panel/Body* into the *Box1* field and the one with the highest ID in the *Box2* field.

### Usage

Determines if the *Aero Spline* applies to *Force* transformation, *Displacement* transformation, or *Both*. Writes FORCE, DISP, or BOTH to the USAGE field for the SPLINEi entry.

### Surface Spline

These options are only used for *Aero Spline entities* with *Spline Type* set to “0..Surface Spline” and will be written to the appropriate field on the SPLINE1 entry.

**Attachment Flexibility** - specifies the linear attachment flexibility. Value written to the DZ field on SPLINE1

**Nelem**- number of structural elements along the local spline x-axis if using “2..FPS” option for *Spline Fit Method*. Value written to NELEM field on SPLINE1

**Melem** - number of structural elements along the local spline y-axis if using “2..FPS” option for *Spline Fit Method*. Value written to MELEM field on SPLINE1

**Spline Fit Method** - designates which spline fit method to use for the *Aero Spline*. Choose between 0..IPS (Harder-Desmarais Infinite Plate Spline), 1..TPS (Thin Plate Spline), or 2..FPS (Finite Plate Spline). Writes IPS, TPS, or FPS to METH field on SPLINE1

### Beam Spline

These options are only used for *Aero Spline entities* with *Spline Type* set to “1..Beam Spline” and will be written to the appropriate field on the SPLINE2 entry.

**Attachment Flexibility** - specifies the linear attachment flexibility. Value written to the DZ field on SPLINE2

**Torsional Flexibility** - specifies the torsional flexibility ratio (EI/GJ). Value written to DTOR field on SPLINE2. Use 1.0 for “aero bodies”.

**X Rot Flex** - specifies the rotational attachment flexibility about the spline’s x-axis (in-plane bending rotations) is specified in *YCSys*. Not used for “aero bodies”, only “aero panels”. Value written to DTHX field on SPLINE2.

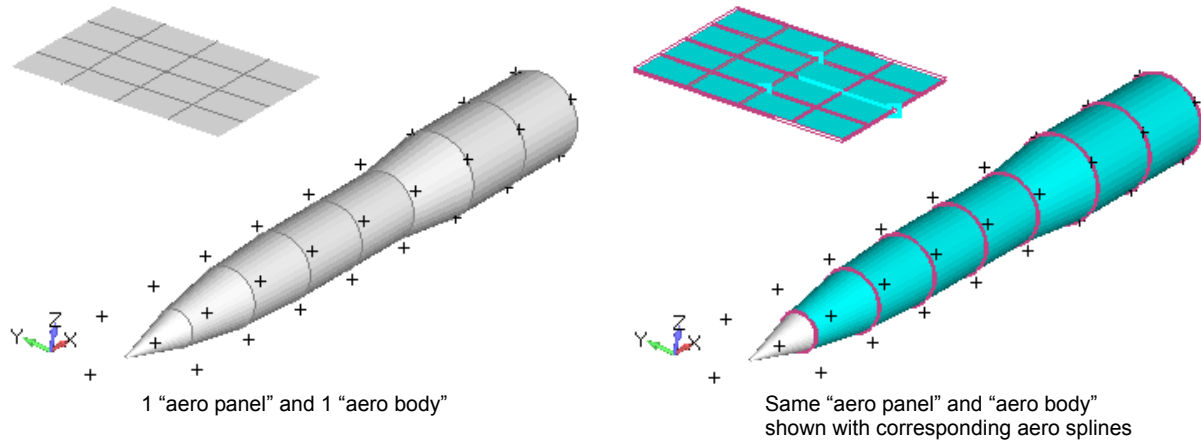
**Y Rot Flex** - specifies the rotational attachment flexibility about the spline’s y-axis (torsion) is specified in *YCSys*. May be used for “slope” of “aero bodies”. Value written to DTHY field on SPLINE2.

**Note:** The values for *Attachment Flexibility*, *X Rot Flex*, and *Y Rot Flex* are used for smoothing. Flexibility values of 0.0 in these fields imply rigid attachment (i.e., no smoothing). Negative values for *X Rot Flex* and *Y Rot Flex* imply infinity, therefore, no attachment.

**Y Csys** - Rectangular coordinate system where the y-axis defines the axis of the spline. Not used for “aero bodies”, only “aero panels”. Only rectangular coordinate systems will be available for selection. Value written to DCID field on SPLINE2.

For display purposes, each *Aero Spline* will be drawn “on top” of the selected “aero mesh” of the referenced *Aero Panel/Body*. In addition, straight “connection lines” will be drawn from each node in the referenced *Structural Grid Group* to the centroid of the referenced *Aero Panel/Body*.

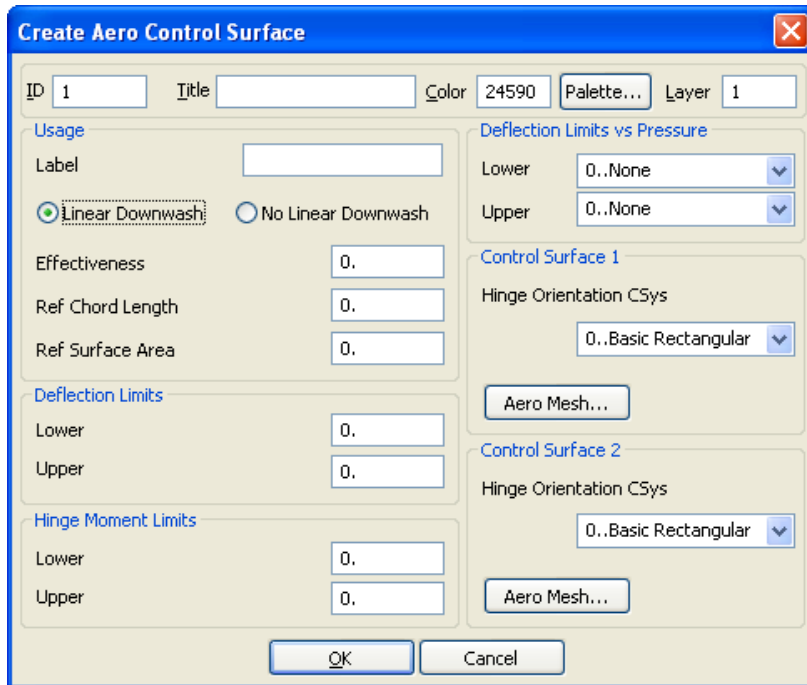
For example, an “aero panel” and an “aero body” are shown on the left. The corresponding *Aero Splines* for these *Aero Panel/Body* entities are shown on the right.



**Model, Aeroelasticity, Control Surface...**



...creates an *Aero Control Surface*, which is used to specify an aerodynamic control surface. Each *Aero Control Surface* uses ranges of aero elements on “aero panels” (not “aero bodies”) to represent the aerodynamic control surface. Two ranges of aero elements may be specified on each *Aero Control Surface*, with each “control surface” range able to use a different “hinge orientation coordinate system”.



The *ID*, *Title*, *Color*, and *Layer* fields work as the do for other entities.

**Usage**

These options allow you to create an easy to recognize label which will be written to the Nastran input file and effect how each *Aero Control Surface* is used in the aeroelastic analysis.

**Label** - specifies the name of the control surface. Limited to 7 characters. Text written to the LABEL field on AESURF.

**Linear Downwash/No Linear Downwash** - specifies if “Linear DownWash” is computed as part of the database (*Linear Downwash*) or if the effects of the control surface must be entered by the user directly (*No Linear Downwash*). Writes LDW or NOLDW to the LDW field on AESURF.

**Effectiveness** - specifies the control surface effectiveness, which cause forces to be modified by this value (i.e., to achieve 40% reduction of effectiveness, specify this value as 0.6). Value written to EFF field on AESURF entry.

**Ref Chord Length** - specifies the reference chord length of the control surface. Value written to CREFC field on AESURF entry.

**Ref Surface Area** - specifies the reference surface area of the control surface. Value written to CREFS field on AESURF entry.

### Deflection Limits

Specifies the *Lower* and *Upper* deflection limits for the control surface in radians. Values written to PLLIM and PULIM fields on AESURF entry.

### Hinge Moment Limits

Specifies the *Lower* and *Upper* hinge moment limits for the control surface in force-length units. Values written to HMLLIM and HMULIM fields on AESURF entry.

### Deflection Limits vs Pressure

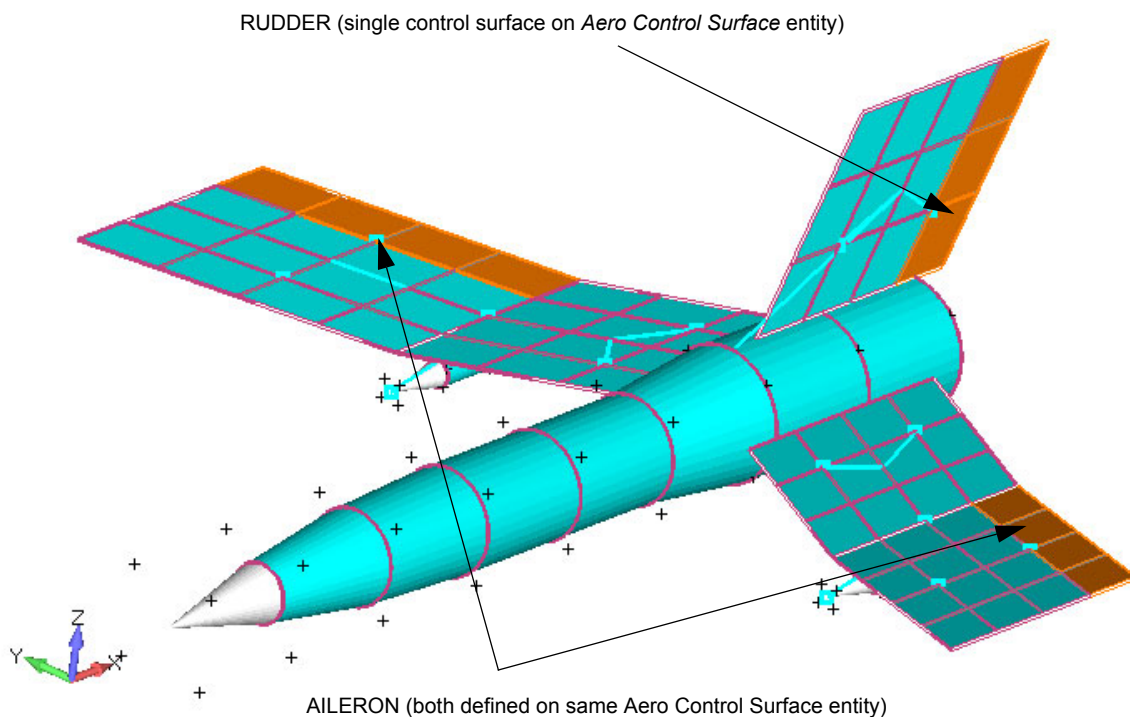
Allows you to choose functions to specify *Lower* and *Upper* deflection limits for the control surface as a function of dynamic pressure. Functions written as TABLED1 entries to Nastran then referenced by TQLLIM and TQULIM fields on AESURF entry.

### Control Surface 1 and Control Surface 2

Specify a rectangular coordinate system as the *Hinge Orientation CSys* (writes CIDi to AESURF entry), then click the *Aero Mesh...* button to choose “aero panel elements” using a typical *Entity Selection* dialog box. The selected “aero mesh” in each section will be written as an AELIST to Nastran, then referenced by the corresponding ALIDi field(s) on the AESURF entry

*Aero Control Surfaces* are displayed “on top” of the “aero panel elements”.

A “complete” Aero model shown below with Aero Control Surfaces:



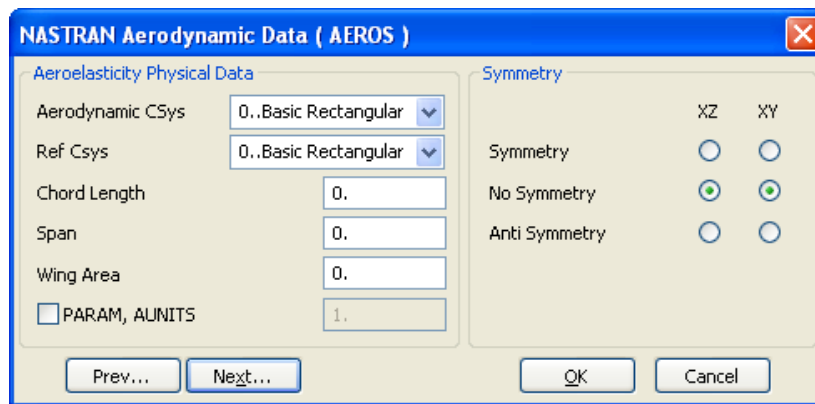
## Static Aeroelasticity Analysis

NX and MSC/MD Nastran have the ability to perform Static Aeroelasticity analysis using Solution Sequence 144 (SOL 144).

Specific Solution 144 dialog boxes will appear in the Analysis Set Manager when the *Analysis Type* has been set to 25..Static Aeroelasticity. The *NASTRAN Aerodynamic Data (AEROS)* dialog box allows you to enter basic parameters for static aeroelasticity and an optional conversion factor PARAM used for all subcases. On the other hand, the *NASTRAN Aeroelastic Trim Parameters* dialog box contains a number of “Trim Parameters”, which may be specified in the “Master Requests and Conditions” for an analysis with no subcases or specified individually for each subcase.

AEROF and APRES will be written to case control to request results from static aeroelastic analysis.

### NASTRAN Aerodynamic Data (AEROS)



#### Aerodynamic Physical Data

**Aerodynamic CSys** - specifies the aerodynamic coordinate system. Must be a rectangular coordinate system. Flow is in the +X direction. Value written to the ACSID field of the AEROS entry.

**Ref CSys** - specifies the reference coordinate system. Must be a rectangular coordinate system. All AESTAT degrees-of-freedom defining trim variables will be defined in this coordinate system. Value written to the RCSID field of the AEROS entry.

**Chord Length** - specifies reference chord length. Value written to the REFC field of the AEROS entry.

**Span** - specifies reference span. Value written to the REFB field of the AEROS entry.

**Wing Area** - specifies reference wing area. Value written to the REFS field of the AEROS entry.

**PARAM, AUNITS** - writes PARAM, AUNITS to the Nastran input file with the specified value. This parameter is used to convert accelerations specified in units of gravity on the TRIM entries to units of distance per time squared.

#### Symmetry

**XZ** - specifies the symmetry “key” for the x-z plane of the *Aerodynamic CSys*. Based on option selected for XZ, writes an integer to the SYMXZ (*Symmetry* = +1, *No Symmetry* = 0, *Anti-Symmetry* = -1).

**XY** - specifies the symmetry “key” for the x-y plane of the *Aerodynamic CSys*, which can be used to simulate “ground effects”. Based on option selected for XY, writes an integer to the SYMXY (*Symmetry* = -1, *No Symmetry* = 0, *Anti-Symmetry* = +1).

### NASTRAN Aeroelastic Trim Parameters

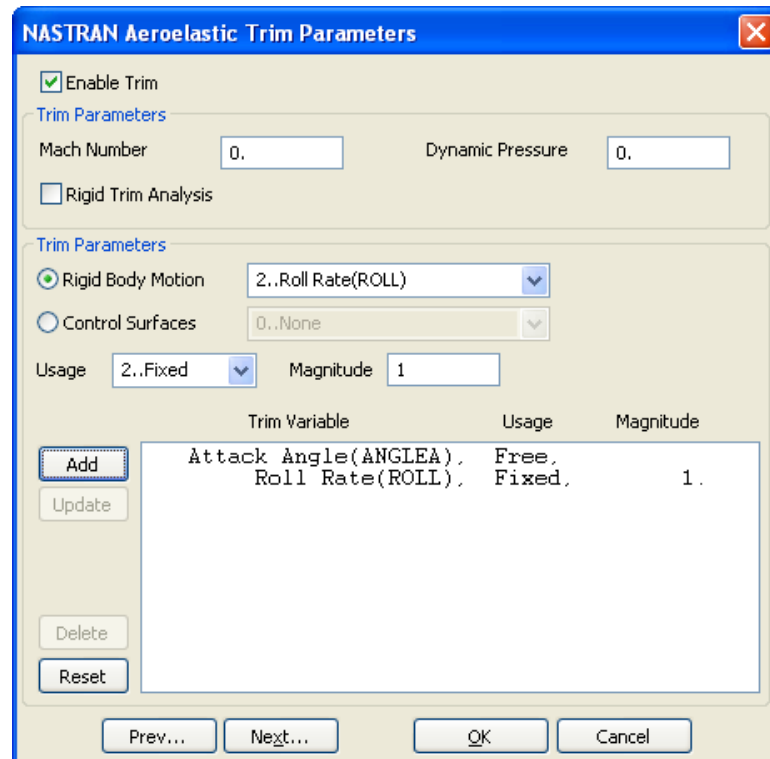
The *Enable Trim* check box may be used to toggle the options set in the *NASTRAN Aeroelastic Trim Parameters* dialog on/off in the Master case and for each subcase.

The *Trim Parameters* in the upper portion of the dialog box are used to define values on the TRIM bulk data entry.

**Mach Number** - specifies the mach number. Value written to the MACH field of the TRIM entry.

**Dynamic Pressure** - specifies a value for dynamic pressure. Value written to the Q field of the TRIM entry.

**Rigid Trim Analysis** - specifies if trim analysis is rigid. When “on” a value of 0.0 is written to the AEQR field of the TRIM entry. When “off”, a value of 1.0 is written to the AEQR field of the TRIM entry.



The *Trim Parameters* in the lower portion of the dialog box write AESTAT and/or TRIM entries using values entered for various “Trim Variables” in the list.

When set to *Rigid Body Motion*:

- Select from the list of "Standard Labels Defining Rigid Body Motions" on the AESTAT (ANGLEA, SIDES, ROLL, PITCH, YAW, URDD1, URDD2, URDD3, URDD4, URDD5, and URDD6)
- Select a *Usage* (1..Free or 2..Fixed). If 2..Fixed, enter a magnitude as well (UXi value on TRIM entry).
- Click *Add* to add the “Trim Variable” to the list in the lower portion of the dialog box.

When set to *Control Surfaces*:

- Select from the list of *Aero Control Surfaces* in your model, then follow steps b and c above.

To update a “Trim Variable”, highlight one in the list, set the appropriate values, then click *Update*. The *Delete* button is used to delete a single highlighted “Trim Variable” from the list, while *Reset* will delete all “Trim Variables” from the list.

## Aerodynamic Flutter Analysis

NX and MSC/MD Nastran have the ability to perform Aerodynamic Flutter analysis using Solution Sequence 145 (SOL 145).

Specific Solution 145 dialog boxes will appear in the Analysis Set Manager when the *Analysis Type* has been set to 26..Aerodynamic Flutter. The *NASTRAN Aerodynamic Data (AEROx, MKAEROx)* dialog box allows you to enter basic parameters for unsteady aerodynamics, a table of Mach numbers vs. Reduced frequencies, and some additional dynamic analysis information. On the other hand, the *NASTRAN Flutter Parameters* dialog box contains a number of “Flutter Parameters”, which may be specified in the “Master Requests and Conditions” for an analysis with no subcases or specified individually for each subcase.

The standard *NASTRAN Modal Analysis* dialog box is also used to setup a Flutter analysis. See Section 8.7.1.9, “Modal Analysis” for more information about the options available in this dialog box.

When using the PK method, results from the Flutter Summary Table will be imported into FEMAP as functions.

### NASTRAN Aerodynamic Data (AEROx, MKAEROx)

#### Aerodynamic Physical Data

**Aerodynamic CSys** - specifies the aerodynamic coordinate system. Must be a rectangular coordinate system. Flow is in the +X direction. Value written to the ACSID field of the AERO entry.

**Velocity** - specifies the velocity for aerodynamic force data recovery and to calculate the BOV parameter. Value written to the VELOCITY field of the AERO entry.

**Ref Length** - specifies reference length for reduced frequency. Value written to the REFC field of the AERO entry.

**Ref Density** - specifies reference density. Value written to the RHOREF field of the AERO entry.

#### Symmetry

**XZ** - specifies the symmetry “key” for the x-z plane of the *Aerodynamic CSys*. Based on option selected for XZ, writes an integer to the SYMXZ (*Symmetry* = +1, *No Symmetry* = 0, *Anti-Symmetry* = -1).

**XY** - specifies the symmetry “key” for the x-y plane of the *Aerodynamic CSys*, which can be used to simulate “ground effects”. Based on option selected for XY, writes an integer to the SYMXY (*Symmetry* = -1, *No Symmetry* = 0, *Anti-Symmetry* = +1).

#### Mach Number - Frequency Table

Select a function to specify a list of Mach Numbers vs. Reduced Frequencies (Type of function MUST be “34..Mach Number vs. Freq”). To create a new function “on-the-fly”, click the *New Function* icon button. Writes as many MKAERO2 entries as needed for all XY data pairs in the function (4 data pairs per MKAERO2).

#### Dynamics Options

These options allow you to limit the modes used to analyze the response of the structure by allowing you to set a subset of the frequency range specified in the *NASTRAN Modal Analysis* dialog box or simply enter a fewer number of modes to use. This can be useful if restarting from a *Modal Analysis* which had a larger frequency range or more modes than are needed to run an accurate *Modal Transient* analysis.

*Number of Modes* will write the PARAM,LMODES entry, *Lowest Freq (Hz)* will write PARAM,LFREQ and *Highest Freq (Hz)* will write PARAM,HFREQ. Specify *Rigid Body Zero Modes (FZERO)* to have modes with values under specified value be considered “0”.

When checked, the *As Structural* check box will write out PARAM,KDAMP,-1, which causes the viscous modal damping, specified by the *Modal Damping Table* in the *NASTRAN Flutter Parameters*, to be entered into the complex stiffness matrix as structural damping.

### NASTRAN Flutter Parameters

The *Modal Damping Table* can be specified here (function Type must be “6..Structural Damping vs. Freq”, “7..Critical Damping vs. Freq”, or “8..Q Damping vs. Frequency”) and writes a TABDMP1 entry..

The *Enable Flutter* check box may be used to toggle the options set in the *NASTRAN Flutter Parameters* dialog on/off in the Master case and for each subcase. A FMETHOD= # case control entry will be written to each subcase, specifying which FLUTTER entry to use for each subcase.

### Flutter Parameters

**Flutter Method** - specifies the flutter analysis method. There are four methods available:

- 0..K-Method (K written to METHOD field on FLUTTER entry)
- 1..PK-Method (PK written to METHOD field on FLUTTER entry). Is the default method.
- 2..PKNL-Method (PKNL written to METHOD on FLUTTER entry). Is PK-Method with no looping.
- 3..KE-Method (KE written to METHOD on FLUTTER entry). Is K-Method restricted for efficiency.

**Density Ratios** - select a function to specify the density ratio vs. aerodynamic factor. Type of function must be “35..vs.Aerodynamic Factor”. Function values written to FLFACT entry which is then referenced by the DENS field of the FLUTTER entry.

**Mach Numbers** - select a function to specify the mach numbers vs. aerodynamic factor. Type of function must be “35..vs.Aerodynamic Factor”. Function values written to FLFACT entry which is then referenced by the MACH field of the FLUTTER entry.

**Velocity/Reduced Freq** - select a function to specify the velocity (PK and PKNL methods) or reduced frequencies (K and KE methods) vs. aerodynamic factor. Type of function must be “35..vs.Aerodynamic Factor”. Function values written to FLFACT entry which is then referenced by the RFREQ field of the FLUTTER entry.

**Interpolation Method (K and KE methods only)** - specify an interpolation method for aerodynamic matrix interpolation. Choose between *Linear* (writes L to IMETH field on FLUTTER entry. Default) or *Surface* (writes S to IMETH field on FLUTTER entry).

**Number Eigenvalues (PK and PKNL methods only)** - specify the number of eigenvalues. Value written to NVALUE field on the FLUTTER entry.

**Convergence (PK and PKNL methods only)** - specify a convergence value for k, which a value used to accept eigenvalues. Value written to EPS field on the FLUTTER entry.

## Loads and Constraints

- Updated *Model, Load, From Freebody* command to allow selection of a *Freebody* entity currently in the model.
- Added *Map Tolerance* field for *Model, Load, Map Output from Model* command and in *Data Surface* editor

*Map Tolerance* - When a “Target” location is projected onto the “Source” data surface and the distance to a discrete data point is less than the tolerance, the “Source” value of the "coincident" location is directly mapped to the “Target” without interpolation. If multiple nodes fall within this tolerance, then the first one encountered numerically will be directly mapped. Default value is the "Merge Tolerance" of the "Target” model.

- Enhanced *Model, Constraint, Expand* command.

## Connections (Connection Region, Properties, and Connectors)

- Added *Activation Distance* section to *Penetration* section on the NEiNastran tab. Allows you to specify a value (real or AUTO) for MAXAD or specify values for MAXNAD and/or MAXRAD)
- Added *Friction* section to LS-DYNA tab to restore ability to set these values for LS-Dyna contact.
- Updated Fluid Regions to not use the PLANE1, PLANE2, RMAX, FMEXACT inputs when NEi Nastran is default solver.

## Groups and Layers

- Added on *Group, Coord Sys, On Property* to add coordinate systems on a Property to a group and *Group, Coord Sys, on CSys* to add coordinate systems referenced by selected coordinate systems to a group.
- Enhanced *Group, Operations, Add Related Entities* to now add Coordinate Systems referenced on Properties and coordinate systems referenced by other coordinate systems to a group.

## Views

- Added on *View, Align by, Surface* to align the view normal to a selected surface and *View, Align by, Normal to Plane* to align view normal to a specified plane.
- Added Aero Panel, Aero Mesh, Aero Interference, Aero Splines, and Aero Control Surfaces to Labels, Entities and Color Category of View, Options.
- Added Preview option to Tools and View Style Category of View, Options. Controls the size of the “marker” shown in the graphics window when using the Preview button to preview the location of a coordinate.
- Added Freebody, Freebody Node Markers, Freebody Vectors, Freebody Total Force, Freebody Total Moment, Freebody Nodal Force, and Freebody Nodal Moment to PostProcessing Category of View, Options.
- Added Max Only and Min Only options to the Label Mode of the Contour/Criteria Style option in the PostProcessing Category of View, Options.
- Only tabs of entity types which currently exist in the model will be displayed in the *View, Visibility* dialog box.

## Output and Post-Processing

- Freebody display has been enhanced and is now managed via the Freebody tool in the PostProcessing Toolbox.
- Added “Select By Vector” options for Nodal and Elemental output in *Model, Output, Forced Response*. This allows you to limit the amount of output created by this command.

## Geometry Interfaces

The following FEMAP interfaces have been updated to support newer geometry formats:

FEMAP Interface	Latest Supported Version
Parasolid	Parasolid 24.0
Solid Edge	Solid Edge with Synchronous Technology 4
NX	NX 8.0

FEMAP Interface	Latest Supported Version
Catia	CATIA V5 R20
Pro/Engineer	Wildfire 5.0
ACIS	ACIS 21, SP3
SolidWorks	SolidWorks 2010

- Updated stereolithography export to export both solid and plate elements at the same time if they are both selected. If some plates are coincident with

For details, see “Geometry Interfaces” in the *FEMAP User Guide*.

## Analysis Program Interfaces

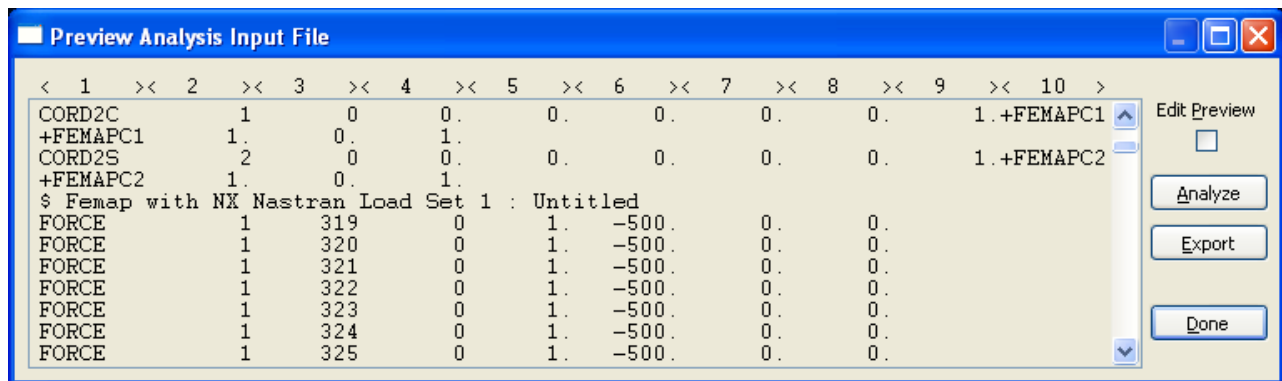
Several of the analysis program interfaces have been improved. These changes include:

- Analysis Set Manager Enhancements
- FEMAP Neutral File Interface
- NX Nastran Interface
- Nastran Interfaces (NX and MSC/MD)
- MSC/MD Nastran Interface
- NEi Nastran Interface
- ANSYS Interface
- ABAQUS Interface
- DYNA Interface

For details, see “Analysis Program Interfaces” in the *FEMAP User Guide*.

## Analysis Set Manager Enhancements

- Updated *Preview Analysis Input File* dialog box to show 80 characters per line by default.



## FEMAP Neutral File Interface

- Updated Neutral Read and Write for v10.3 changes

## NX Nastran Interface

- Added support for BGRESULTS Glue Output results.
- Added support for PLOADE1 entry.
- Added support for “Mean Dilatational Formulation” on the PPLANE entry.
- Added support MATVE and TABVE entries. GFUNC and KFUNC are defined using dimensionless FEMAP functions where x = decay factor and y = bulk or shear modulus. MOD0 is defined by adding decay time = 0 and MOD0 first term.

- Added support for MATHEV and MATHEM to the MATHE material definition for SOL 601/701.
- Added support for PARAM,CNTSET

A number of bugs were corrected

For details, see “Analysis Program Interfaces” in the *FEMAP User Guide*.

### **Nastran Interfaces (NX and MSC/MD)**

- Added Preference to write continuation cards as “+” only.
- Added support for the Automatic Householder Method (AHOU) for modal analysis.
- Added support for multicase SUPORT1 definition.
- Added support for SOL AESTAT (SOL 144), SOL SEFLUTTER (SOL 145), CAERO1, CAERO2, PAERO1, PAERO2, SPLINE1, SPLINE2, AESURF, AEFACT, AEROS, SET1, TRIM, AERO, FLUTTER, FLFACT, FMETHOD, MKAERO1, and MKAERO2 to support Static Aeroelasticity and Aerodynamic Flutter.
- Added support for PARAM,AUNITS to support Static Aeroelasticity.

A number of bugs were corrected

For details, see “Analysis Program Interfaces” in the *FEMAP User Guide*.

### **MSC/MD Nastran Interface**

- Added support for nonlinear results on solid elements from versions above 2008. Results from versions 2008 and before are also still supported.

A number of bugs were corrected.

For details, see “Analysis Program Interfaces” in the *FEMAP User Guide*.

### **NEi Nastran Interface**

- Added support for Laminate Failure Theories: Max Stress (STRESS), NASA LaRC (LAERC02), Puck PCP (PUCK), and Multicontinuum (MCT). Specified on Laminate Property.
- Added support for PARAM, RIGIDELEM2ELAS, ON and PARAM, RIGIDELEMTYPE, BAR to support thermal expansion of Rigid elements.
- Added support for EXTRACTMETHOD (options = LANCZOS, AUTO, or SUBSPACE) for Modal Analysis.
- Added support for PARAM,INREL,AUTO.
- Added support for NITINOL material type. Found in dialog box when Type = *Other Types*.
- Added support for MAXAD and MAXNAD/MAXRAD for contact.

A number of bugs were corrected.

For details, see “Analysis Program Interfaces” in the *FEMAP User Guide*.

### **ANSYS Interface**

- Added support for ANSYS 13.0

A number of bugs were corrected.

For details, see “Analysis Program Interfaces” in the *FEMAP User Guide*.

### **ABAQUS Interface**

A number of bugs were corrected.

For details, see “Analysis Program Interfaces” in the *FEMAP User Guide*.

### **DYNA Interface**

- Added support for ABCD Contact entries.

A number of bugs were corrected.

For details, see “Analysis Program Interfaces” in the *FEMAP User Guide*.

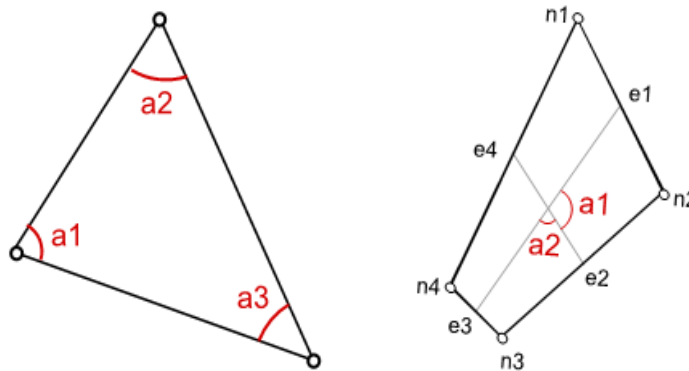
## Tools

- Added *Color*, *Next ID*, and *Inc* values for *Aero Panel*, *Aero Property*, *Aero Spline*, and *Aero Surface* to *Tools, Parameters*.
- Added ability to determined surface area of “combined surfaces” to *Tools, Measure, Surface Area*.
- Added “Skew” element quality check to *Tools, Check, Element Quality* command.

Skew measures internal angular deviation of a face using the edge bisector method. This check matches results reported by NX Nastran. Only this command will return Skew results for faces of supported solid elements.

For Triangular elements and element faces, Skew measures internal angle and reports minimum for all angles of 2-D (a1, a2 & a3 in figure below) and all angles of all faces of supported 3-D type elements.

For Quadrilateral elements and element faces, Skew test for quadrilateral faces reports minimum angle between face edge bisectors ( a1 & a2 in figure below ). Minimum for all faces is reported for supported 3-D elements.



## OLE/COM API

### New API Objects and Attributes

- Added Element Quality (feElementQuality) object to the API. Also added AspectRatioOn, TaperOn, AlternateTaperOn, InternalAngleOn, SkewOn, WarpingOn, NastranWarpingOn, TetCollapseOn, JacobianOn, CombinedOn, ExplicitTimeStepOn, AspectRatioLimit, TaperLimit, AlternateTaperLimit, InternalAngleLimit, SkewLimit, WarpingLimit, NastranWarpingLimit, TetCollapseLimit, JacobianLimit, CombinedLimit, and ExplicitTimeStepLimit to Element Quality Object.
- Added Aero Panel/Body (feAeroPanel) object to the API. Also added color, layer, title, propID, defCSys, nSpan, nChord, iIgid, Pt1, Pt4, dChord12, dChord43, nLspanID, nLchordID, and type attributes to the Aero Panel/Body object.
- Added Aero Property (feAeroProp) object to the API. Also added color, layer, title, pdval, pnval, ap\_d\_width, ap\_d\_ar, ap\_i\_orient, ap\_i\_lrsb, ap\_i\_lrib, ap\_i\_lth1, ap\_i\_lth2, ap\_i\_thi1, ap\_i\_thi2, ap\_i\_thi3, ap\_i\_thn1, ap\_i\_thn2, ap\_i\_thn3, and type attributes to the Aero Property object.
- Added Aero Spline (feAeroSpline) object to the API. Also added color, layer, title, type, icaero, ibox1, ibox2, isetg, dz, meth, nelem, melem, usage, dtor, cid, dthx, and dthy attributes to the Aero Spline object.
- Added Aero Control Surface (feAeroSurf) object to the API. Also added color, layer, title, csys, csys1, aeid, aeid1, eff, ldw, crefc, crefs, pllim, pulim, hmllim, hmulum, tqllim, tqulim, and label attributes to the Aero Control Surface object.
- Added Freebody (feFreebody) object to the API. Also added title, DisplayMode, Group, CSys, NodeMarkerColor, TotalVectorMode, ShowTotalVec, SumComponents, TotalVecColor, x, y, z, NodalVectorMode, ShowNodalVec, NodalVecColor, and SumContributions attributes to the Freebody object.
- Added Geometry Preparation and Meshing (feMesher) object to the API. This object has been partially added and is for “Future Use” and should not be used.

- Added NasAeroOn, NasAeroCsID, NasAeroRefCsID, NasAeroRefLength, NasAeroRefSpan, NasAeroRefArea, NasAeroSymXY, NasAeroSymxz, NasAeroAeunit, NasAeroAeunitVal, NasAeroVelo, NasAeroRefDens, NasAeroMkFuID, vNasAeroFreqKeep, NasAeroModesKeep, NasAerobPARAMfzero, NasAerodPARAMfzero, and NasAeroDampMethod attributes to Analysis Manager (AnalysisMgr) object for Static Aeroelasticity and Aerodynamic Flutter. Also, added NasBulkCntAset for Bulk Data.
- Added NasCaeOn, NasCaeMachNumber, NasCaeDynPressure, NasCaeRigidTrim, NasCaeWrtieTrim, NasCflOn, NasCflMethod, NasCflDenID, NasCflMachFactID, NasCflRfreqFactID, NasCflFliMethod, NasCflEig, NasCflEps, NasCflWriteFlutter, NasCflSdamp attributes to Analysis Case (AnalysisCase) object for Static Aeroelasticity and Aerodynamic Flutter.

### New API Methods

- Added NextExistingInSet to Entity API objects
- Added Clear, SetModelDefaults, GetModelDefaults, CheckQuality, GetAspectRatio, AspectRatio, GetTaper, Taper, GetAlternateTaper, AlternateTaper, GetInternalAngle, InternalAngle, GetSkew, Skew, GetWarping, Warping, GetNastranWarping, NastranWarping, GetTetCollapse, TetCollapse, GetJacobian, Jacobian, GetCombined, Combined, GetExplicitTimeStep, and ExplicitTimeStep to Element Quality object.
- Added GetDivisionList, PutDivisionList, SlenderBodyCount, InterferenceBodyCount, PanelSpanCount, PanelChordCount, and GetBoxSet to Aero Panel/Body object
- Added GetThetaList, PutThetaList, GetRadiList, PutRadiList, ClearSbList, ClearIbList, ClearTheta1List, and ClearTheta2List to Aero Property object
- Added GetNodeSet and GetBoxSet to Aero Spline object
- Added PutSurfaceSet1, PutSurfaceSet2, GetSurfaceSet1, GetSurfaceSet2, ClearSurfaceSet1, and ClearSurfaceSet2 to Aero Control Surface object
- Added GetElements, SetElements, ClearElements, GetNodes, SetNodes, ClearNodes, CalculateNodalCenter, and CalculateSummation to Freebody object.
- Added Axis and TwoAxis to Csys object
- Added ClearMeshLoc and PointsAsSet to Curve object
- Added FindMappedMeshingCorners, AddMeshPoint, CountMeshPoint, and PointLoops to Surface object
- Added CountCommon, CountNotCommon, HasNotCommon, and NextAfter to Set object
- Added SetMultiGroupListFromSets to View object
- Added IsEmpty to SortSet object
- Added ElementsAsSet2 to Solid object
- Added MapFromModelToSet2 to MapOutput object.
- Added DeleteAnalysisCase to Analysis Case object.
- Added GetList to Group object

### New Global Variables

- Added Pref\_JTFileVersion, Pref\_GIFOptimized, and Pref\_2DTensorPlotOverride
- Added Pref\_RenderXORPicking, Pref\_RenderMultiModelMem, Pref\_RenderDebugElapsedTime, Pref\_DebugAllTime, Pref\_DebugOGLErrors, Pref\_RenderBlockSize, and Pref\_DialogRefresh
- Added Pref\_PickMethod, Pref\_ConfirmDelete, Pref\_ShowMode, Pref\_ShowLables, Pref\_ShowNormals, and Pref\_ShowColor.
- Added Pref\_PreserveNextID, Pref\_DBOpenSaveWindowsIO, and Pref\_DBOpenSaveUnblockedIO
- Added Pref\_Prev10TetMesh, Pref\_Prev10SurfaceMesh, Pref\_ElemQualAspectRatio, Pref\_ElemQualTaper, Pref\_ElemQualAltTaper, Pref\_ElemQualIntAngles, Pref\_ElemQualSkew, Pref\_ElemQualWarping, Pref\_ElemQualNastranWarping, Pref\_ElemQualTetCollapse, Pref\_ElemQualJacobian, Pref\_ElemQualCombined, Pref\_ElemQualExplicitTime, Pref\_ElemQualAspectRatioVal, Pref\_ElemQualTaperVal, Pref\_ElemQualAltTaperVal, Pref\_ElemQualIntAnglesVal, Pref\_ElemQualSkewVal,

Pref\_ElemQualWarpingVal, Pref\_ElemQualNastranWarpingVal, Pref\_ElemQualTetCollapseVal, Pref\_ElemQualJacobianVal, Pref\_ElemQualCombinedVal, and Pref\_ElemQualExplicitTimeVal. Also, added Pref\_OrientSolidIsoOutput, Pref\_OrientSolidAnisoOutput, Pref\_OrientSolidHyperOutput, Pref\_Tria3StressOutput, PrefTria3StrainOutput, PrefTria3ForceOutput, Pref\_Tria6StressOutput, PrefTria6StrainOutput, PrefTria6ForceOutput, Pref\_Quad4StressOutput, PrefQuad4StrainOutput, PrefQuad4ForceOutput, Pref\_Quad8StressOutput, PrefQuad8StrainOutput, PrefQuad8ForceOutput

- Added Pref\_NastranScratchLocation, Pref\_NasAlternateContinue and Pref\_NasDballScratch
- Added Pref\_LibLayup
- Added vPref\_SpaceballFactors, Pref\_SpaceballFactors, Pref\_SpaceballSensitivity, and Pref\_SpaceballDebug
- Added slots 18-21 for Aero Panel, Aero Prop, Aero Spline, and Aero Control Surface to Pref\_EntityColor
- Updated Pref\_LengthBasedMeshSize from BOOL to INT4

**The following functions have been added:**

- feFileIsModified
- feGroupBoolean2
- feSurfaceExtend
- feOutputForceBalance2
- feLoadFromFreebody
- feVectorNormalizedDotProduct
- feSurfaceMidAttrib
- feCoordCenterOfPoints

## Preferences

### Views

- Added *JT File Version* drop-down

Sets the default value displayed for “JT File Version” in the *JT Options* dialog box when saving a picture as a JT file (see Section 2.5.4.7, “File, Picture, Save JT...”). Determining the proper “JT File Version” for software packages which support JT files is up to the user. Available file versions are 8.0, 8.1, 8.2, 9.0, 9.1, 9.2, 9.3, 9.4, and 9.5.

- Added *Optimized* check box to the *Color Optimization* section of *GIF Options*.

The *Optimized* option will remove infrequently used colors in the picture first when reducing to 256 colors.

- Both the actual and log values will be included as text when the *Include Text for XY Plots* option is on.

### Render

- Added the *All*, *Elapsed Time*, and *OpenGL Errors* check boxes under *Print Debug Messages* in the *Advanced/Debug* options section.

If you turn this option on, FEMAP will write print debug messages to the *Messages* window. FEMAP Support may request this information to help you resolve a graphics display problem. The “Elapsed Time” option reports “elapsed times” of various operations related to FEMAP graphics. Turning on the “All” option will report quite a bit more information, but will also be slower. “OpenGL Errors” will report any errors specifically related to OpenGL. If there are no OpenGL errors, then nothing different will be reported. This option is off by default as it takes additional time to query FEMAP for these types of errors.

### User Interface

- Added *Pick Method* drop-down to *Graphical Selection* section to allow selection of a default “Pick Method”.

This option controls the default Pick Method for Entity Selection dialog boxes. Choices are “Pick Normal”, “Pick Query”, and “Pick Front”. See Section 4.3.1, “Entity Selection” for more information about the Pick Methods.

### Database

- Added “16K test” to *Read/Write Test* for determining optimal *Open/Save Method*.

**Geometry/Model**

- Added “Skew” to enter default value in the *Element Quality Preferences* dialog box.
- Changed *Use Length Based Mesh Sizing* option to *Mesh Sizing* drop-down to allow choice of the new default option, “2..Parametric/Equal Length”.

**Interfaces**

- Added *Write Alternate Line Continuation* option to the *Nastran Solver Write Options* section

When this option is on, FEMAP will write all Nastran line continuation markers to “+” only. The only exception is for the “Basic Cylindrical” and “Basic Spherical” coordinate systems written out to every Nastran file by default. Previous versions of FEMAP would write “descriptive” continuation markers, which can be turned on again by turning off this option

- Added *Include Database Files in Scratch* option to the *Nastran Solver Write Options* section

When on, this option will make sure the Nastran files (i.e., \*.DBALL, \*.MASTER, \*.OBJSCR, \*.SCR300, \*.SCRATCH, etc.) are also written to the specified *Scratch Directory* for Nastran. This is accomplished by using the “dbs” Nastran command line option when running the analysis.

- *Check References on Open* and *Create Geometry References* in *File Reference Options* section are now “Off” by default.

**Color**

- Added options to set the default color for *Aero Panel*, *Aero Prop*, *Aero Spline*, and *Aero Control Surface*.