simulating REALITY™

MSC SOFTWARE

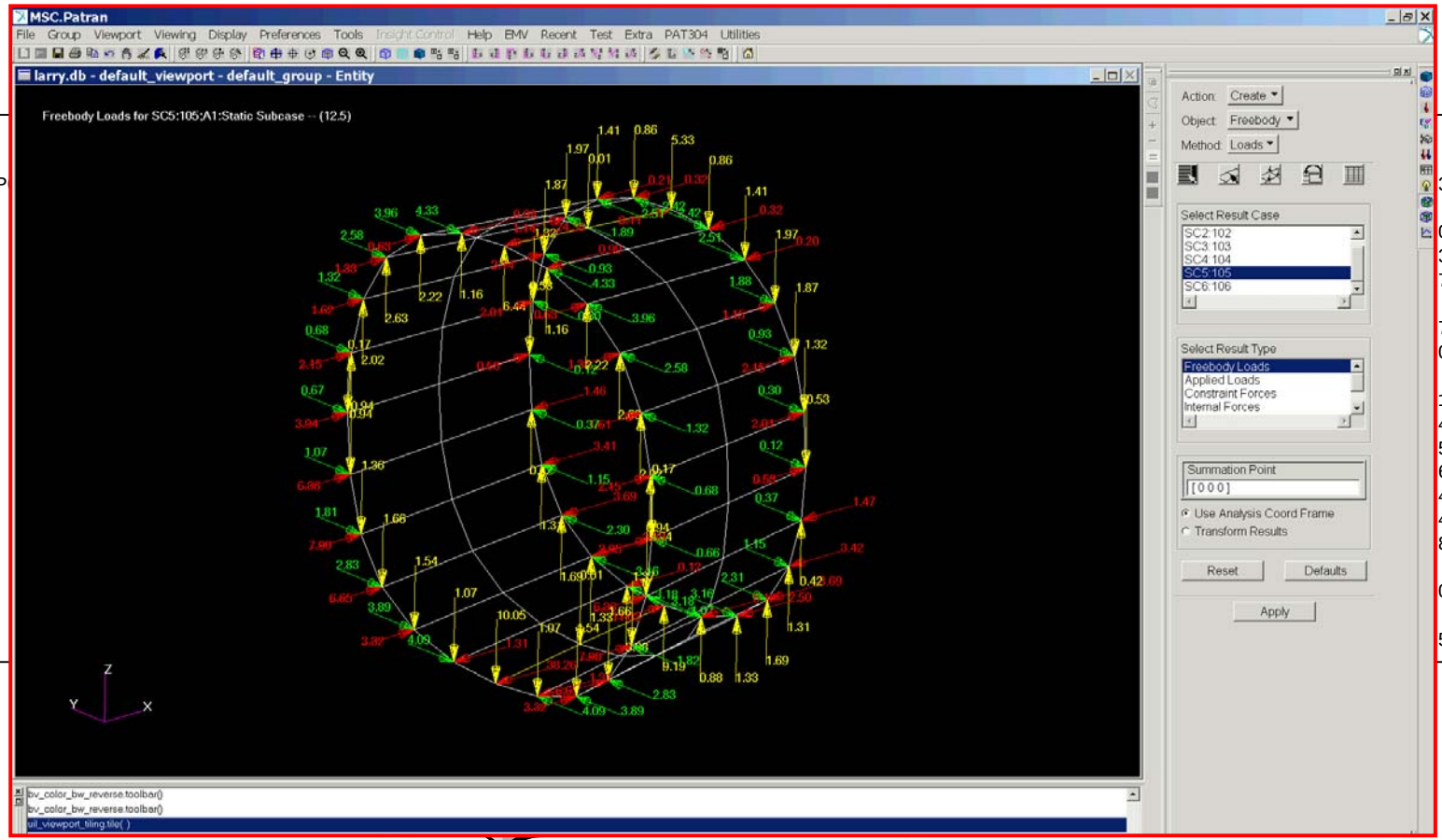
MSC.Patran's Freebody Tool

Isaac Newton's First and Favorite

Created: 6/15/2005
Updated: 8/8/2005

Freebody Tool

- **Designed to provide an intuitive interface to MSC.Nastran's Grid Point Force Balance data**

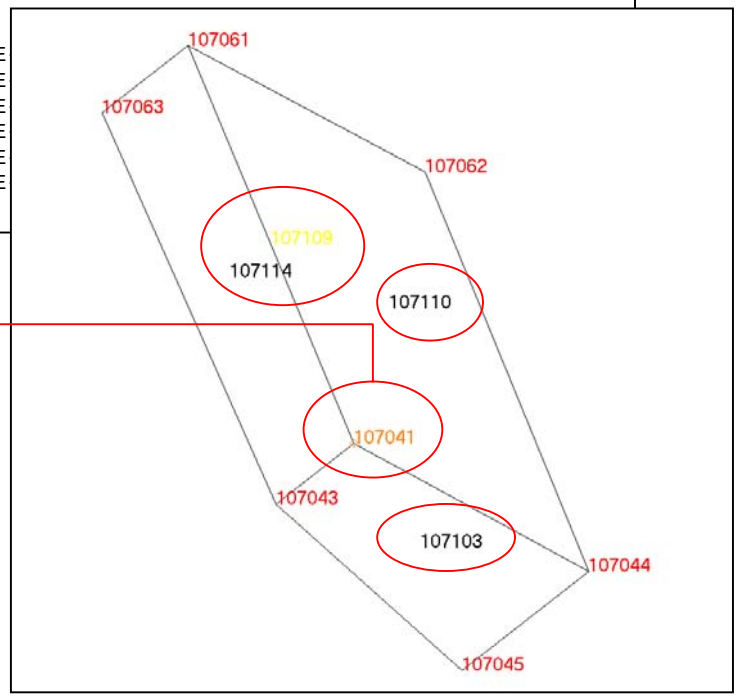


Grid Point Force Balance Primer

- **MSC.Nastran's Grid Point Force Balance table shows the forces and moments acting on the grid point from each source (element, applied load, etc.) in the MSC.Nastran global coordinate system**

G R I D P O I N T F O R C E B A L A N C E

	POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3
0	107041		APP-LOAD	-8.350770E-30	-5.811717E-30	-5.421011E
	107041	107103	QUAD4	2.474201E-20	-8.693462E-22	7.402629E
	107041	107110	QUAD4	-9.191307E-21	-1.823814E-19	-1.646739E
	107041	107114	QUAD4	-1.555071E-20	3.505370E-19	-4.949167E
	107041	107109	ROD	0.0	-1.672863E-19	1.600372E
	107041		*TOTALS*	4.130524E-30	-6.819188E-30	-1.624773E



Contributions Included

- Applied loads
- SPC forces
- Element elastic forces
- GENEL forces
- DMIG referenced by K2GG case control command
- Thermal loads
- MPC and rigid element forces

Grid Point Force Balance Primer

- Contribution from *applied loads*
- Contribution from an *element*
- Contribution from *MPC* or *rigid element*
- Contribution from *SPC*

G R I D P O I N T F O R C E B A L A N C E

	POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
0	107041		APP-LOAD	-8.350770E-30	-5.811717E-30	-5.421011E-20	0.0	0.0	0.0
	107041	107103	QUAD4	2.474201E-20	-8.693462E-22	7.402629E-20	5.381035E-22	3.810127E-23	2.602805E-21
	107041	107110	QUAD4	-9.191307E-21	-1.823814E-19	-1.646739E-20	-6.675039E-22	-8.717337E-21	2.683731E-22
	107041	107114	QUAD4	-1.555071E-20	3.505370E-19	-4.949167E-21	1.294004E-22	8.679235E-21	-2.871178E-21
	107041	107109	ROD	0.0	-1.672863E-19	1.600372E-21	0.0	0.0	0.0
	107041		*TOTALS*	4.130524E-30	-6.819188E-30	-1.624773E-29	-2.450033E-30	1.329619E-29	1.311077E-31
0	107042		APP-LOAD	-1.070619E-30	-6.617446E-24	-6.776264E-21	5.169877E-26	-4.235165E-22	8.271807E-25
	107042		F-OF-MPC	-6.783340E-19	0.0	-1.865601E-18	0.0	0.0	0.0
	107042	109222	BAR	-2.400134E-19	-3.364523E-20	2.476006E-19	-6.872330E-22	5.197320E-21	-7.388117E-22
	107042	109223	BAR	4.462914E-19	3.903186E-20	6.085369E-19	8.753099E-22	-4.809967E-21	-6.182948E-22
	107042	109230	BAR	-2.500841E-19	-4.006281E-20	2.368622E-19	6.901442E-22	5.235637E-22	7.358053E-22
	107042	109231	BAR	4.387695E-19	3.453384E-20	6.139055E-19	-8.708236E-22	-4.858531E-22	6.214862E-22
	107042	107115	TRIA3	2.833706E-19	1.489456E-22	1.586956E-19	-7.449185E-24	-1.546972E-24	-1.012245E-24
	107042		*TOTALS*	9.648221E-29	9.531467E-30	-6.776263E-21	3.510111E-31	-4.328102E-31	-1.439540E-30
0	108197		APP-LOAD	-8.960055E-30	-1.132316E-30	-5.421011E-20	2.710505E-20	-2.117582E-22	-3.657681E-30
	108197		F-OF-SPC	0.0	0.0	0.0	0.0	-2.947410E-24	0.0
	108197	110608	BAR	1.370690E-18	-1.830393E-16	8.577942E-18	-2.710505E-20	2.147056E-22	3.660808E-30
	108197	108801	ROD	-1.370690E-18	1.830393E-16	-8.523731E-18	0.0	0.0	0.0
	108197		*TOTALS*	5.488900E-32	6.656014E-31	-2.617724E-30	-2.198371E-30	1.573841E-32	3.126456E-33

Totals **is the summation of all included contributions. If the total is not identically *zero* it is due to round-off error *or* sources that are ignored, i.e., slideline forces, etc.**

Grid Point Force Balance Primer

- **GPFORCE is the case control request for grid point force balance**
- **Usage**
  - **GPFORCE = ALL**
  - **GPFORCE = n**
    - **n is the set identification number of a previously appearing SET command. Only grid points identified by the SET will be included in grid point force balance output**

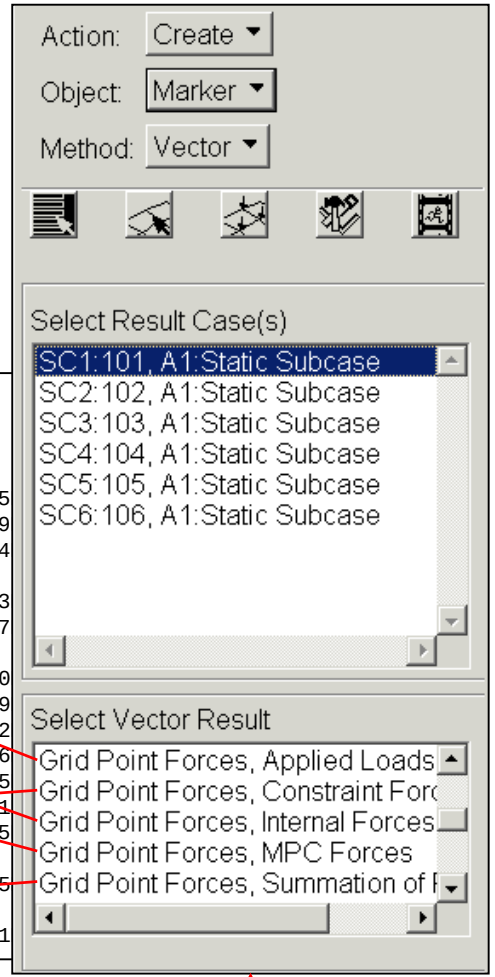
```
SUBCASE 2
   SUBTITLE=Loaded for bear
   LOAD = 4
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   MPCFORCES(SORT1,REAL)=ALL
   GPFORCE=ALL
   STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
   FORCE(SORT1,REAL,BILIN)=ALL
   OLOAD=ALL
```

Data Storage in MSC.Patran

- **GPFORCE data is stored as nodal and element vector quantities**
- **Can be viewed via other Result Plot Types, i.e., Create/Marker/Vector**

G R I D P O I N T F O R C E B A L A N C E

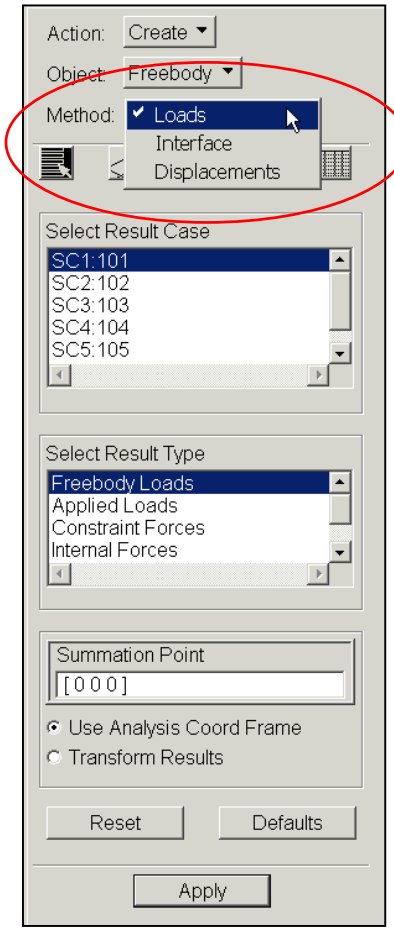
	POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1
0	107041		APP-LOAD	-8.350770E-30	-5.811717E-30	-5.421011E-20	0.0
	107041	107103	QUAD4	2.474201E-20	-8.693462E-22	7.402629E-20	5.381035
	107041	107110	QUAD4	-9.191307E-21	-1.823814E-19	-1.646739E-20	-6.675039
	107041	107114	QUAD4	-1.555071E-20	3.505370E-19	-4.949167E-21	1.294004
	107041	107109	ROD	0.0	-1.672863E-19	1.600372E-21	0.0
	107041		*TOTALS*	4.130524E-30	-6.819188E-30	-1.624773E-29	-2.450033
0	107042		APP-LOAD	-1.070619E-30	-6.617446E-24	-6.776264E-21	5.169877
	107042		F-OF-MPC	-6.783340E-19	0.0	-1.865601E-18	0.0
	107042	109222	BAR	-2.400134E-19	-3.364523E-20	2.476006E-19	-6.872330
	107042	109223	BAR	4.462914E-19	3.903186E-20	6.085369E-19	8.753099
	107042	109230	BAR	-2.500841E-19	-4.006281E-20	2.368622E-19	6.901442
	107042	109231	BAR	4.387695E-19	3.453384E-20	6.139055E-19	-8.708236
	107042	107115	TRIA3	2.833706E-19	1.489456E-22	1.586056E-19	-7.449185
	107042		*TOTALS*	9.648221E-29	9.531467E-30	-6.776263E-21	3.510111
0	108197		APP-LOAD	-8.960055E-30	-1.132316E-30	-5.421011E-20	2.710505
	108197		F-OF-SPC	0.0	0.0	0.0	0.0
	108197	110608	BAR	1.370690E-18	-1.830393E-16	8.577942E-18	-2.710505
	108197	108801	ROD	-1.370690E-18	1.830393E-16	-8.523731E-18	0.0
	108197		*TOTALS*	5.488900E-32	6.656014E-31	-2.617724E-30	-2.198371

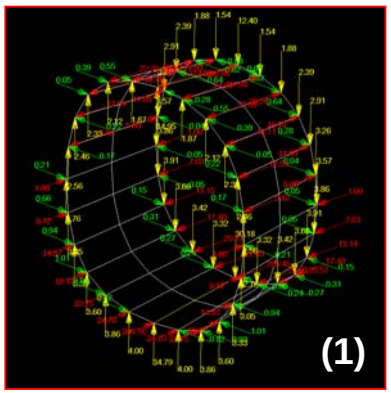


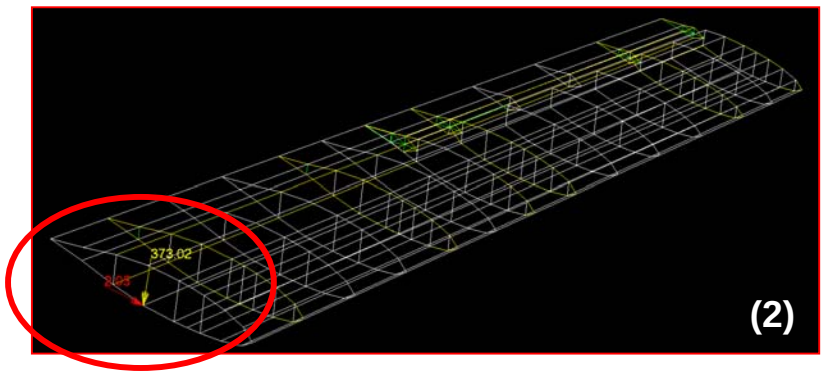
Grid Point Moments are stored in a similar fashion as Grid Point Forces

What Does the Freebody Tool Do?

- **The freebody tool performs 3 distinct functions which are represented by 3 Methods**
  1) **Loads – Displays a "freebody" of the structure based on all internal/external loads or just the applied loads or just the constraint loads, etc.**
  2) **Interface – Plots net loads at structure interfaces**
  3) **Displacements – Facilitates global/local modeling by creating a discrete FEM field of displacements at the freebody boundary**

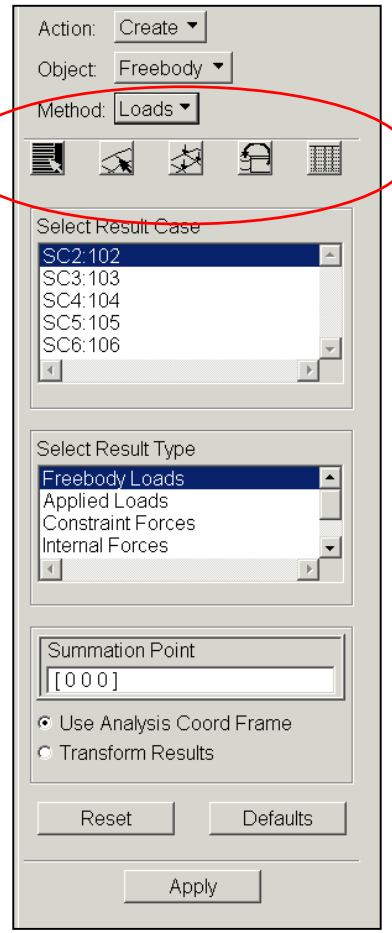






Freebody Loads Plot

- Form layout is similar to other result plot types
- Select Results
  - Select one or multiple result cases
  - Select result type
  - Etc.
- Select Entities
  - Select target elements. These represent the elements which will become the “freebody.”
- Display Attributes
  - Plot forces, moments, or both
  - Resultants or components
  - Vector scale, label options
  - Etc.
- Save Data
  - Saves freebody forces/moments as LBC Set
- Show Spreadsheet
  - Forces and moments at each node
  - Shows summation. Should be ZEROs if the “freebody” is in equilibrium.



Result Types

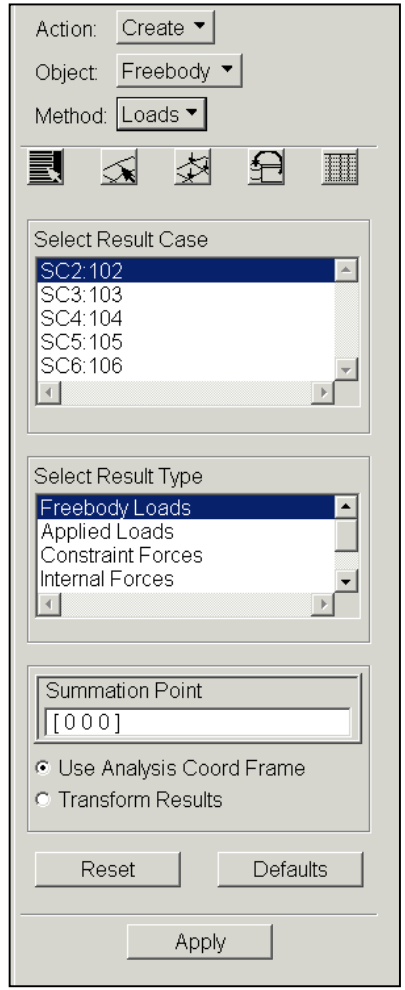
- At each node, the grid point force balance table includes contributions from elements, applied loads, SPCs, and MPCs, or

$$F_{Total} = \Sigma(F_{elms}) + F_{Applied} + F_{SPC} + F_{MPC}$$

G R I D P O I N T F O R C E B A L A N C E

	POINT-ID	ELEMENT-ID	SOURCE	T1
0	108197		APP-LOAD	-8.960055E-30
	108197		F-OF-SPC	0.0
	108197	110608	BAR	1.370690E-18
	108197	108801	ROD	-1.370690E-18
	108197		*TOTALS*	5.488900E-32

- These nodal contributions form the basis for the Result Type selections in the Freebody Tool
  - Freebody Loads — $-\Sigma(F_{elms})$ = - Internal Forces
  - Applied Loads — $F_{Applied}$
  - Constraint Forces — F_{SPC}
  - Internal Forces — $\Sigma(F_{elms})$
  - MPC Forces — F_{MPC}
  - Summation of Forces — F_{Total}

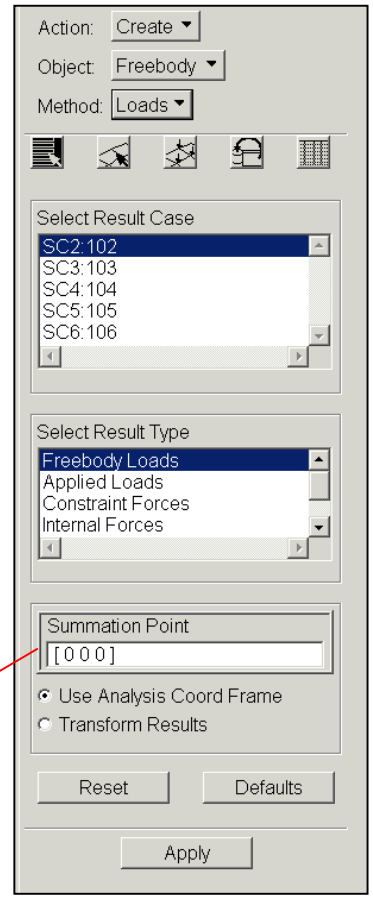


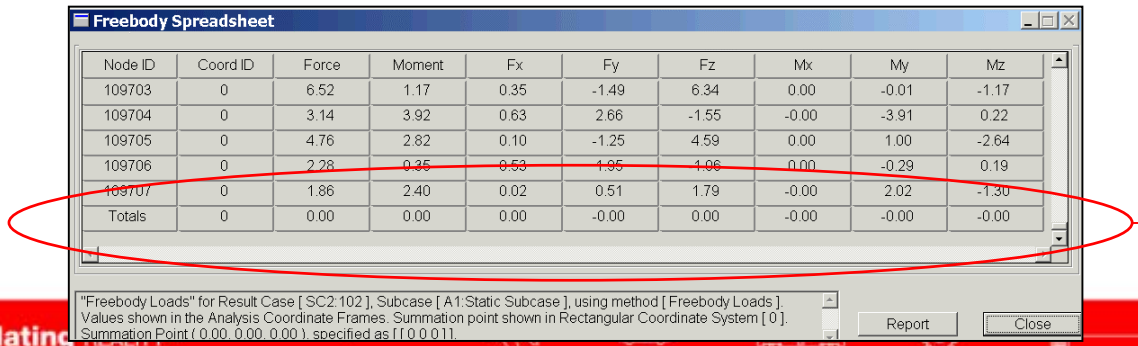
Result Types: Freebody Loads

- **Used to display a true freebody showing loads applied *to* the structure from all sources including the applied loads, constraints (SPCs), MPCs/rigid elements, and other sources (Totals)**

  Freebody Loads = $- \Sigma(F_{elms}) = F_{Applied} + F_{SPC} + F_{MPC} - F_{Total}$

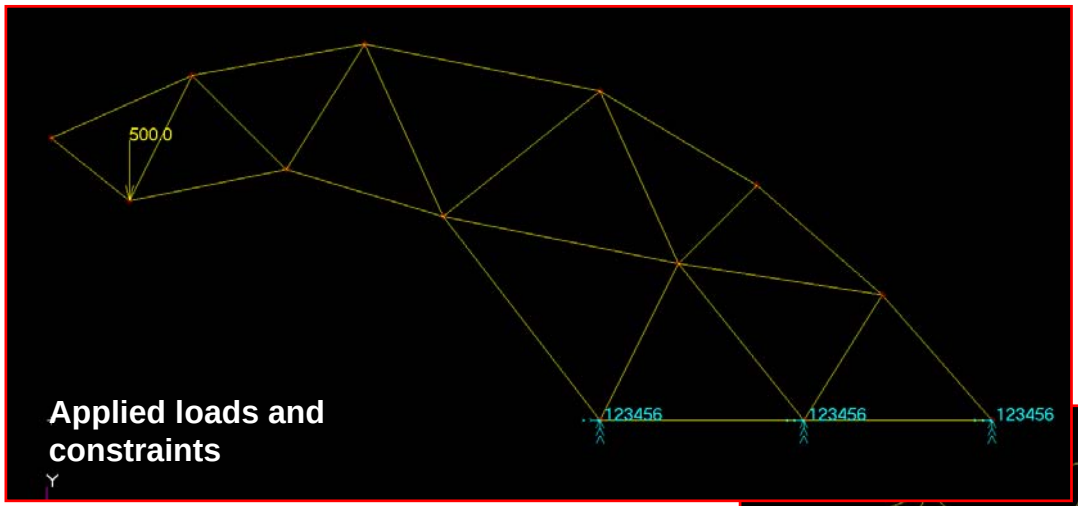
- **Equal to the negative of the Internal (or element) Forces**
- **The summation point is the point about which moments will be summed. Obviously for equilibrium, the sum of forces and moments about any point should be ZERO.**



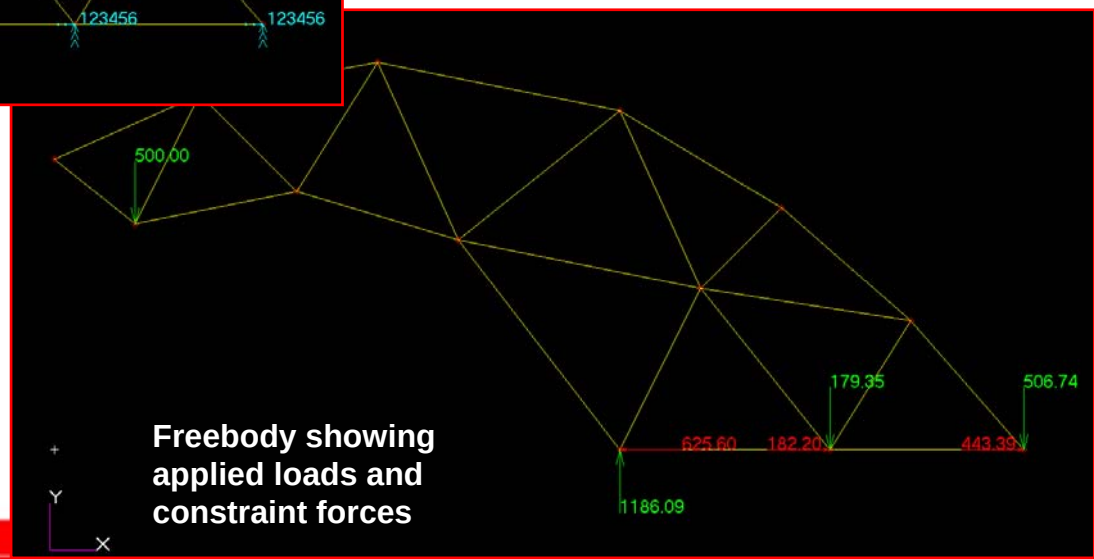


Freebody Loads: Application

- **Application: Load Path**
  - **Create freebody diagrams of various portions of model to determine the load going in and reacted out …**



Applied loads and constraints



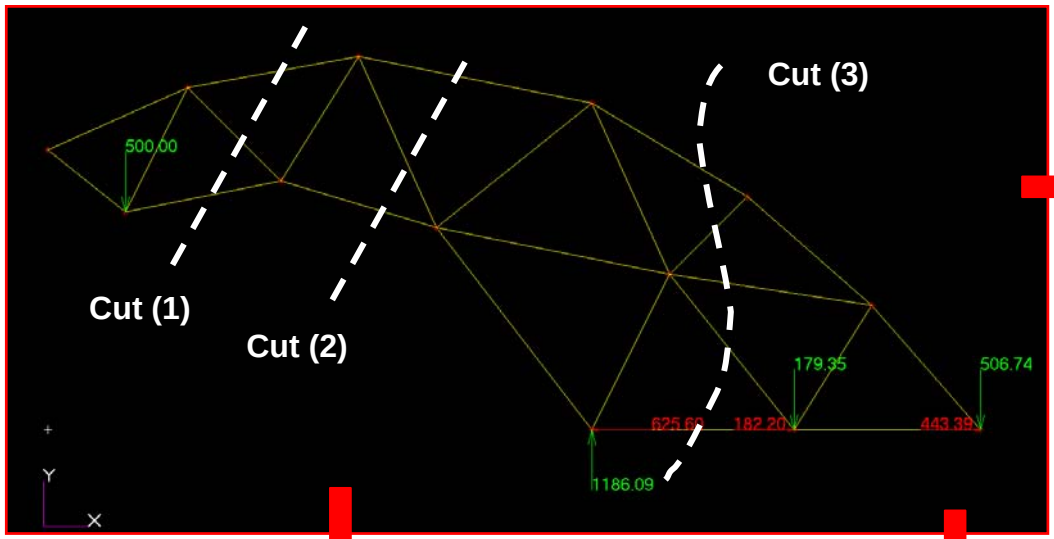
Freebody showing applied loads and constraint forces

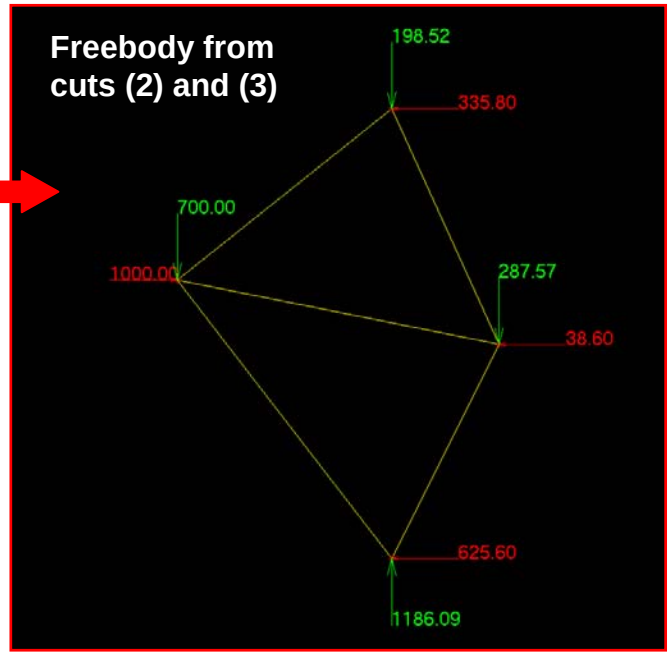
<u>NAS120 Stadium Roof Case Study</u>

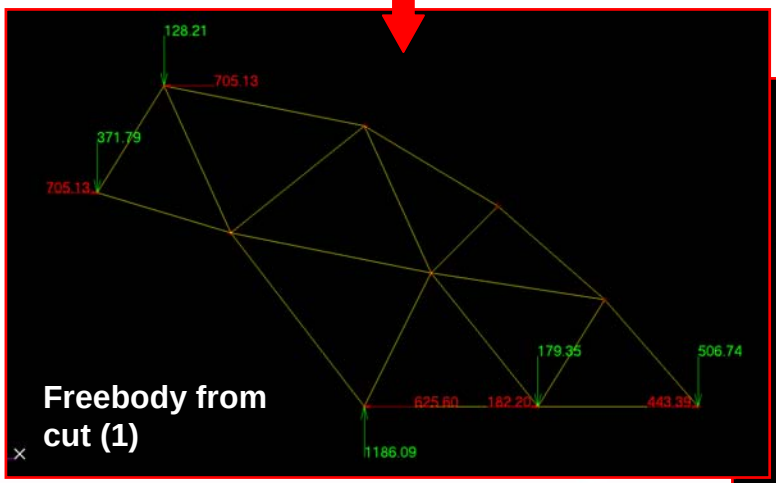
- **13 nodes**
- **24 CROD elements**

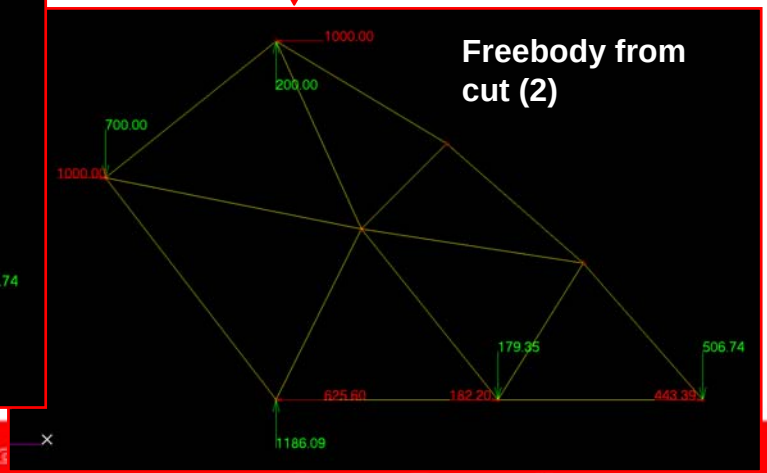
Freebody Loads: Application

- **Application: Load Path**



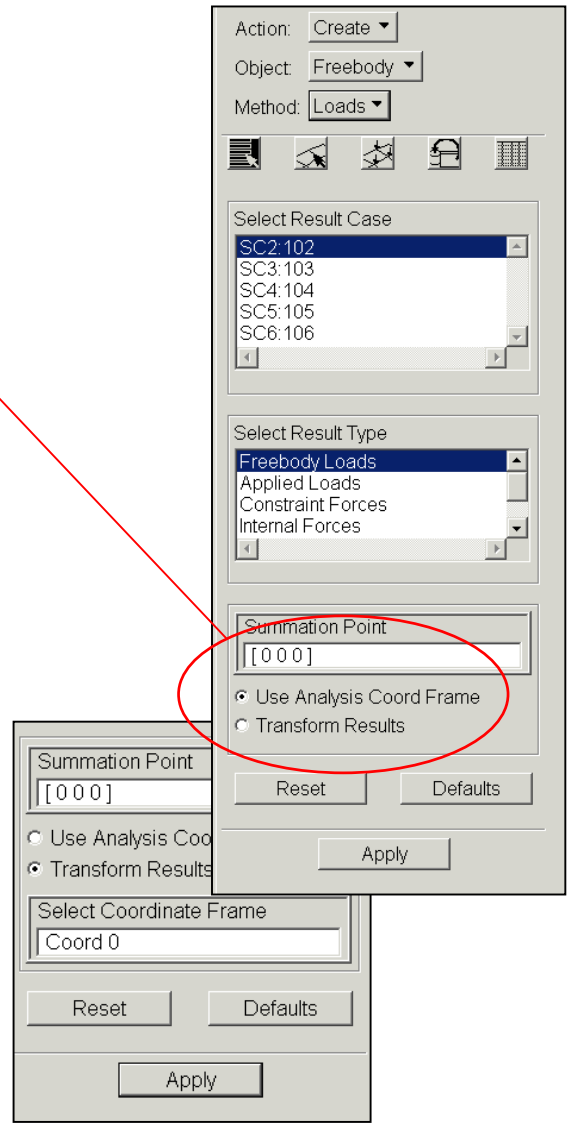




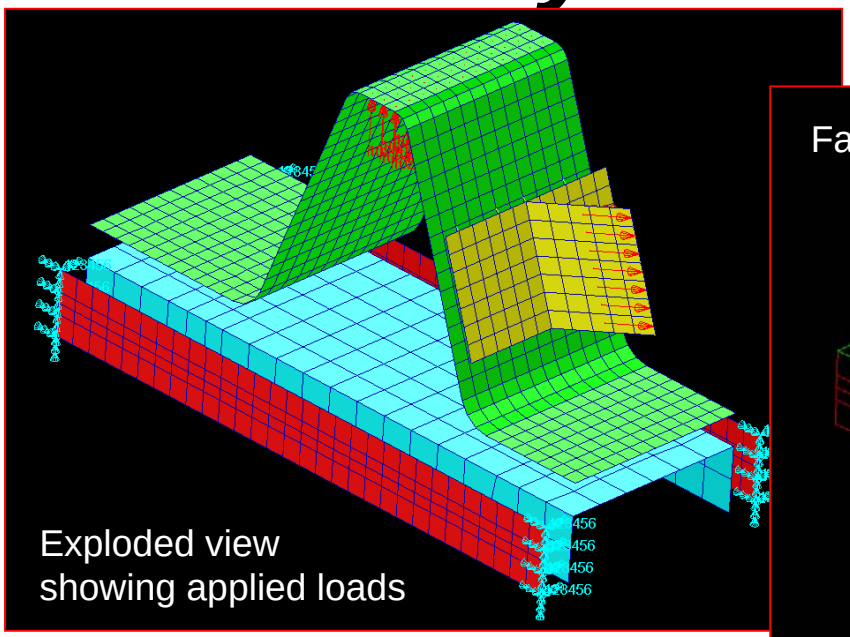


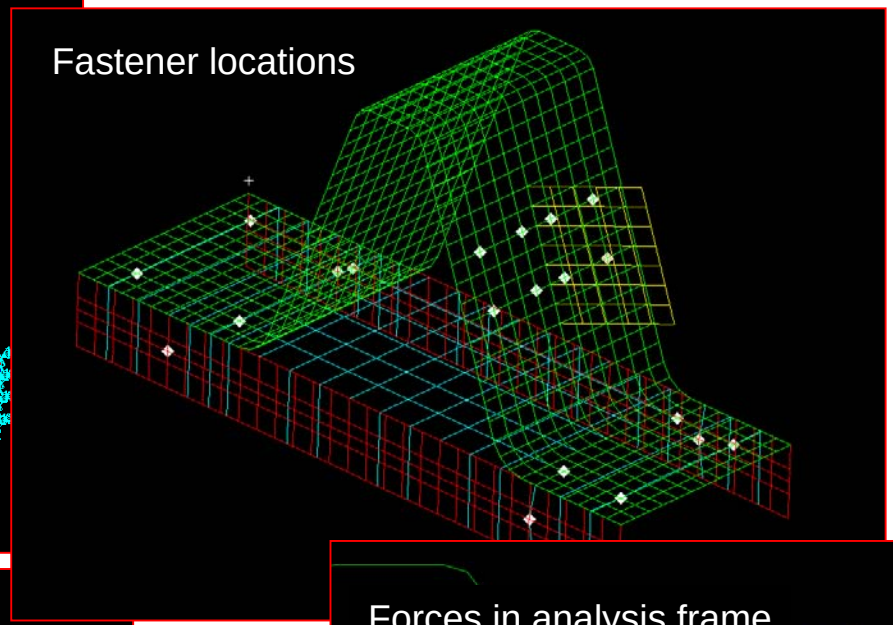
Freebody Loads: Application

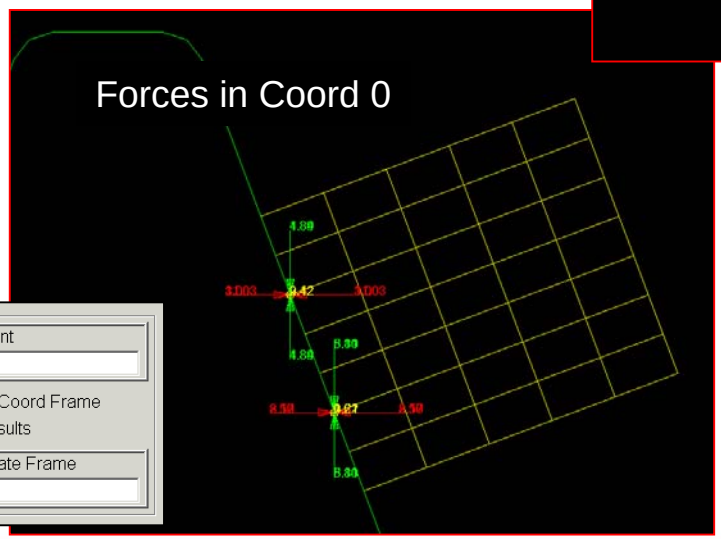
- Application: Fastener Forces
- Freebody forces/moments can be displayed in either
  - Analysis coordinate frame of the nodes
  - Or, any single coordinate system, i.e., Coord 0, Coord 87, etc.
- Displaying freebody data in the analysis coordinate frame can be useful for viewing fastener forces in a local fastener system.
  - Especially useful for fasteners modeled as CELAS elements

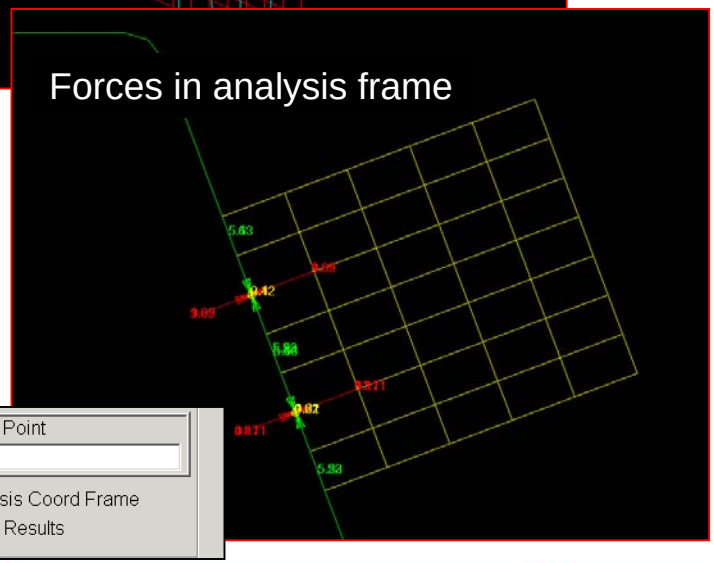


Freebody Loads: Application



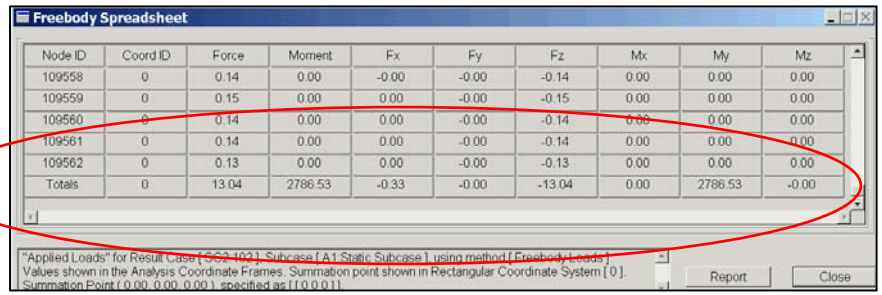




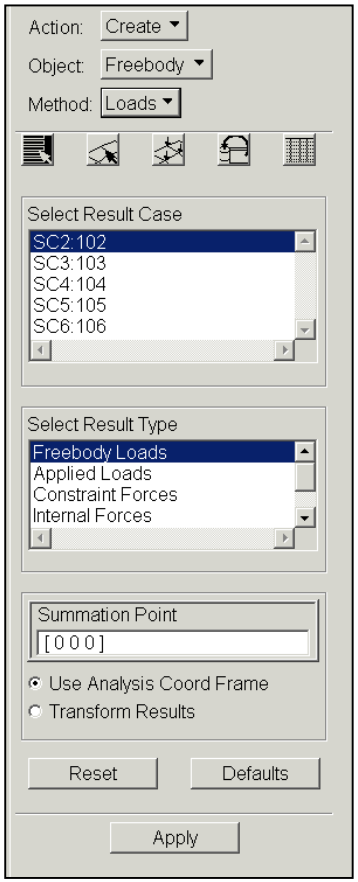


More Result Types

- **Applied Loads ($F_{Applied}$)**
  - Displays the applied loads acting on the target elements.
  - Not a “true” freebody since it does not include all loading.
  - The spreadsheet Totals row will sum to the total load applied to the freebody elements

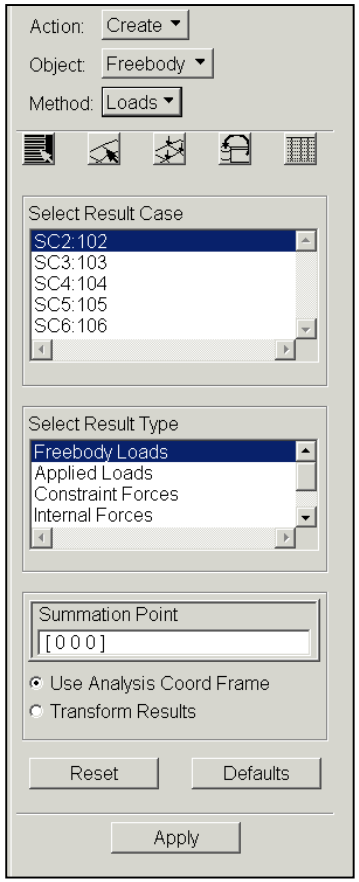


  - A similar display can be created by plotting MSC.Nastran OLOAD results (Applied Loads, Translational or Applied Loads, Rotational) via Create/Marker/Vector.
- **Constraint Forces (F_{SPC})**
  - Displays the constraint forces acting on the target elements.
  - Not a “true” freebody since it does not include all loading.
  - A similar display can be created by plotting MSC.Nastran SPCFORCE results (Constraint Forces, Translational or Constraint Forces, Rotational) via Create/Marker/Vector.



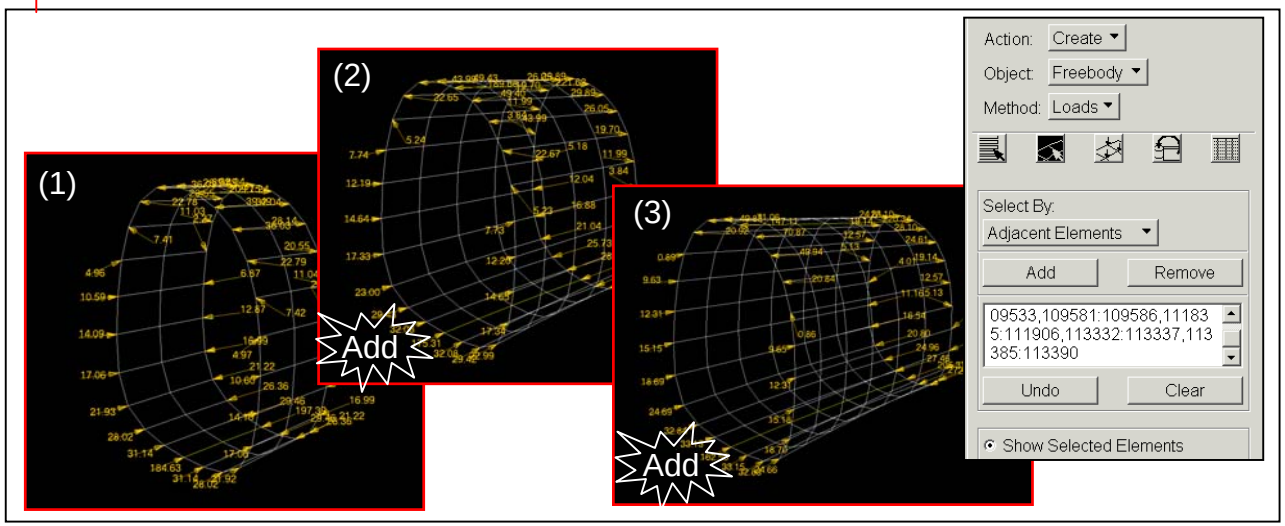
More Result Types

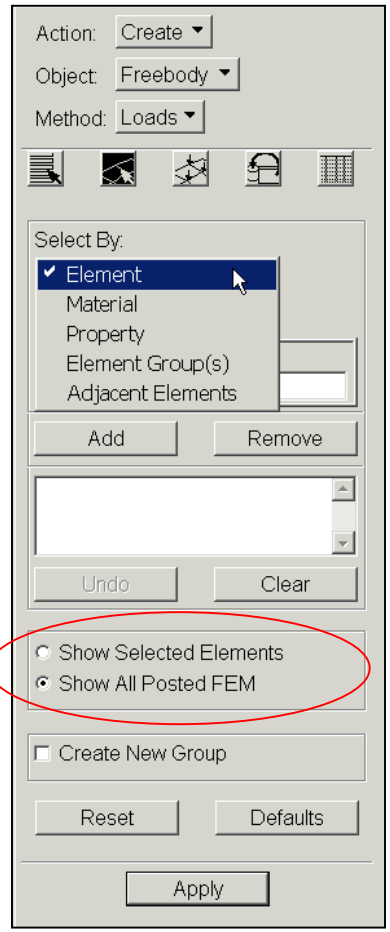
- MPC Forces (F_{MPC})
  - Displays forces applied to the target elements from MPCs or rigid elements
  - Not a “true” freebody since it does not include all loading.
  - A similar display can be created by plotting MSC.Nastran MPCFORCE results (MPC Constraint Forces, Translational or Rotational)
- Summation of Forces (F_{Total})
  - Displays forces applied to the target elements from non-supported or ignored sources
  - Typically, will be all ZEROs
  - Not a “true” freebody since it does not include all loading



Selecting Elements

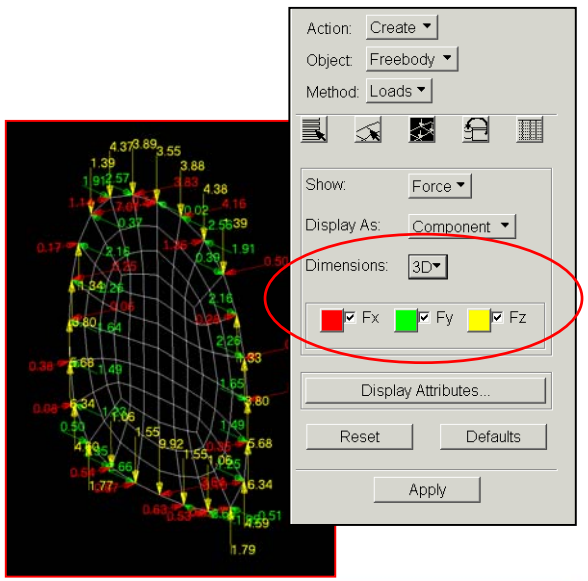
- Freebody elements can either be selected directly, by material or property association, and/or groups
- *Adjacent Elements* adds/removes elements to/from freebody based on model connectivity
- *Show Selected Elements* erases all elements except for those belonging to the freebody

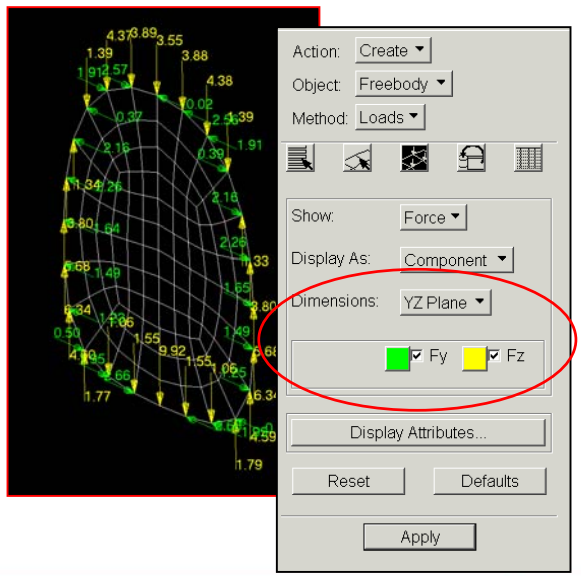


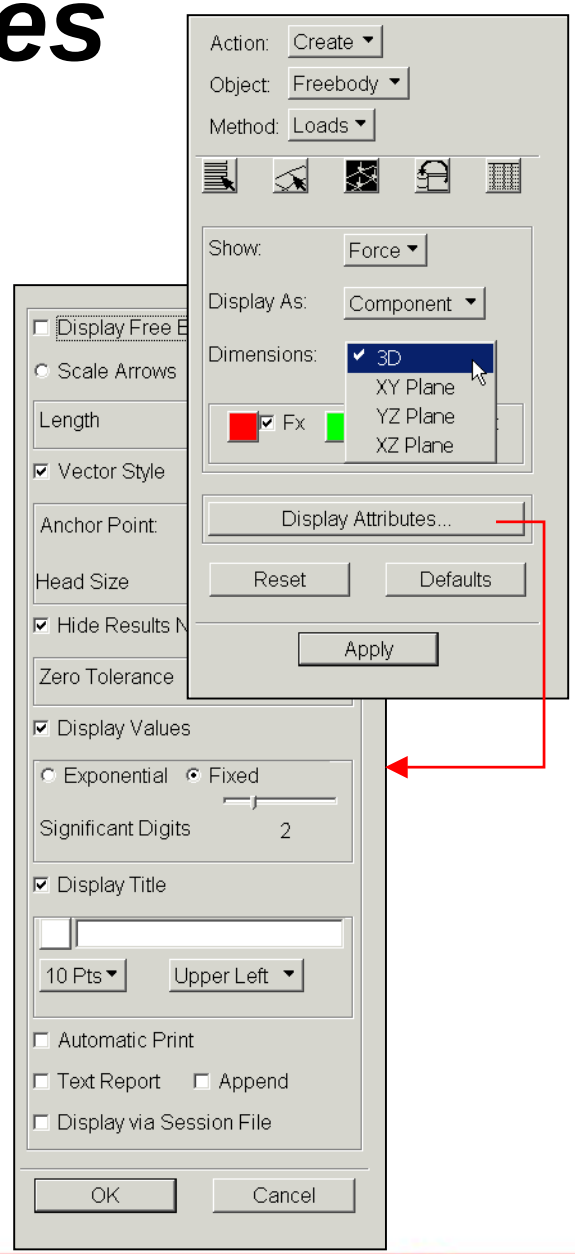


Display Attributes

- ***Show*** **force, moment or both together**
- ***Display as*** **resultant or components**
- ***Dimensions*** **allows data to be resolved to a particular plane**
  - **Useful to resolve loads into running loads along the edge of a bulkhead, stringer, etc.**

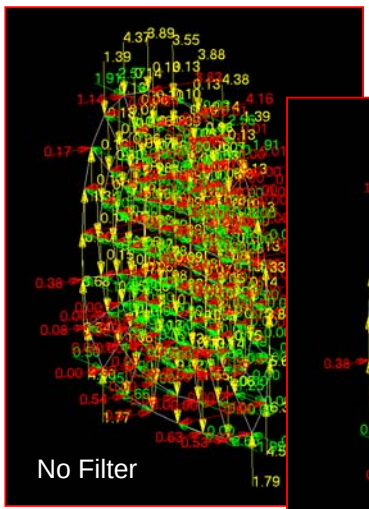


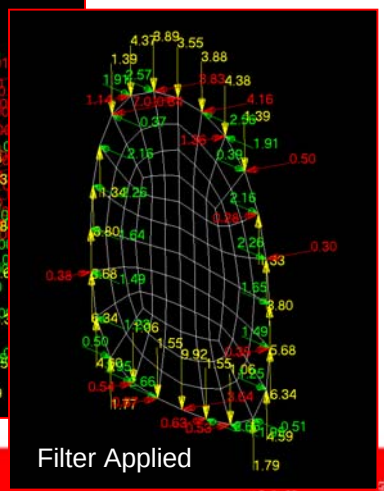


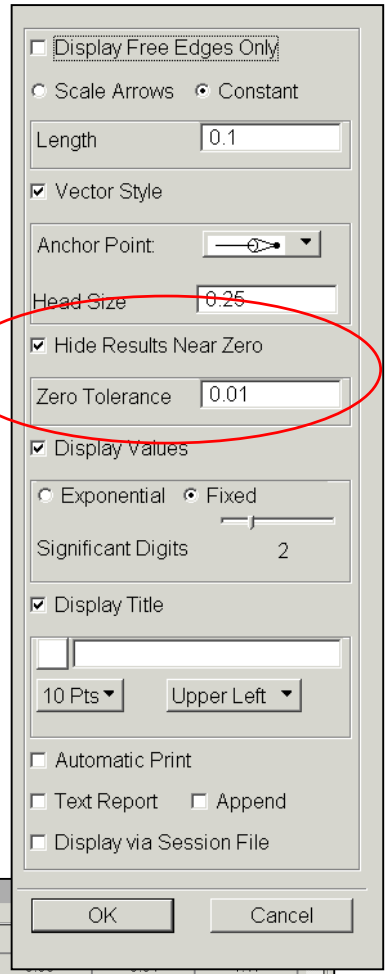


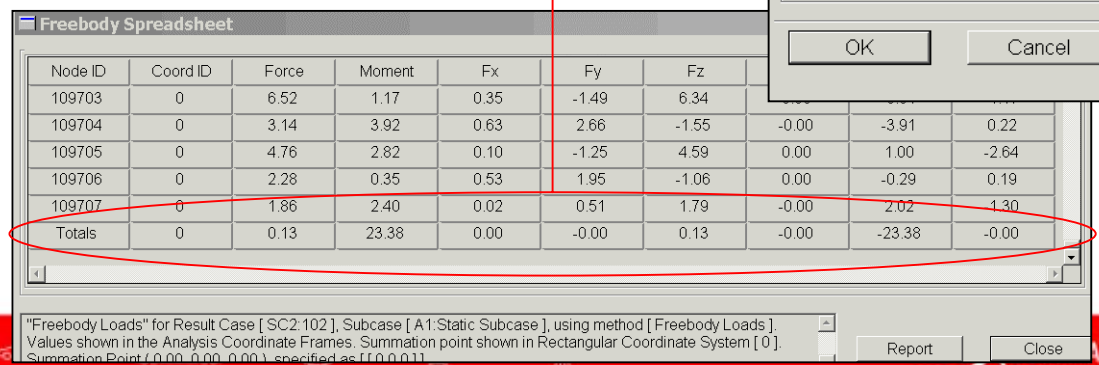
Display Attributes

- **Control vector scale, vector style, labeling, etc.**
- ***Hide results near zero* acts as a filter**
  - **Useful to unclutter the display**
  - ***Note that the spreadsheet sums the vectors that are displayed*. Hence, filtering data below a threshold will cause the summation to no longer be ZERO!**



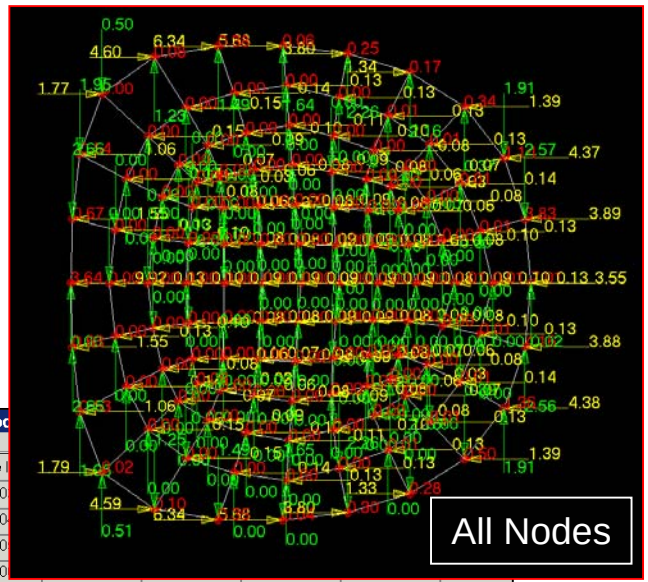






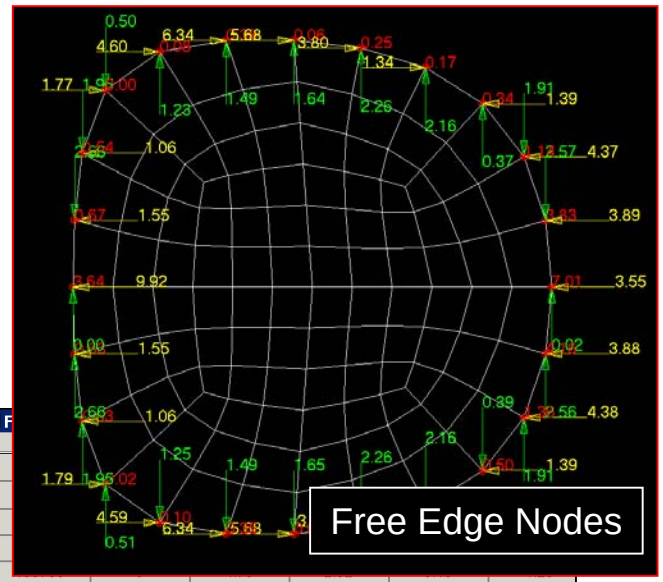
Display Attributes

- ***Display free edges only*** **reduces the vectors to those on free edges of the freebody**
  - **Visualize loading at freebody cut-edges**
  - ***Spreadsheet will not show a Totals row!***



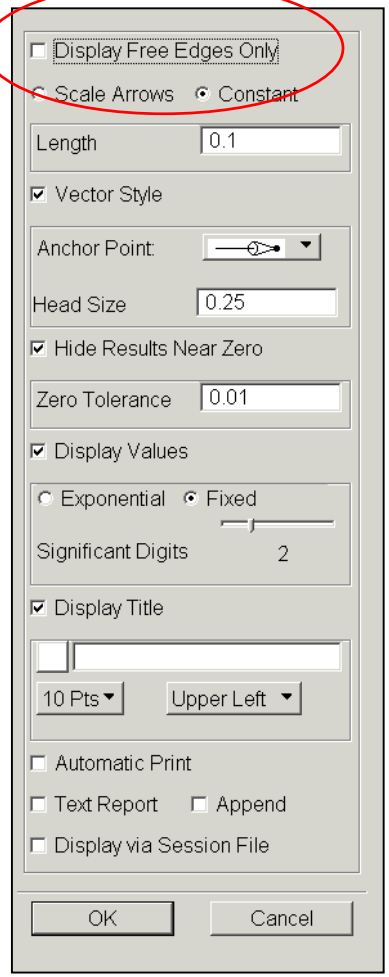
Node I					
10970					
10970					
10970					
10970					
109707	0	1.86	2.40	0.02	0.51
Totals	0	0.00	0.00	0.00	-0.00

"Freebody Loads" for Result Case [SC2:102], Subcase [A1:Static Subcase], using me
Values shown in the Analysis Coordinate Frames. Summation point shown in Rectangular
Summation Point (0.00, 0.00, 0.00), specified as [[0 0 0]].



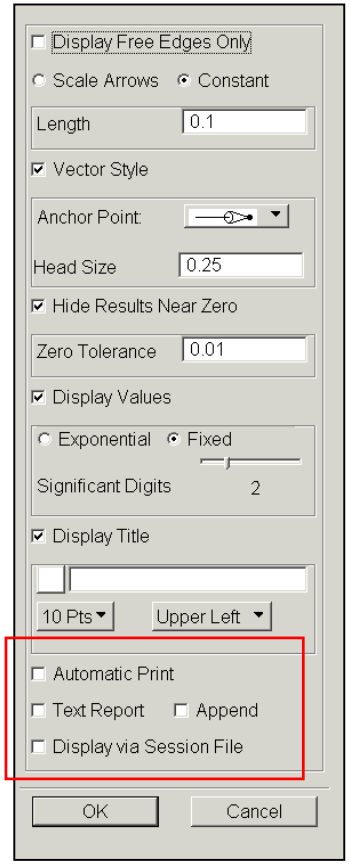
109706	0	2.28	0.35	0.53	1.95
109707	0	1.86	2.40	0.02	0.51

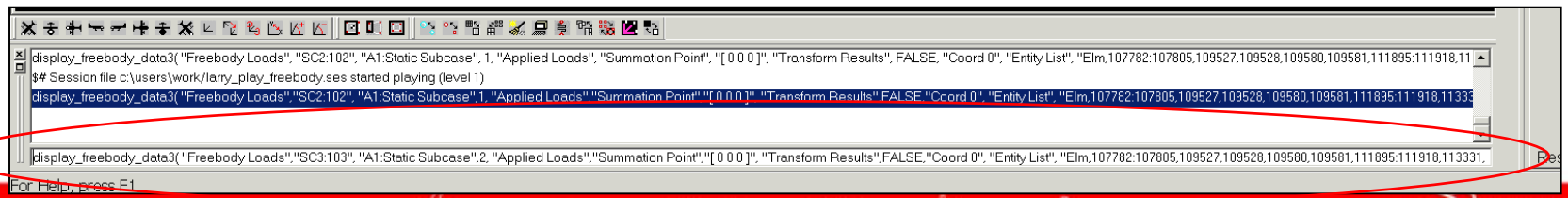
"Freebody Loads" for Result Case [SC2:102], Subcase [A1:Static Subcase], using metho
Values shown in the Analysis Coordinate Frames. Summation point shown in Rectangular C
Summation Point values not shown for Free Edge condition



Display Attributes

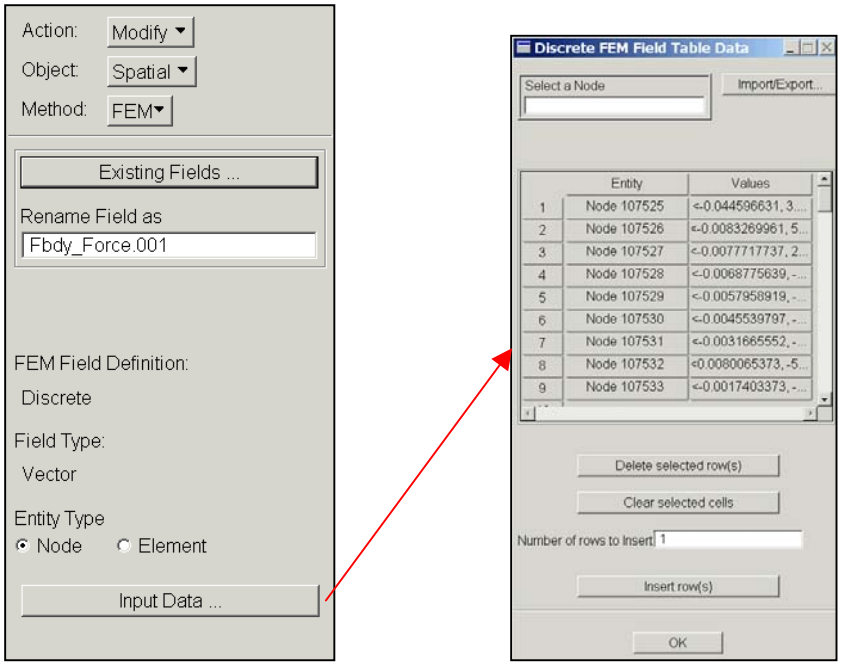
- ***Automatic Print*** **sends hardcopy to the currently selected printer**
  - **If multiple result cases are selected, then multiple plots will be generated**
  - **Does not work in batch mode with no graphics**
- ***Text Report*** **generates a delimited file of all data associated to the plot (<dbname>_freebody_data.dat)**
  - **Essentially the same data that is in the freebody spreadsheet**
- ***Display via Session File*** **effectively allows you to pause while you view a series of freebody plots**
  - **PCL function for each plot is placed in the command line**
  - **Plot is not displayed until a <CR> is issued with the cursor positioned in the command line**
  - **Also creates a session file called <dbname>_play_freebody.ses**

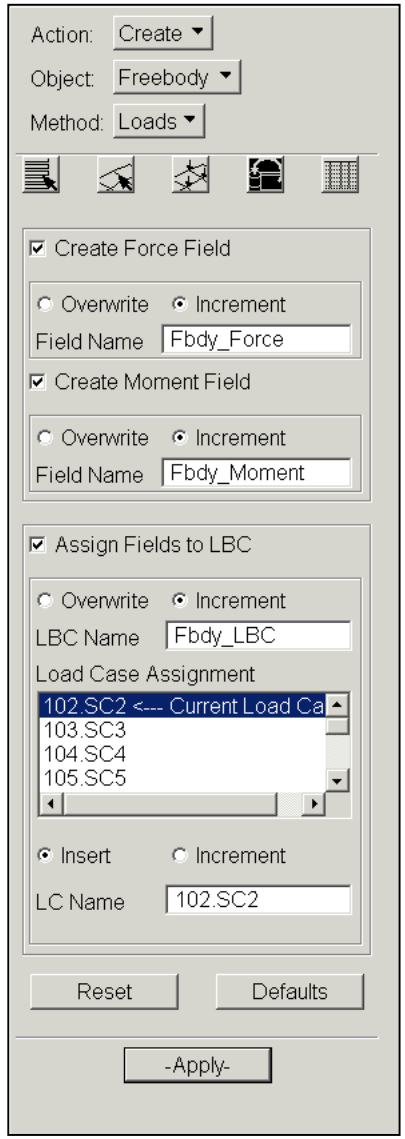




Save Data

- Saves the displayed data as a LBC Set
- Force and moment data is stored as discrete FEM (DFEM) fields, i.e., a table of node IDs vs. values

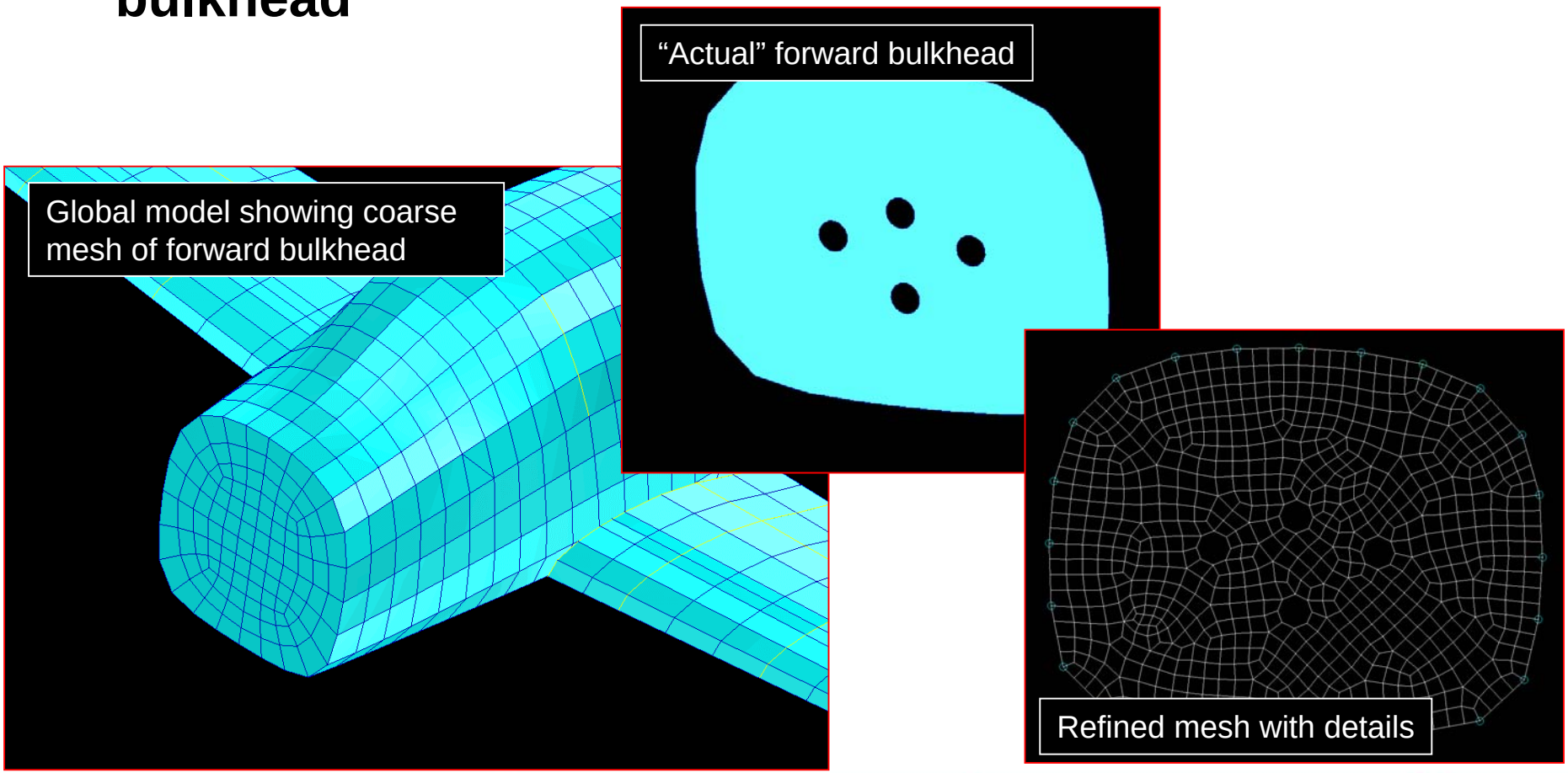




- Fields can optionally be assigned to LBC Sets and Loadcases

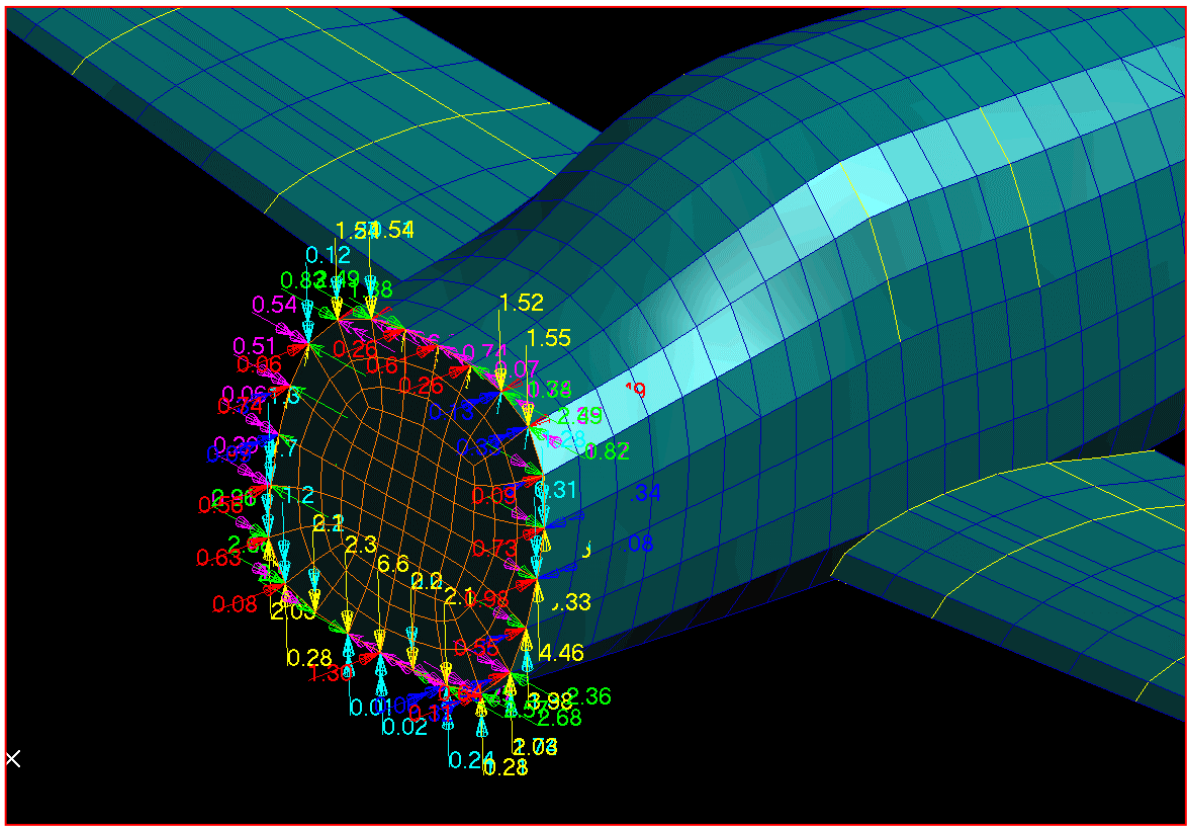
Save Data: Application

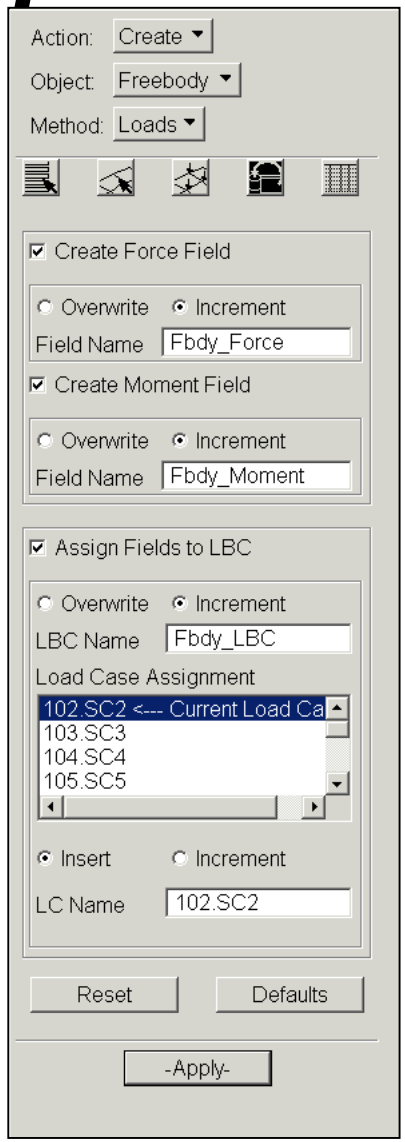
- **Global/local modeling**
- **Example: global loads model omitted details of bulkhead**



Save Data: Application

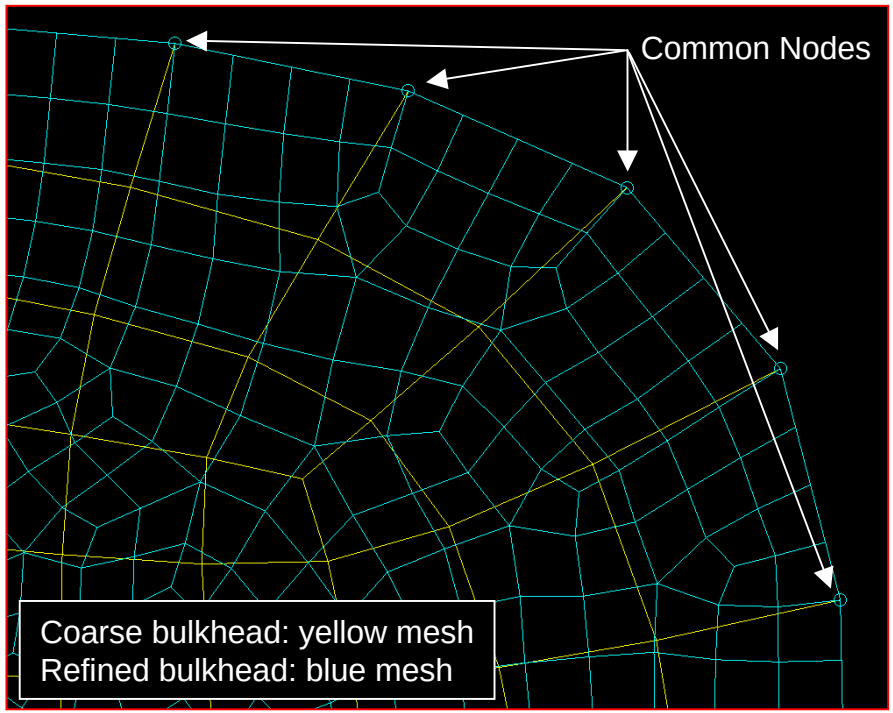
- Create freebody of bulkhead
- Save freebody forces/moments as DFEM fields and LBC set

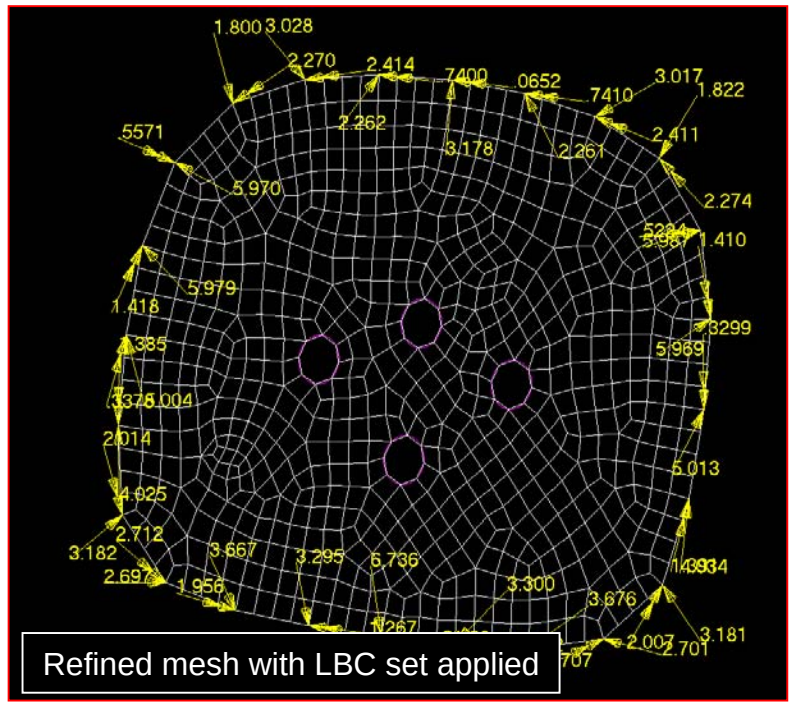




Save Data: Application

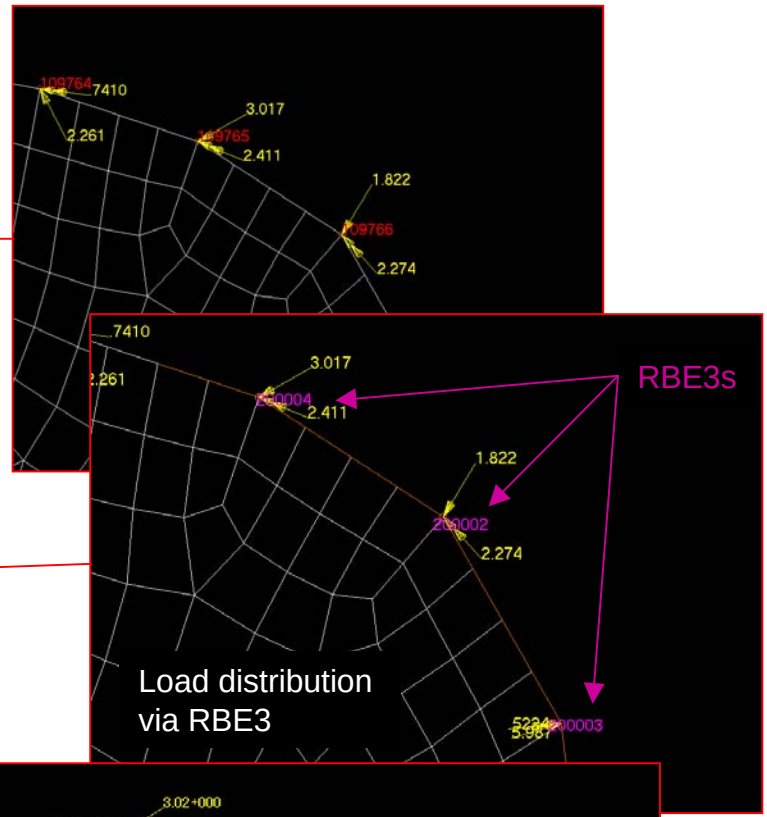
- **DFEM field is a table of node IDs vs. values**
- **MSC.Patran cannot interpolate within DFEM fields**
- **Hence, local bulkhead model and global bulkhead model must share common node IDs at load application points**

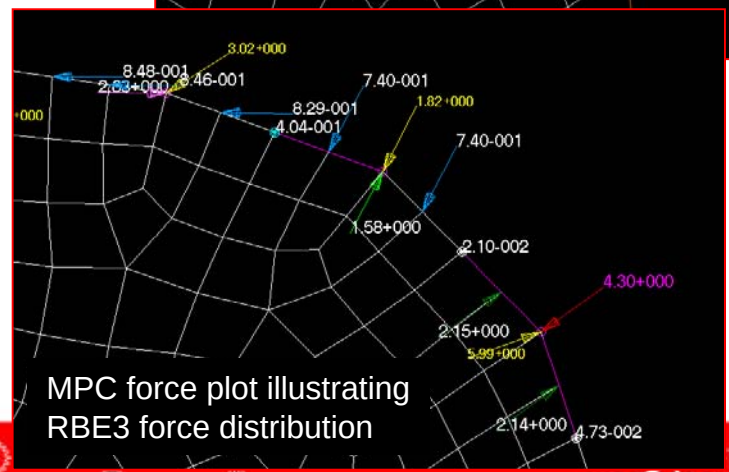




Save Data: Application

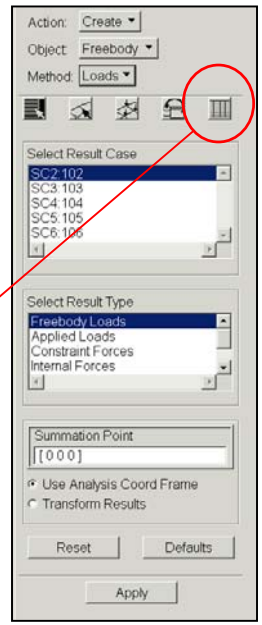
- **What to do about the "extra" boundary nodes in the refined model?**
  - ***Use company best practices!***
  - **If boundary is far removed from area of interest use St Venant's Principle**
  - **Use RBE3's to distribute load at boundary**
  - **Cannot distribute via fields as MSC.Patran cannot interpolate within a DFEM field**
  - **And, MSC.Patran cannot be used to interpolate forces and moments as it uses a purely mathematical interpolation and does not consider *equilibrium*!**





Freebody Spreadsheet

- **Reports the force/moment values for each node in the freebody display**
  - **Node ID**
  - **Coordinate reference for forces/moments**
  - **Force/moment resultants**
  - **Force/moment components**



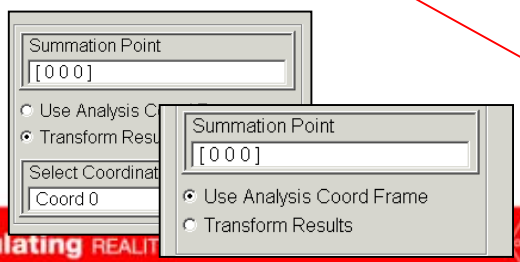
Freebody Spreadsheet

Node ID	Coord ID	Force	Moment	Fx	Fy	Fz	Mx	My	Mz
107044	0	0.35	0.11	0.27	-0.22	-0.05	0.01	0.11	-0.01
107045	0	0.07	0.01	0.06	0.00	0.04	-0.00	0.00	0.01
107061	0	0.47	0.05	-0.28	-0.36	0.09	-0.01	-0.05	0.01
107062	0	0.48	0.02	-0.22	-0.43	0.06	-0.01	-0.02	0.00
107063	0	0.52	0.05	0.08	0.50	0.08	-0.00	0.05	-0.01
Totals	0	0.00	0.00	-0.00	-0.00	0.00	0.00	0.00	0.00

"Freebody Loads" for Result Case [SC2:102], Subcase [A1:Static Subcase], using method [Freebody Loads].
Values shown in the Analysis Coordinate Frames. Summation point shown in Rectangular Coordinate System [0].
Summation Point [0.00 0.00 0.00] specified as [[0 0 0]]

Report | Close

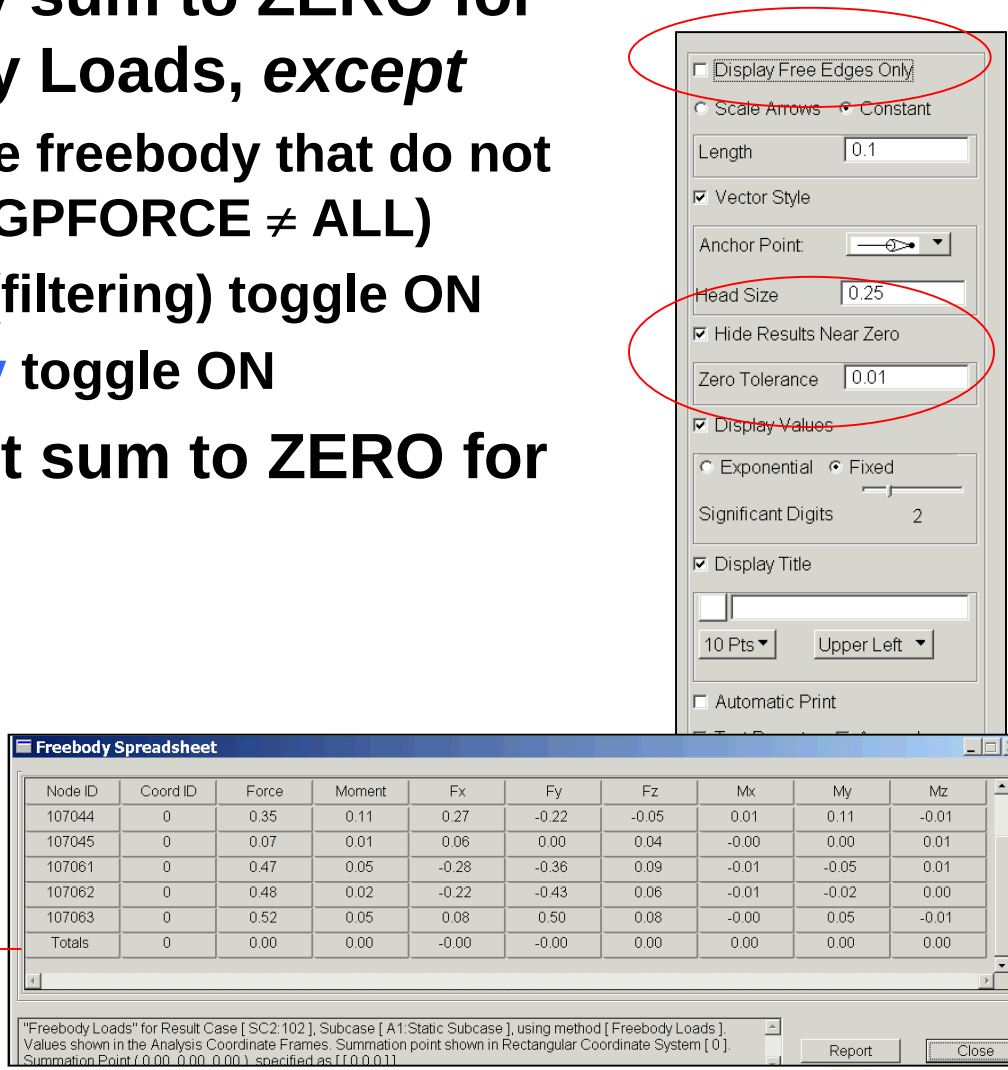
Delimited text file of spreadsheet data, <dbname>_freebody_data.dat



Summation values are for the selected coordinate frame unless the freebody is with respect to the analysis coordinate frames, then the summation is in the MSC.Patran global system (Coord 0) so that the summation is consistent

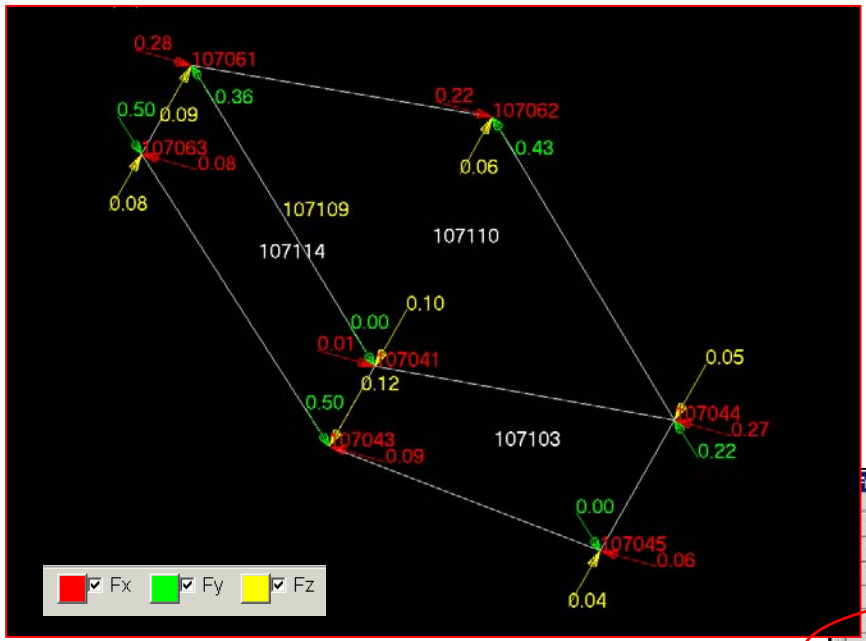
Freebody Spreadsheet

- **Totals should generally sum to ZERO for Result Type = Freebody Loads, *except***
  - **Including elements in the freebody that do not have freebody data (i.e, GPFORCE ≠ ALL)**
  - **Hide Results Near Zero (filtering) toggle ON**
  - **Display Free Edges Only toggle ON**
- **Totals will generally not sum to ZERO for other Result Types**
  - **Applied loads**
  - **Constraint forces**
  - **MPC forces**
  - **Summation of forces**



Freebody Spreadsheet: Example

- **Result Type = Freebody Loads**
  Freebody Loads = - $\Sigma(F_{elms}) = F_{Applied} + F_{SPC} + F_{MPC} - F_{Total}$
- **Hide Results Near Zero toggle OFF**
- **Display Free Edges Only toggle OFF**



Note that the Totals row sums to ZERO

Freebody Spreadsheet

Node ID	Coord ID	Force	Moment	Fx	Fy	Fz	Mx	My	Mz
107044	0	0.35	0.11	0.27	-0.22	-0.05	0.01	0.11	-0.01
107045	0	0.07	0.01	0.06	0.09	0.04	-0.00	0.00	0.01
107061	0	0.47	0.05	-0.28	-0.36	0.09	-0.01	-0.05	0.01
107062	0	0.48	0.02	-0.22	-0.43	0.06	-0.01	-0.02	0.00
107063	0	0.52	0.05	0.08	0.50	0.08	-0.00	0.05	-0.01
Totals	0	0.00	0.00	-0.00	-0.00	0.00	0.00	0.00	0.00

"Freebody Loads" for Result Case [SC2:102], Subcase [A1:Static Subcase], using method [Freebody Loads]
Values shown in the Analysis Coordinate Frames. Summation point shown in Rectangular Coordinate System [0].
Summation Point (0.00, 0.00, 0.00), specified as [0 0 0]

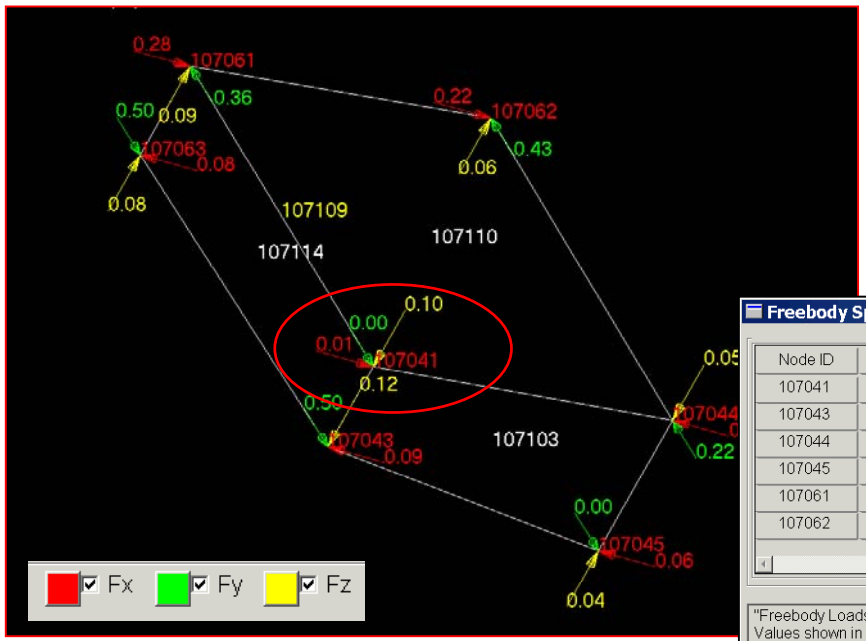
Report Close

Freebody Spreadsheet: Example

- **Spreadsheet data for node 1070401**

Freebody Loads = - $\Sigma(F_{elms})$ = $F_{Applied}$ + F_{SPC} + F_{MPC} - F_{Total}

0 0 0



Freebody Spreadsheet

Node ID	Coord ID	Force	Moment	Fx	Fy	Fz	Mx	My	Mz
107041	0	0.10	0.00	-0.01	0.00	-0.10	0.00	0.00	-0.00
107043	0	0.53	0.01	0.09	0.50	-0.12	-0.00	-0.01	-0.00
107044	0	0.35	0.11	0.27	-0.22	-0.05	0.01	0.11	-0.01
107045	0	0.07	0.01	0.06	0.00	0.04	-0.00	0.00	0.01
107061	0	0.47	0.05	-0.28	-0.36	0.09	-0.01	-0.05	0.01
107062	0	0.48	0.02	-0.22	-0.43	0.06	-0.01	-0.02	0.00

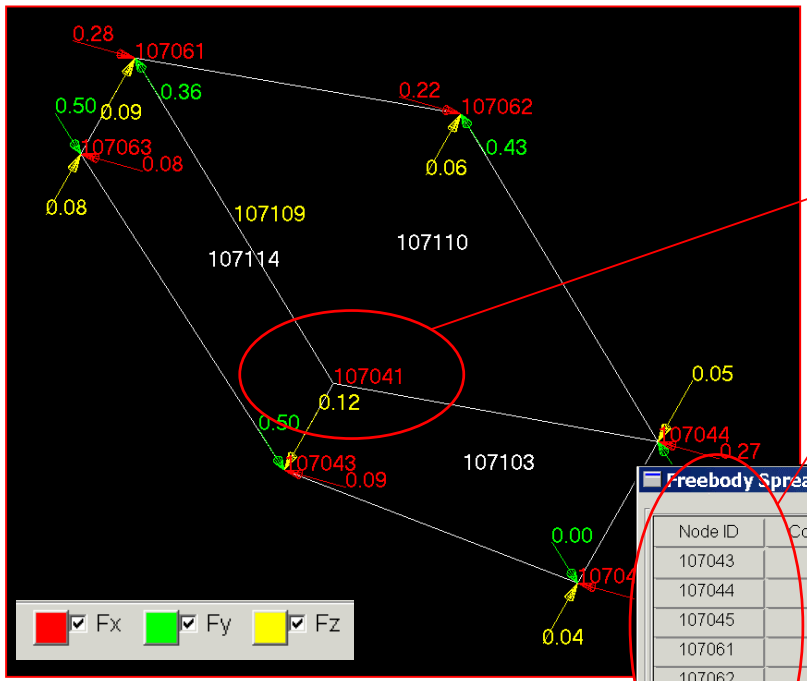
"Freebody Loads" for Result Case [SC2:102], Subcase [A1:Static Subcase], using method [Freebody Loads].
Values shown in the Analysis Coordinate Frames. Summation point shown in Rectangular Coordinate System [0].
Summation Point [0.00 0.00 0.00] specified as [[0 0 0]]

Report Close

0	107041		APP-LOAD	-6.010349E-03	-1.020161E-12	-1.003154E-01	0.0	0.0	0.0
	107041	107103	QUAD4	1.066716E-01	-2.293396E-03	3.910460E-02	6.676069E-04	7.492116E-05	5.054075E-03
	107041	107110	QUAD4	-8.056121E-02	-3.734471E-01	-2.654815E-02	-8.320150E-04	-1.100493E-02	4.014180E-04
	107041	107114	QUAD4	-2.010006E-02	5.016820E-01	8.655410E-02	1.644080E-04	1.093001E-02	-5.455493E-03
	107041	107109	ROD	0.0	-1.259415E-01	1.204840E-03	0.0	0.0	0.0
	107041		*TOTALS*	5.366818E-13	1.143696E-12	-1.253518E-11	-1.890638E-13	5.098838E-13	-2.846681E-15

Freebody Spreadsheet: Example

- **Result Type = Freebody Loads**
  Freebody Loads = - $\Sigma(F_{elms}) = F_{Applied} + F_{SPC} + F_{MPC} - F_{Total}$
- **Hide Results Near Zero toggle OFF**
- **Display Free Edges Only toggle ON**



- **Node 107041 is not on the free edge**
  - **No vector displayed in plot**
  - **No data in the spreadsheet**
- **No Totals row is displayed in the spreadsheet**

Freebody Spreadsheet

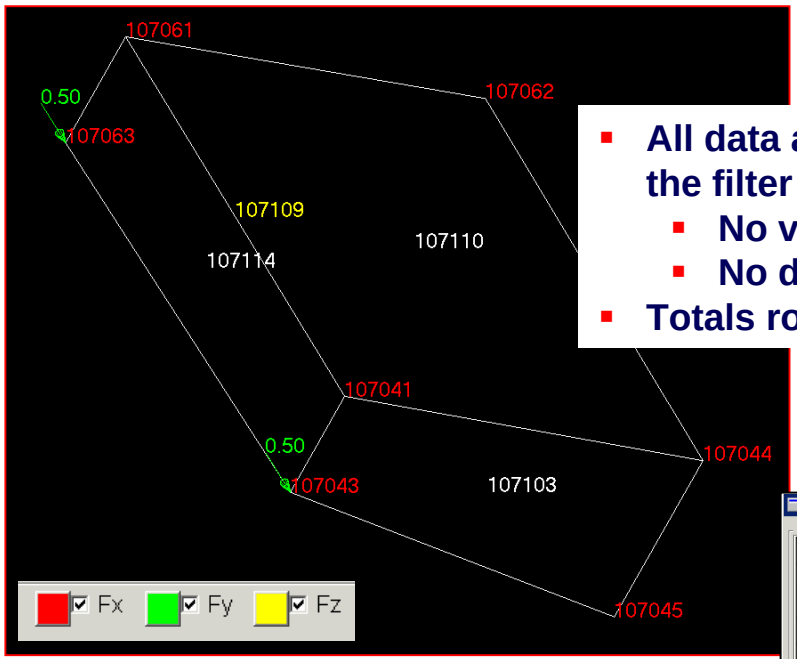
Node ID	Coord ID	Force	Moment	Fx	Fy	Fz	Mx	My	Mz
107043	0	0.53	0.01	0.09	0.50	-0.12	-0.00	-0.01	-0.00
107044	0	0.35	0.11	0.27	-0.22	-0.05	0.01	0.11	-0.01
107045	0	0.07	0.01	0.06	0.00	0.04	-0.00	0.00	0.01
107061	0	0.47	0.05	-0.28	-0.36	0.09	-0.01	-0.05	0.01
107062	0	0.48	0.02	-0.22	-0.43	0.06	-0.01	-0.02	0.00
107063	0	0.52	0.05	0.08	0.50	0.08	-0.00	0.05	-0.01

"Freebody Loads" for Result Case [SC2:102], Subcase [A1:Static Subcase], using method [Freebody Loads].
Values shown in the Analysis Coordinate Frames. Summation point shown in Rectangular Coordinate System [0].
Summation Point values not shown for Free Edge condition

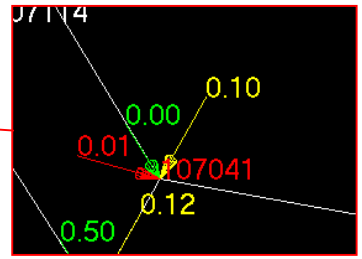
Report Close

Freebody Spreadsheet: Example

- **Result Type = Freebody Loads**
  Freebody Loads = - $\Sigma(F_{elms}) = F_{Applied} + F_{SPC} + F_{MPC} - F_{Total}$
- **Hide Results Near Zero toggle ON (0.5)**
- **Display Free Edges Only toggle OFF**



- **All data at Node 107041 is less than the filter value (0.5)**
  - **No vector displayed in plot**
  - **No data in the spreadsheet**
- **Totals row does not sum to ZERO!**



Freebody Spreadsheet

Node ID	Coord ID	Force	Moment	Fx	Fy	Fz	Mx	My	Mz
107043	0	0.53	0.01	0.09	0.50	-0.12	-0.00	-0.01	-0.00
107063	0	0.52	0.05	0.08	0.50	0.08	-0.00	0.05	-0.01
Totals	0	1.02	307.38	0.17	1.01	-0.04	-17.01	16.26	306.48

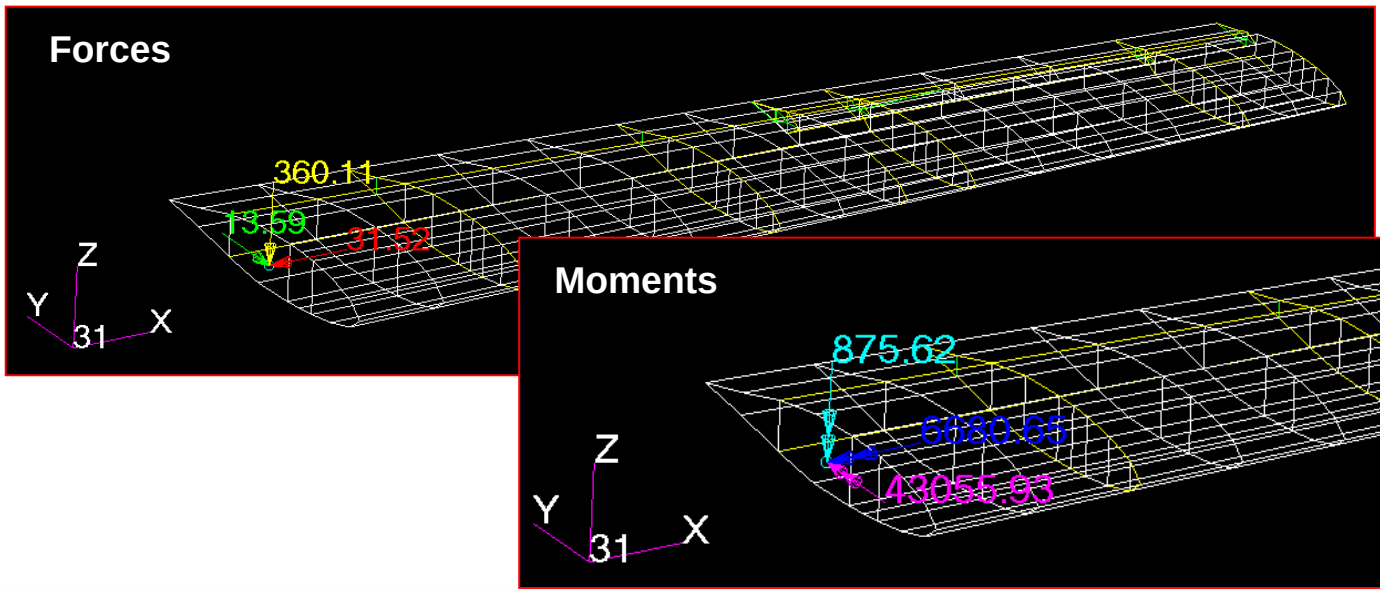
"Freebody Loads" for Result Case [SC2:102], Subcase [A1:Static Subcase], using method [Freebody Loads].
Values shown in the Analysis Coordinate Frames. Summation point shown in Rectangular Coordinate System [0].
Summation Point (0.00, 0.00, 0.00) specified as [[0.0 0.1]

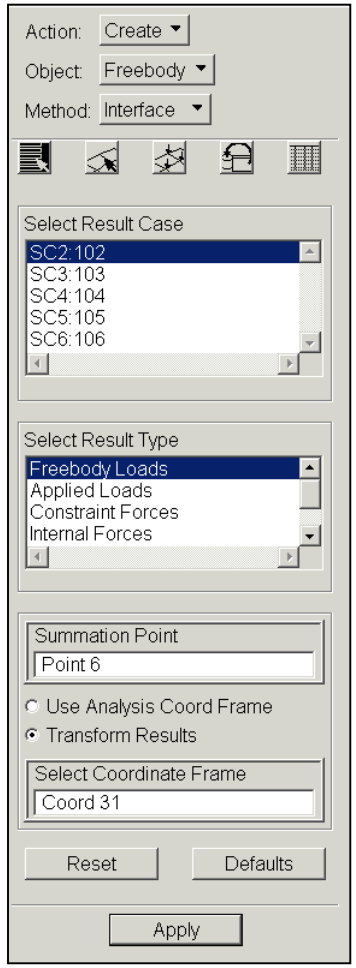
Report Close

Huh? How does 0.01 and 0.05 add up to 307.38?

Freebody Interface Plot

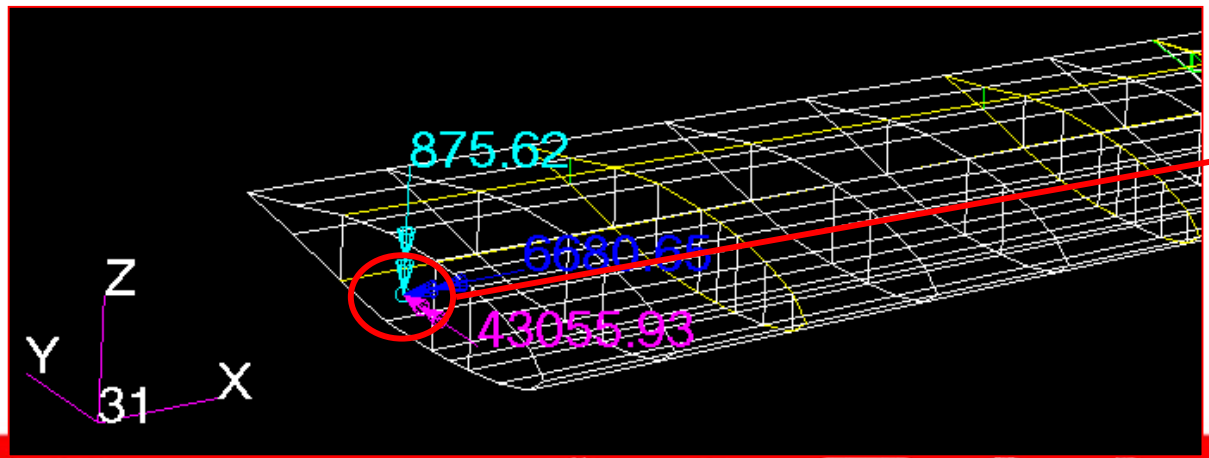
- **The interface method is designed to calculate/display net loads at structure interfaces**
- **A common use is to calculate net forces/moments at various “stations” along a wing or fuselage**

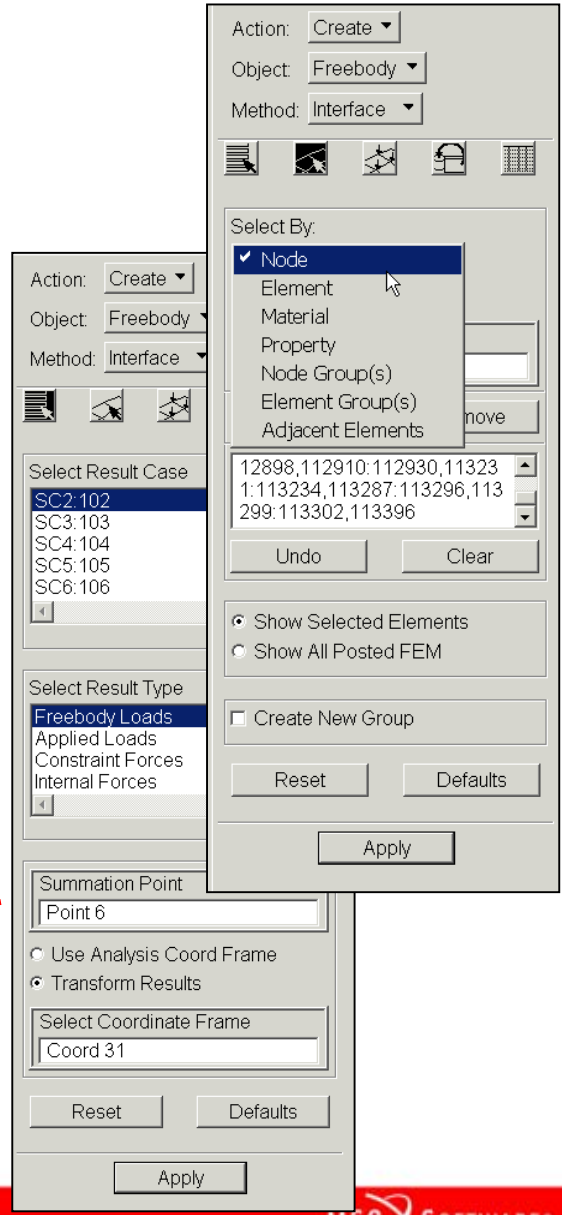




Freebody Interface Plot

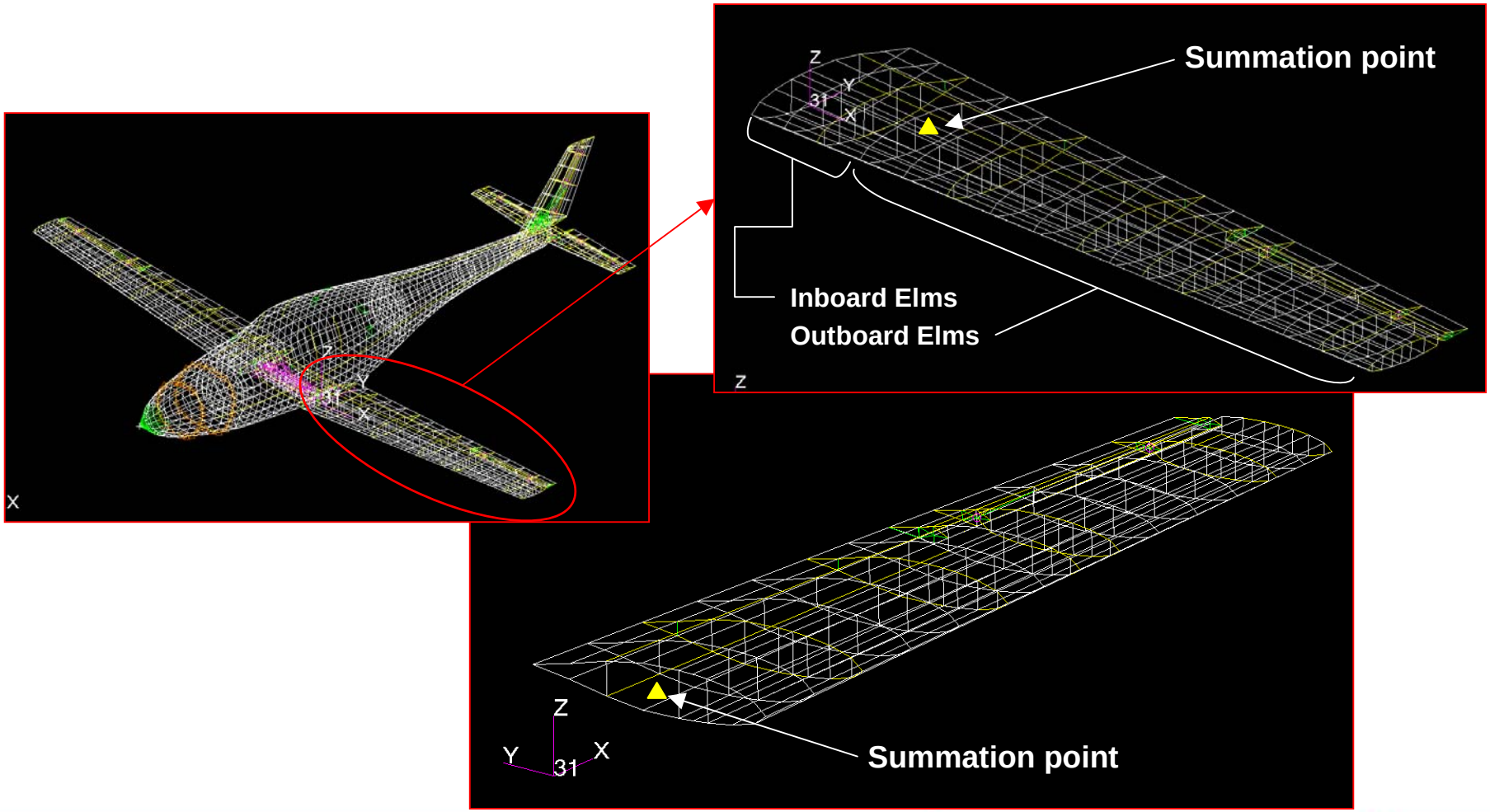
- **Interface method differs from the Loads method in that both *elements and nodes* must be selected**
  - **Results belonging to nodes not associated to the target elements will be ignored**
- **A single net force/moment is calculated at the summation point in the reference system**
  - **The summation point can be any node, point, or location in space.**
    - **Can use Utilities/General/Section Tool to determine neutral axis**





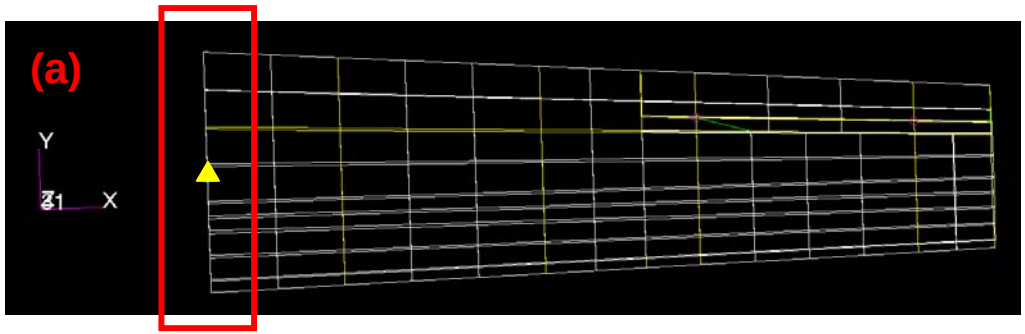
Interface Plot: Example

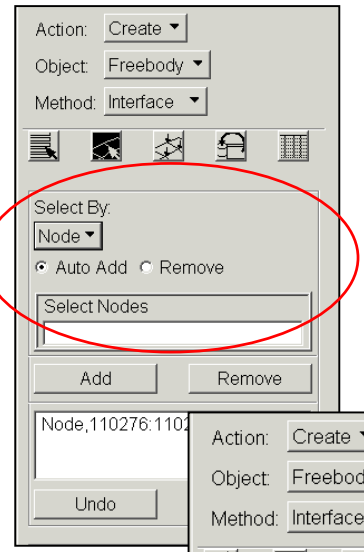
- **Determine net reaction at a point along a wing**

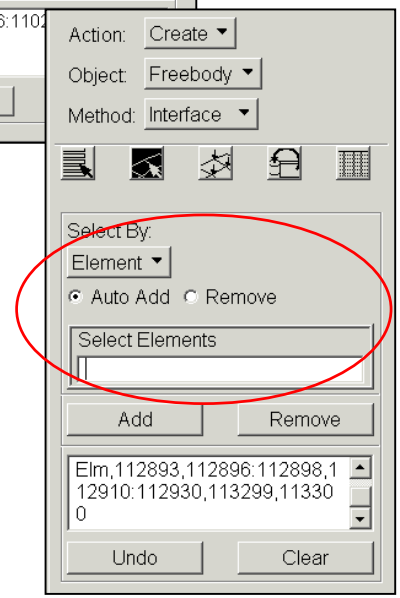


Interface Plot: Example

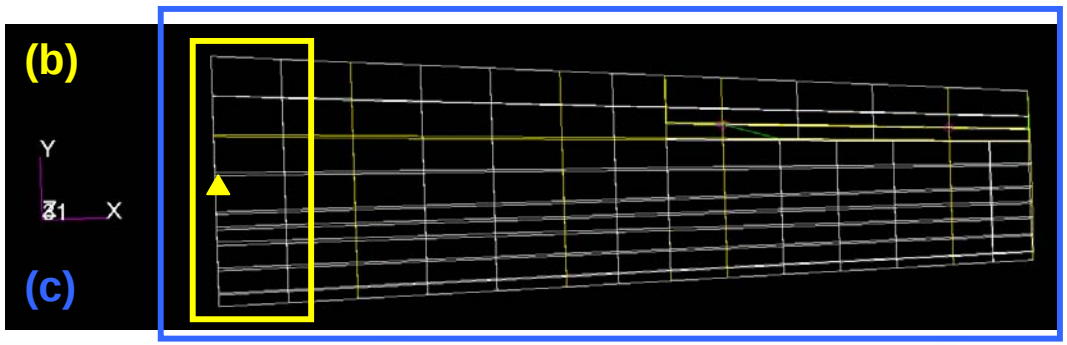
- **Result Type: Freebody Loads**
  - **Recall that: Freebody Loads = $-\Sigma(F_{elm})$**
  - **Also recall that: F_{elm} is stored as vector data for the element at its nodes**
  - **Thus, select nodes along the cut edge (a)**







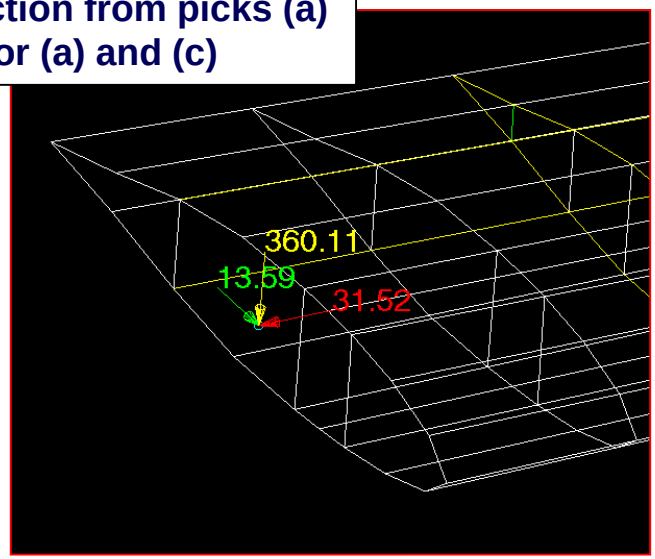
- **Select elements that join the cut edge nodes (b) or select all elements of the outboard portion of the wing (c)**



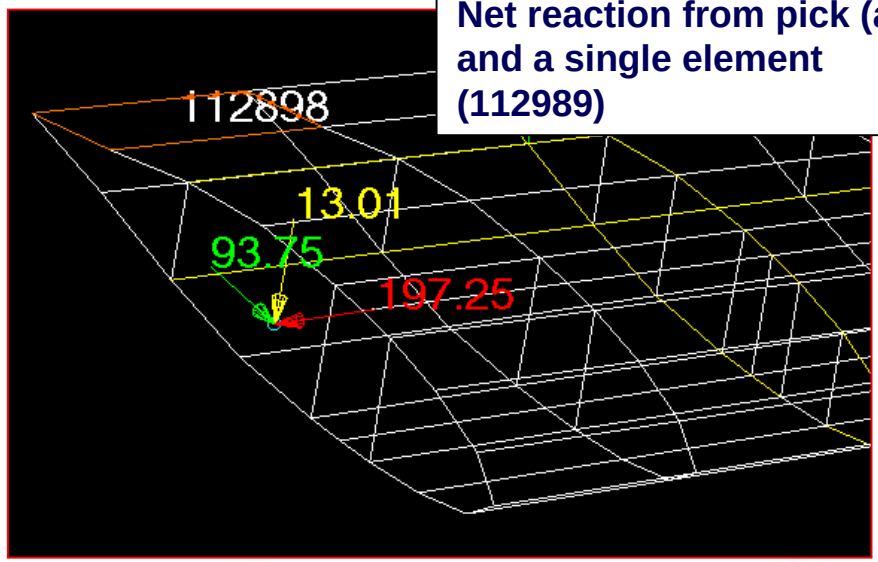
Interface Plot: Example

- **Why does the element selection not matter?**
  - **Actually it does!**
  - ***Freebody Load* results is extracted at element nodal locations from the element list**
  - **Thus, in this case, the node list along the cut boundary means that only elements that connect to these nodes will contribute**

Net reaction from picks (a) and (b) or (a) and (c)

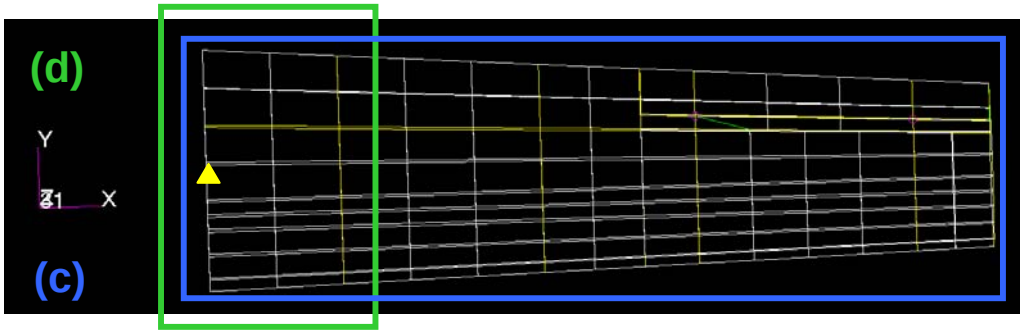


Net reaction from pick (a) and a single element (112989)

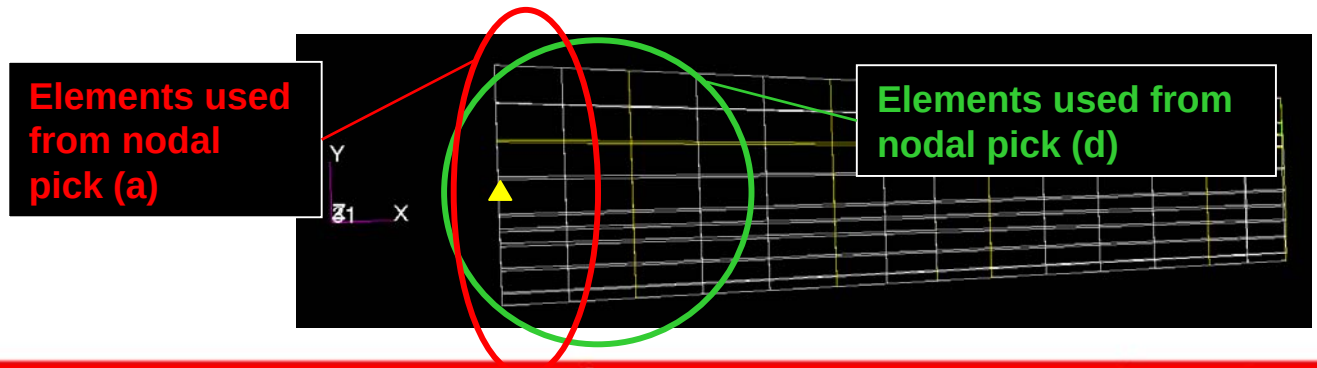


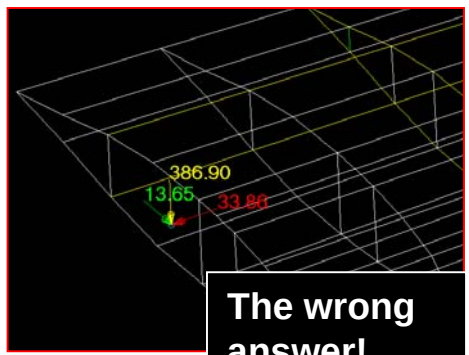
Interface Plot: Example

- **What if I pick more (or less) nodes?**
  - **You will get a different (and most likely wrong) answer!**
  - **Select 3 rows of nodes instead of the single row along the cut edge (d)**



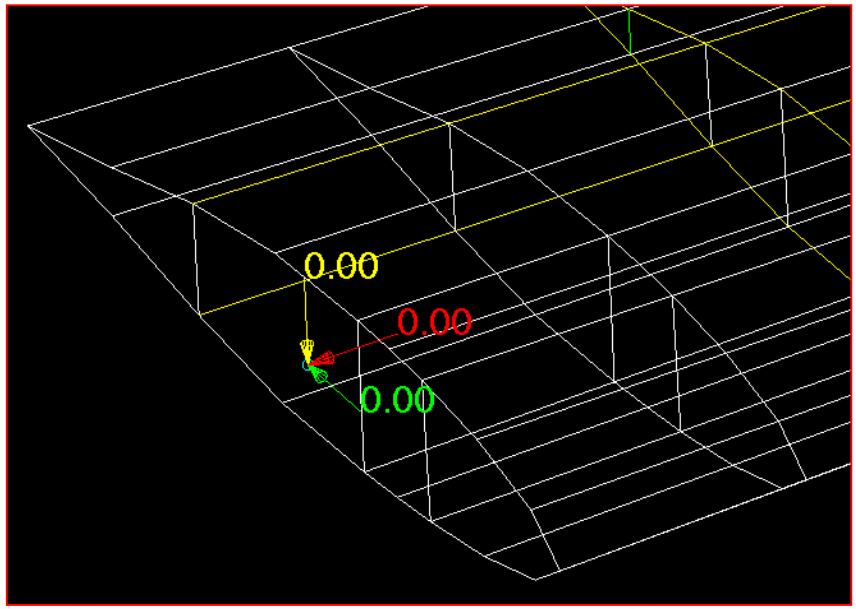
  - **Freebody Load results are pulled for all selected elements connecting to the selected nodes. Thus, selecting the nodes from (d) results in data from 3 rows of elements instead of just the 1st row of elements when picking only the nodes along the cut-edge (a)**





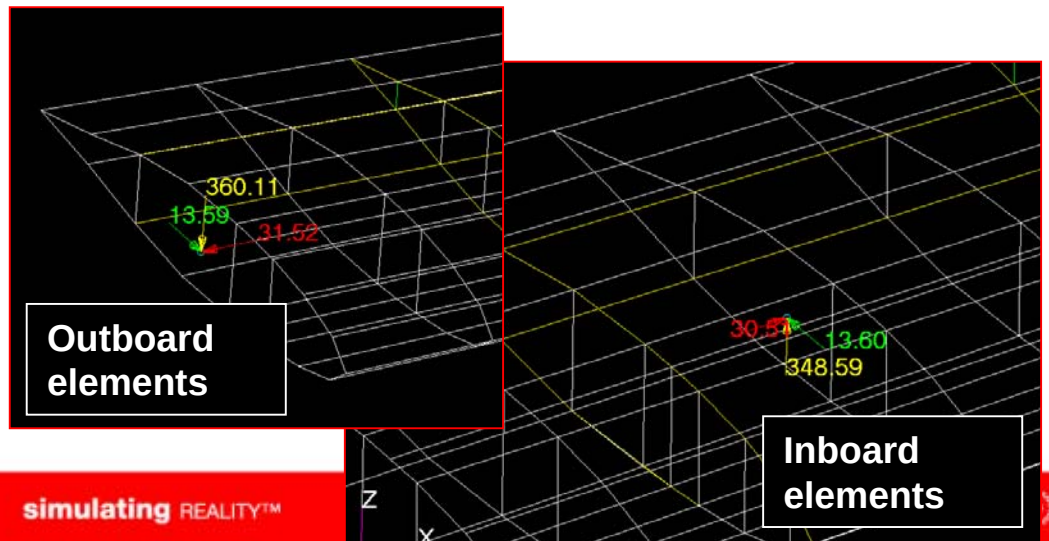
Interface Plot: Example

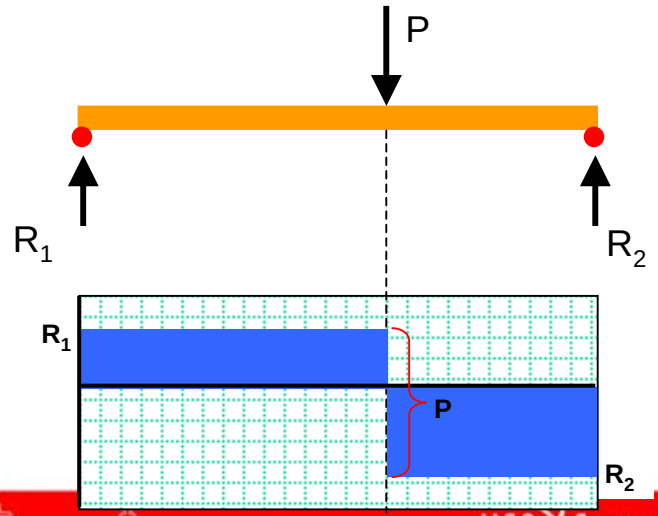
- **The logical (or illogical) extension would be to select all outboard nodes and elements**
- **This amounts to summing all the internal element force contributions at all the nodes**
- **The result simply proves that the wing is indeed in equilibrium with itself, but is useless otherwise**



Interface Plot: Example

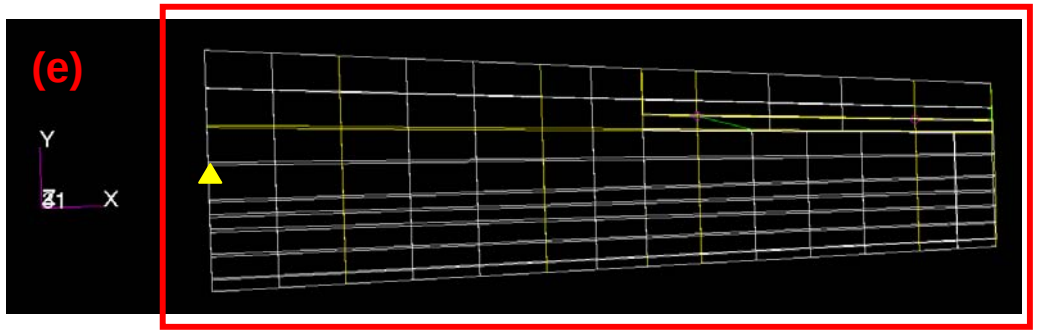
- **What if I select inboard elements instead of outboard elements?**
  - **If no loading or constraint forces or MPC forces are applied to the selected nodes, the results will only differ in sign**
  - **If loading is present at the selected nodes, the results will differ in sign and by the value of the Applied Loads, Constraint Forces, and MPC Forces at the nodes**
    - **Think shear and moment diagram, i.e., shear at the same location changes depending on whether you are considering yourself to be upstream or downstream from the loaded location**



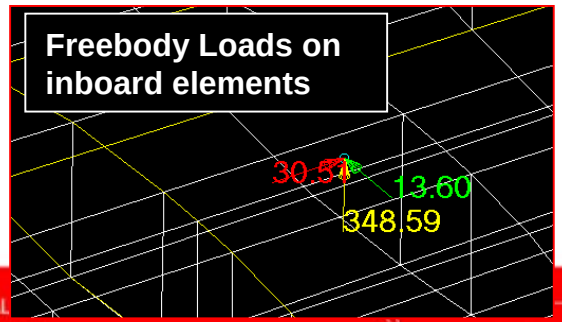


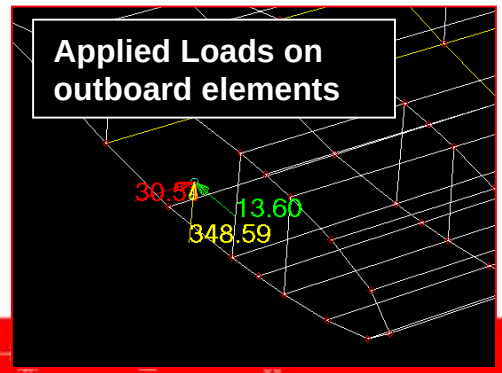
Interface Plot: Example

- **What if I use Applied Loads or Constraint Forces instead of Freebody Loads as the Result Type?**
  - **A key difference is that Freebody Loads are stored as element data and Applied Loads/Constraint Forces are stored as nodal data**
  - **Thus, when using the Interface method must select *all* nodes/elements to be included in the summation (e)**



  - ***In this example*, the outboard Applied Loads should be equal to the inboard Freebody Loads ✓**

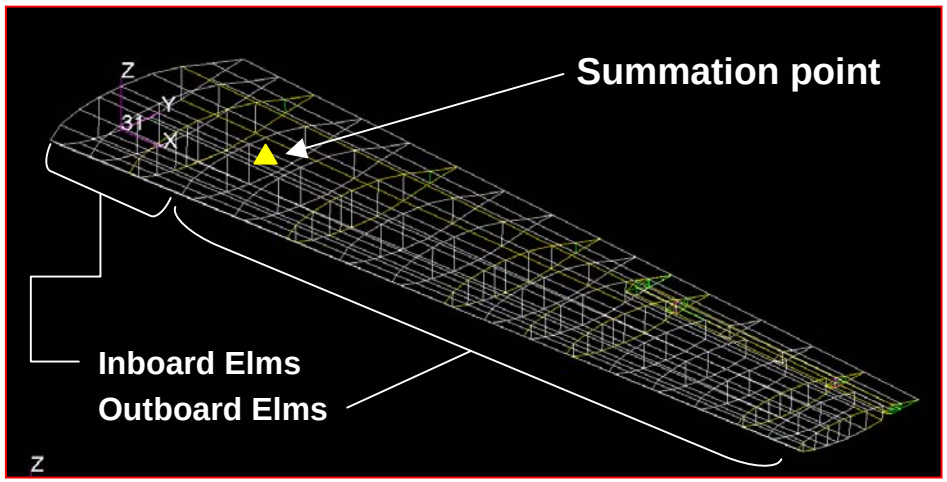




Interface Plot: Example

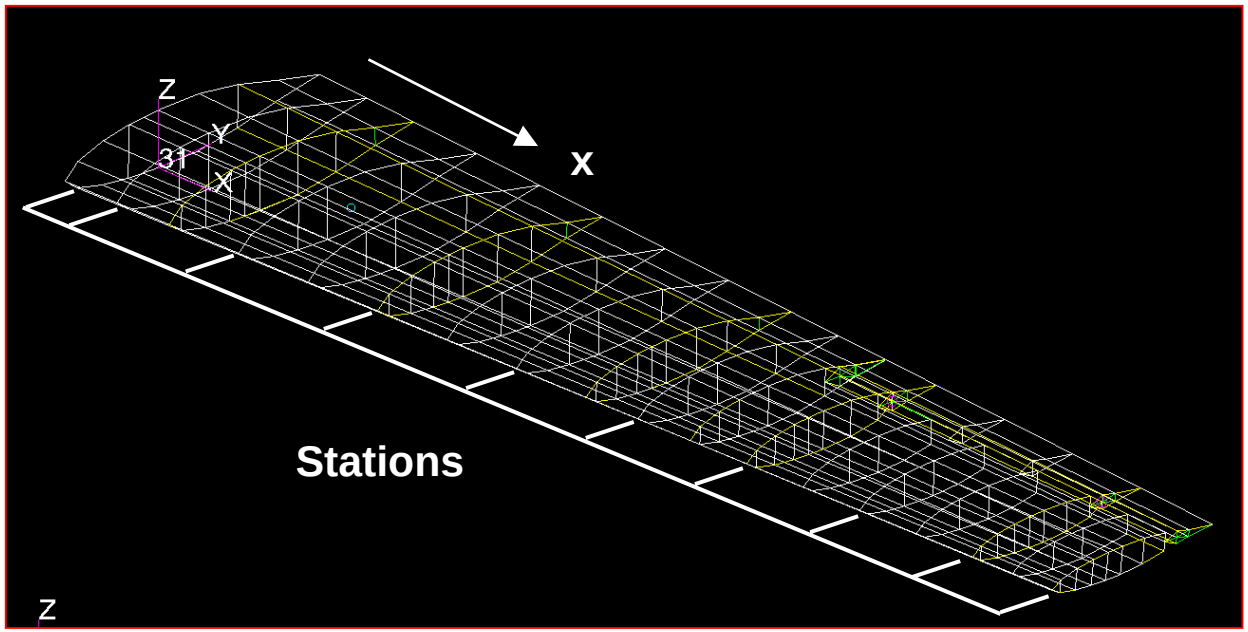
- **What if I use Applied Loads or Constraint Forces instead of Freebody Loads as the Result Type?**
  - ***In general*, the inboard Freebody Loads should be equal to the sum of the outboard Applied Loads, MPC Forces, SPC Forces, and Summation of Forces, or**

$$\text{Freebody Loads}|_{inboard} = -\Sigma(F_{elms})|_{inboard}$$
$$= (F_{Applied} + F_{SPC} + F_{MPC} - F_{total})|_{outboard}$$



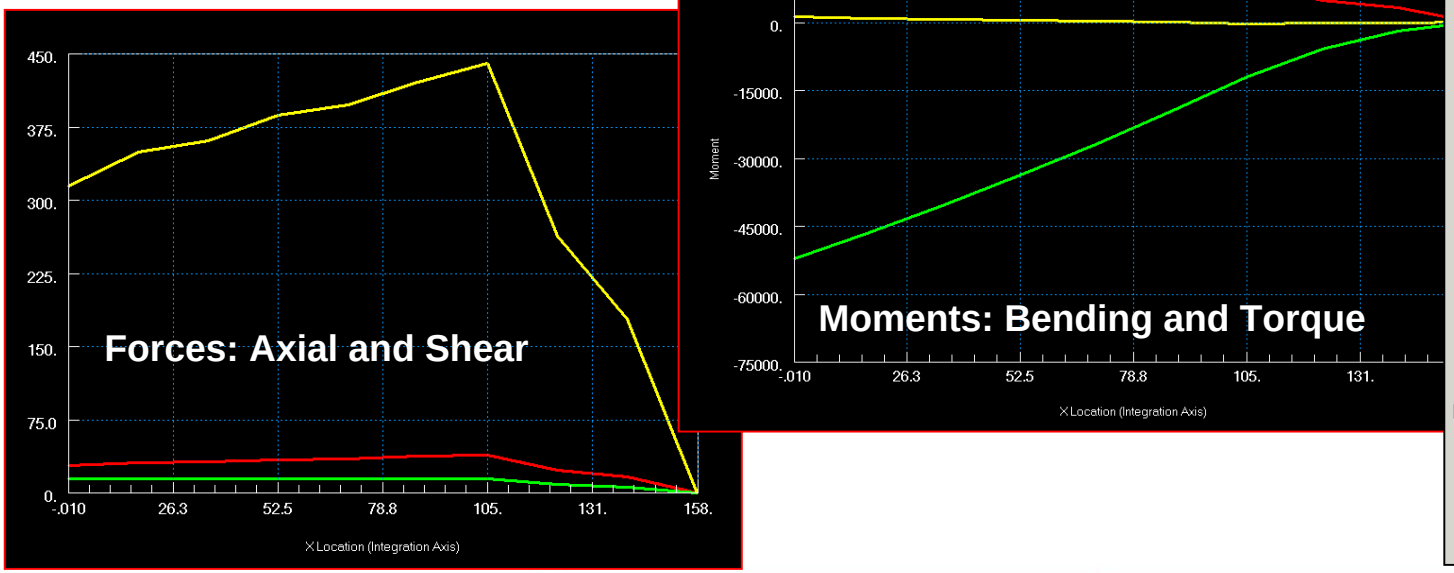
Interface Plot: Example

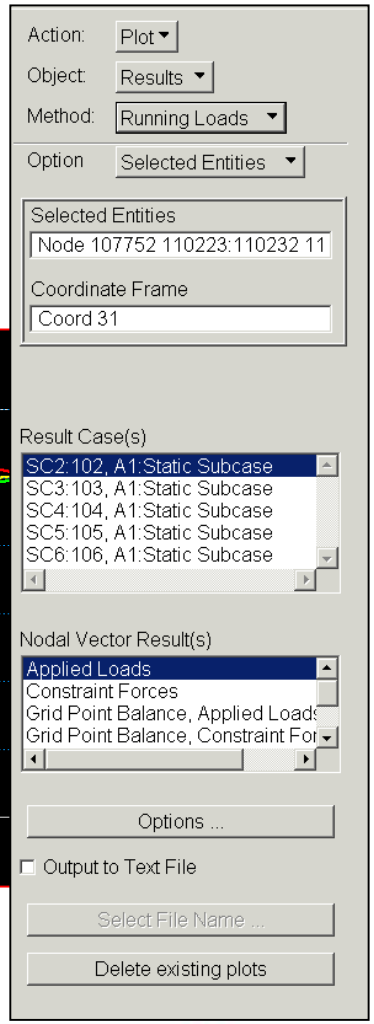
- **Can I automatically generate a shear-moment diagram? Can I do multiple “cuts” or “stations” all a once?**
  - **Not easily via the user interface as a new summation point and new nodes need to be selected for each station**
  - **PCL could be used to automate this**



Interface Plot: Example

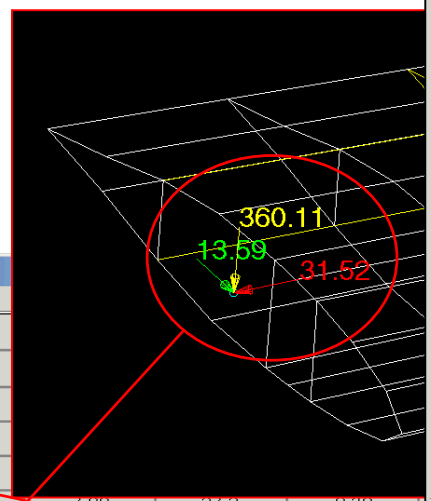
- **Can I automatically generate a shear-moment diagram? Can I do multiple "cuts" or "stations" all a once?**
  - **Best to use the SBMT or Running Loads plot tool that is delivered as part of MSC.SuperModel**
    - **Can plot shear, bending moment, torque based on LBC set data or applied load results data (OLOAD)**

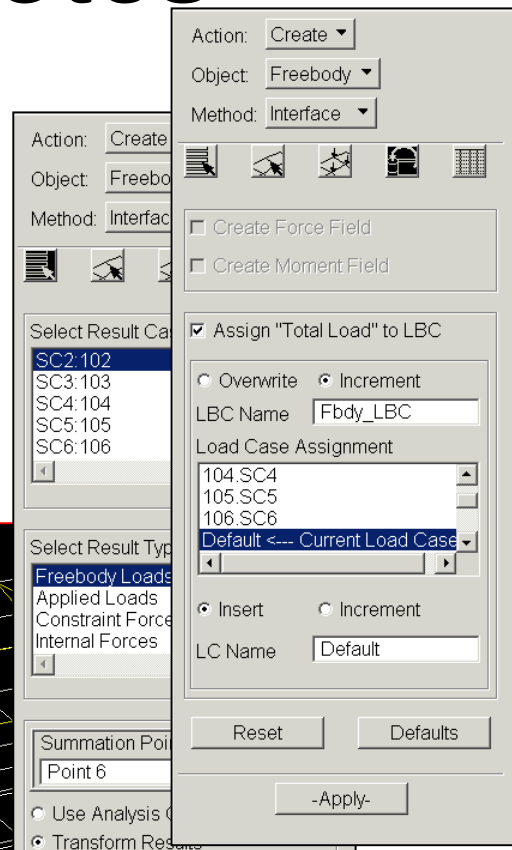




Interface Plot: Other Notes

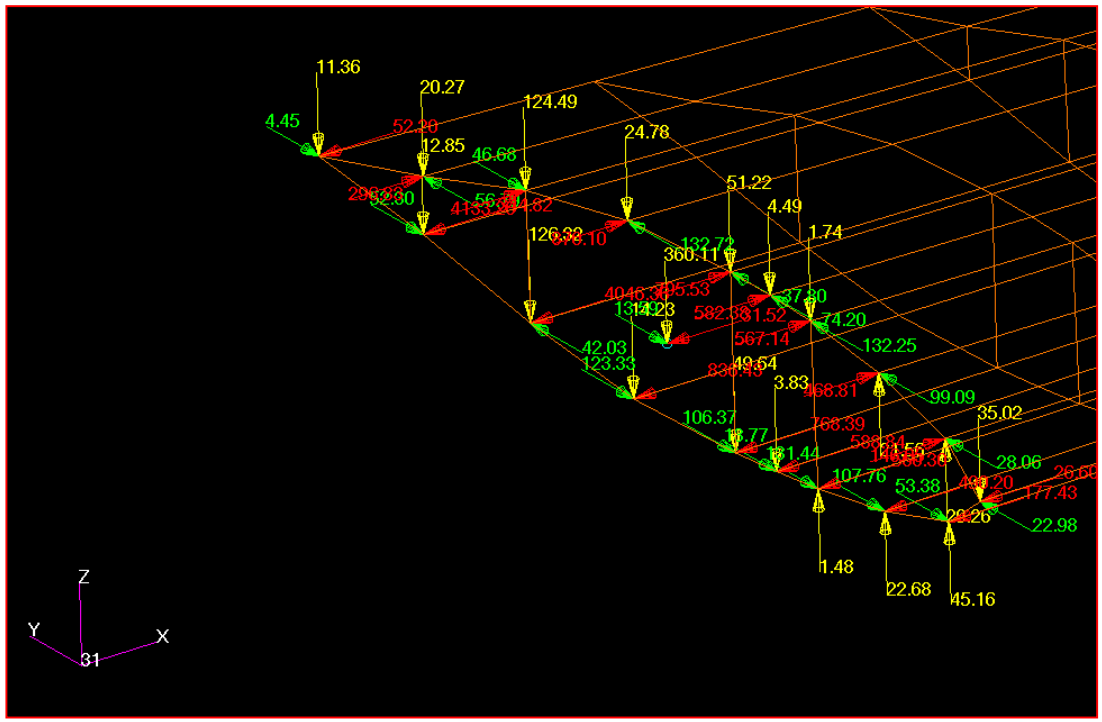
- ***Save Data*** **does not create a field as before since the resultant is at a single point.**
- **The *Freebody Spreadsheet* includes data for all selected nodes. The Totals row should equal the net resultant displayed on the model**

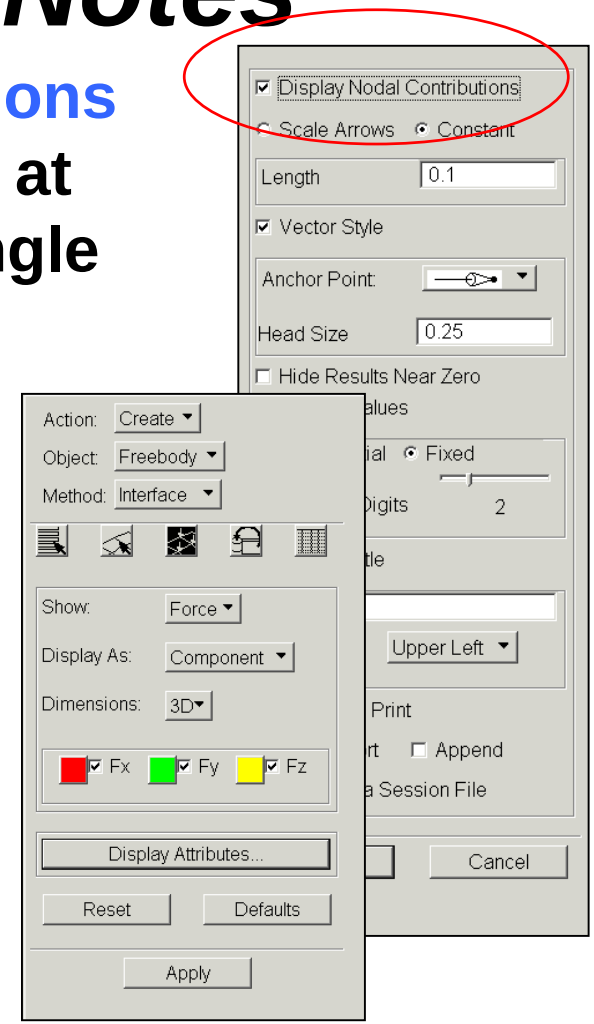




Interface Plot:Other Notes

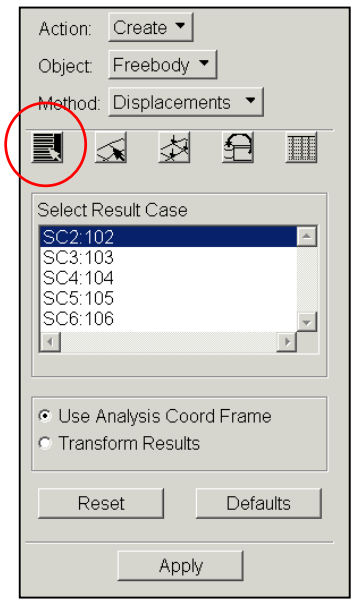
- **Selecting the Display Nodal Contributions toggle causes vectors to be displayed at all selected nodes instead of just a single point**

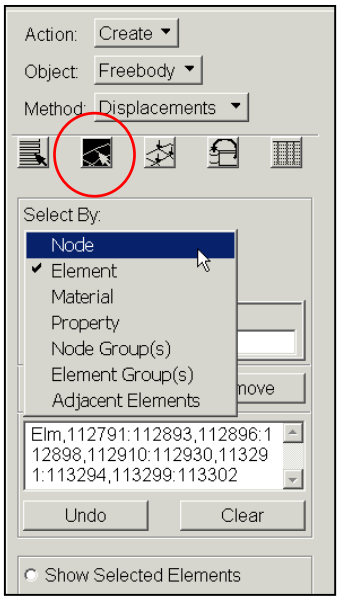


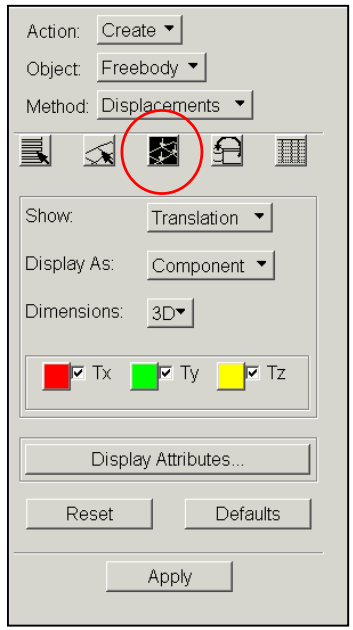


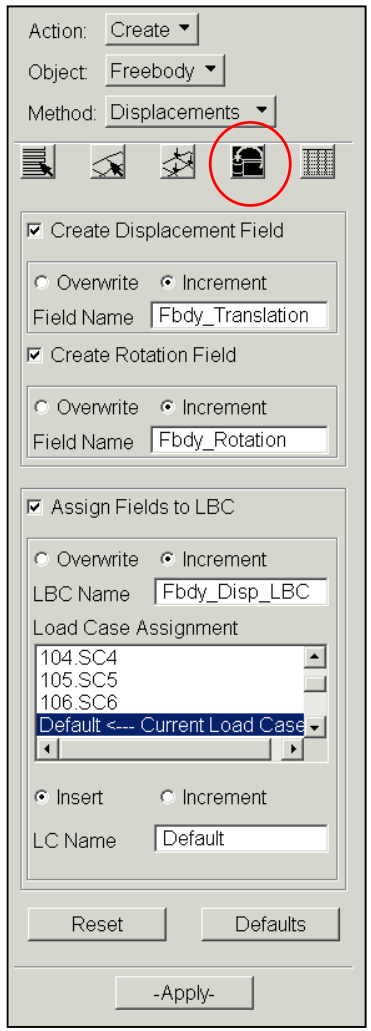
Freebody Displacement Plot

- Displays displacements/rotations instead of forces and/or moments
- Primarily used to create LBC displacement data for global/local modeling
- Select elements or nodes



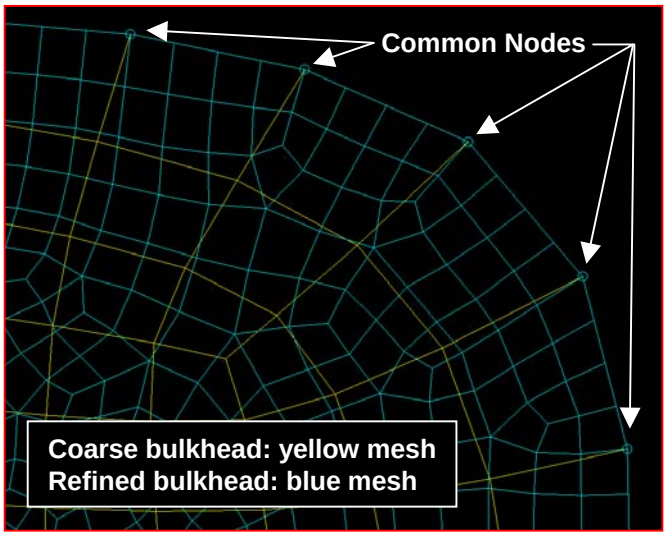


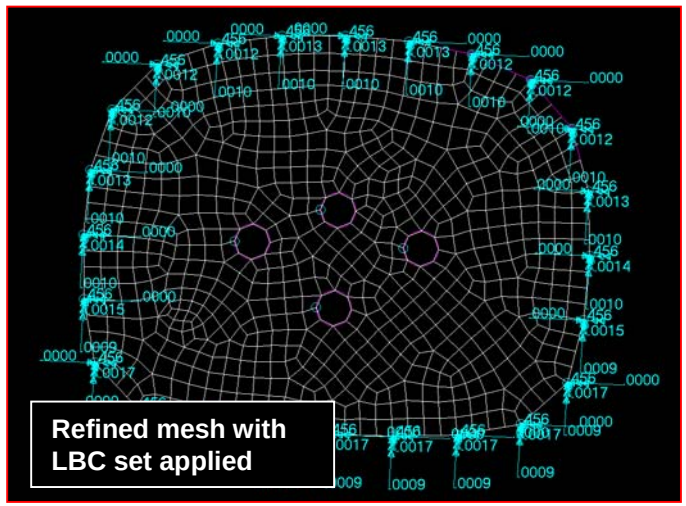


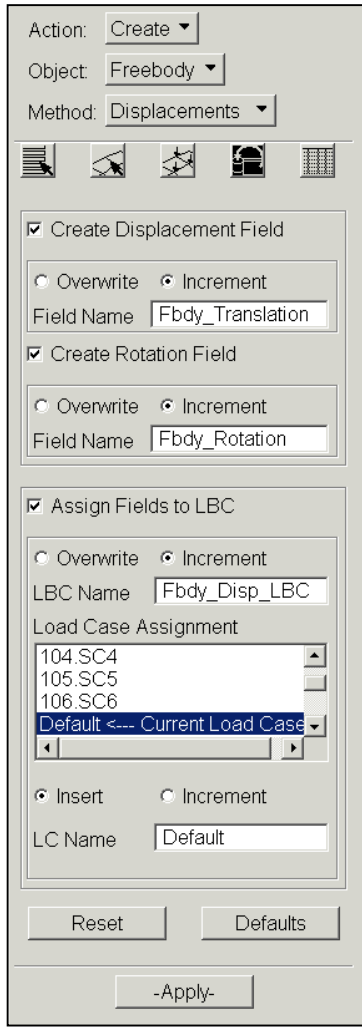


Freebody Displacement Plot

- Global/local modeling
  - Saves displacement data for both translations and rotations in separate discrete FEM fields (DFEM)
  - Optionally these fields may be applied to an LBC set
- A DFEM field is a table of node IDs vs values
  - MSC.Patran cannot interpolate within a DFEM field, so any extra nodes do not get a displacement applied
  - Hence the local model and global model must share common node IDs at displacement application points

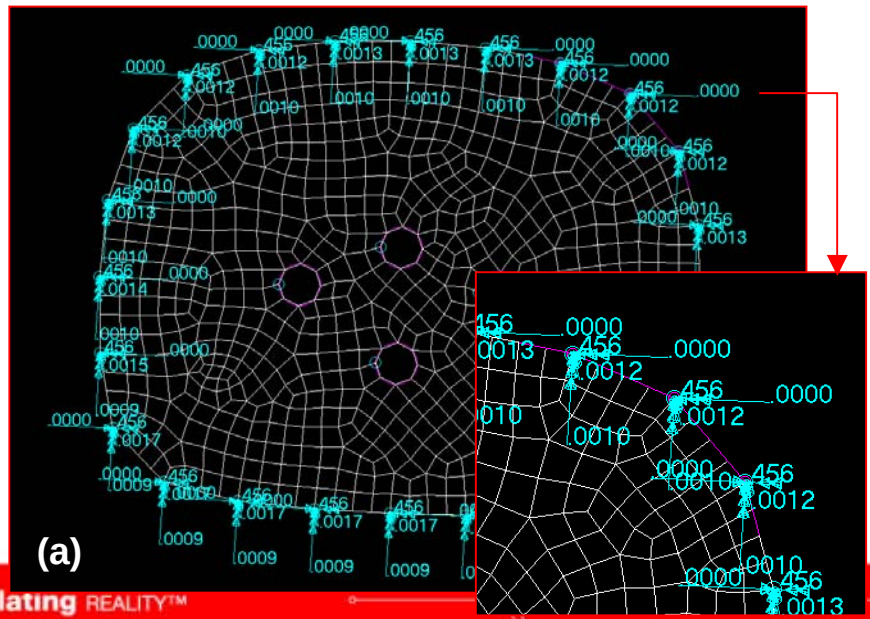


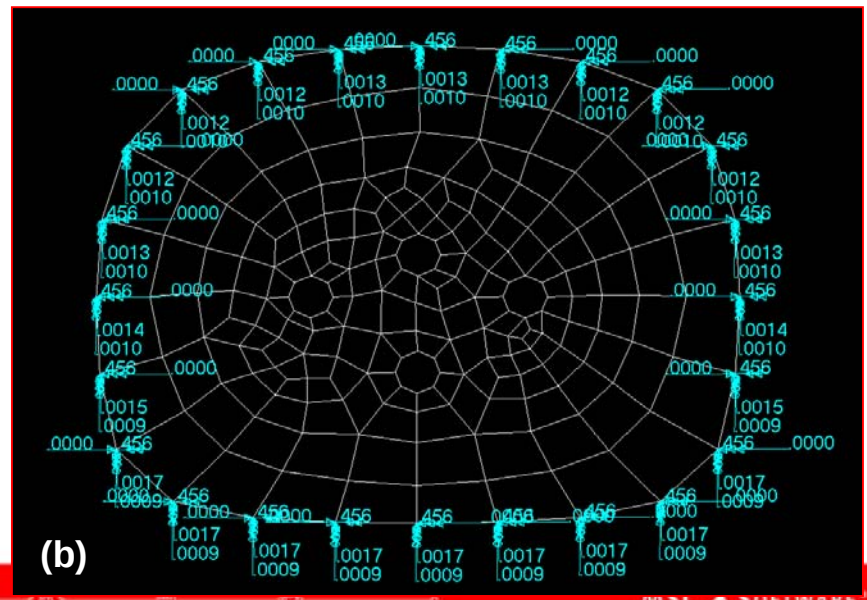




Freebody Displacement Plot

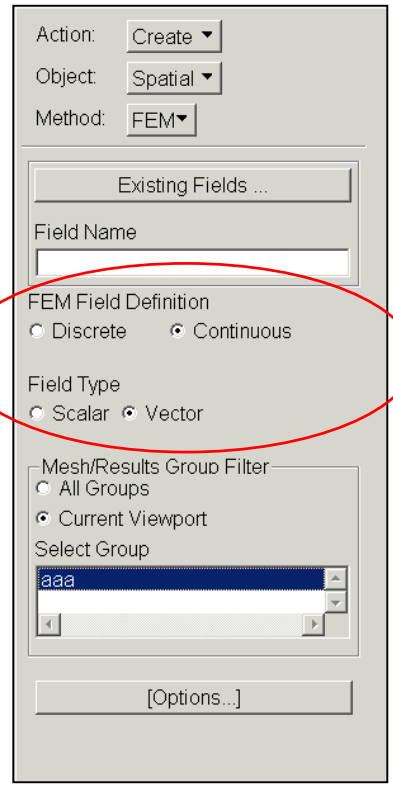
- **What to do about the “extra” boundary nodes in the refined model?**
  - ***Use company best practices!***
  - **If the bounary is far removed from the area of interest use St Venant’s Principle by either**
    - **a) Not assigning any displacement to the extra boundary nodes**
    - **b) Add layer(s) of elements**
  - **Use RBE3’s to distribute load at boundary (cannot apply SPC to RBE3 reference grid dof unless you use UM dofs)**





Freebody Displacement Plot

- **The alternative to using the Freebody Tool to create displacement DFEM fields is to**
  - **Create a vector plot of the displacement resultant in results**
  - **Create a continuous FEM field from this plot**
    - **Continuous FEM fields can be applied to a dissimilar mesh (i.e., no common node IDs)**
    - **MSC.Patran can interpolate within a continuous FEM field**
  - **Repeat for rotations**



Miscellaneous Stuff

- **MSC.Patran freebody tool does not support**
  - **PARAM, NOELOF, +1**
    - **If NOELOF > 0, grid point forces are computed along the edges of the 2D elements**
  - **PARAM, NOELOP, +1**
    - **If NOELOP > 0, the sum of the grid point forces are computed parallel to the edges of adjacent elements**
- **Why can't I include MPCs in the selected freebody?**
  - **MSC.Nastran dumps all MPC contributions at a node together as a single entry in the grid point force balance table**

MSC.Patran's Freebody Tool

Questions and Answers