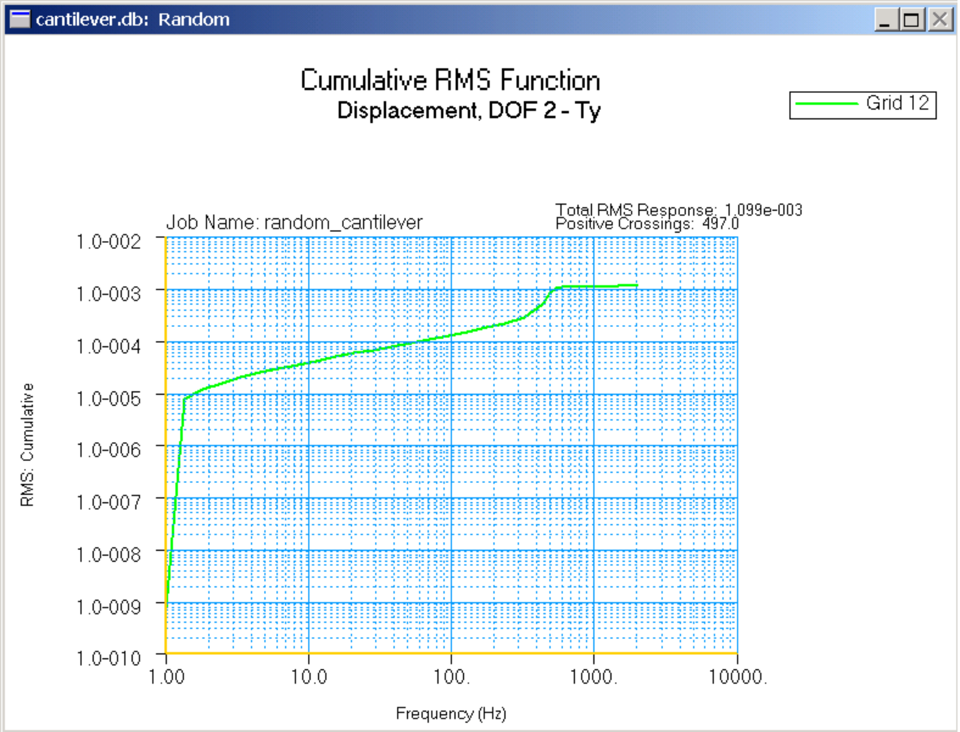
MODAL & RANDOM VIBRATION ANALYSIS OF A SIMPLE CANTILEVER BEAM

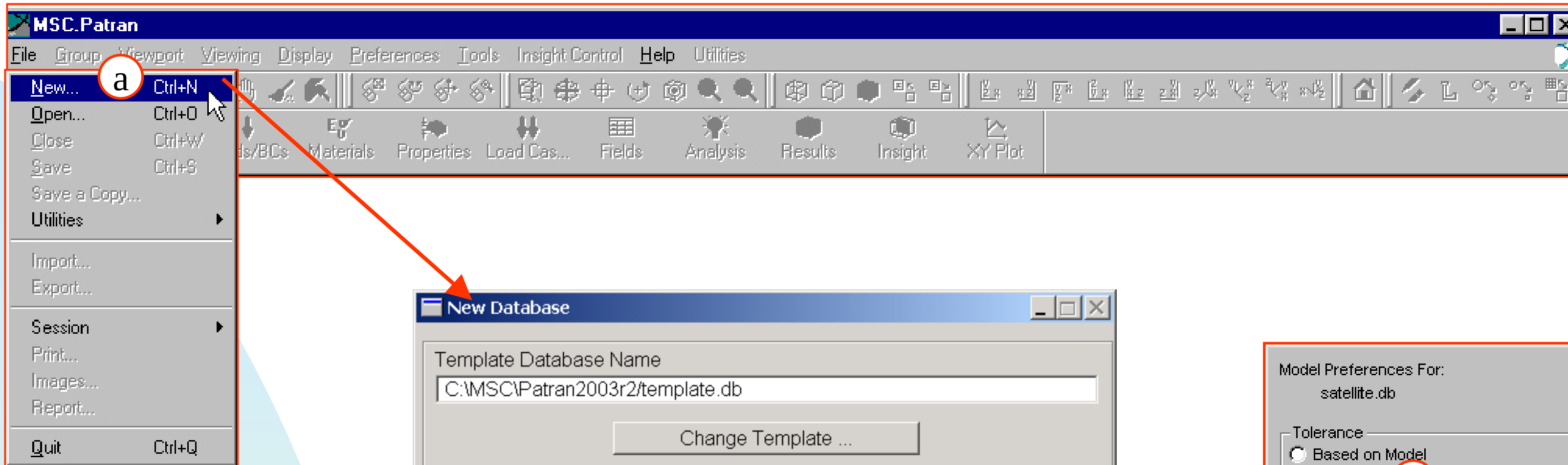


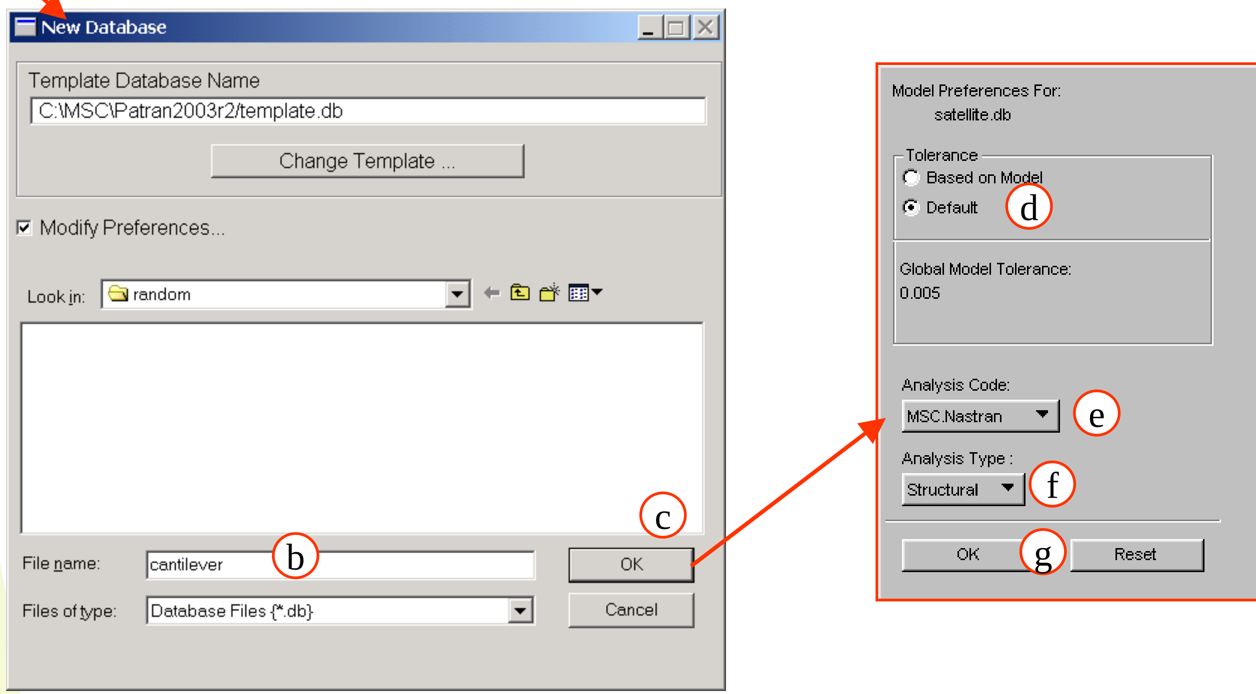
- Problem Description
- The objective of this presentation is to give a clear understanding of solving a Modal & Random Analysis Problem Using MSC.Patran, MSC.Nastran, and MSC.Random

Suggested Exercise Steps

1. Run Modal Analysis
    a. Create the Geometry – Curve that is 10 inches long
    b. Mesh Curve with 1 inch long elements
    c. Create Material Properties For Aluminum (E = $10x10^6$ psi, $\upsilon = 0.33$)
    d. Assign Material And Properties for a 1 in x 1 in rectangular beam to the Curve (or Mesh).
    e. Create Small Mass On Beam
    f. Setup Modal Analysis
    g. Run Modal Analysis
    h. Review Results
2. Run Random Vibration Analysis
    a. Create Damping Field
    b. Create PSD Field
    c. Assign Node 1 to Large Mass
    d. Create Relative Displacement MPC
    e. Run Random Analysis
    f. Review Results

CREATE NEW DATABASE





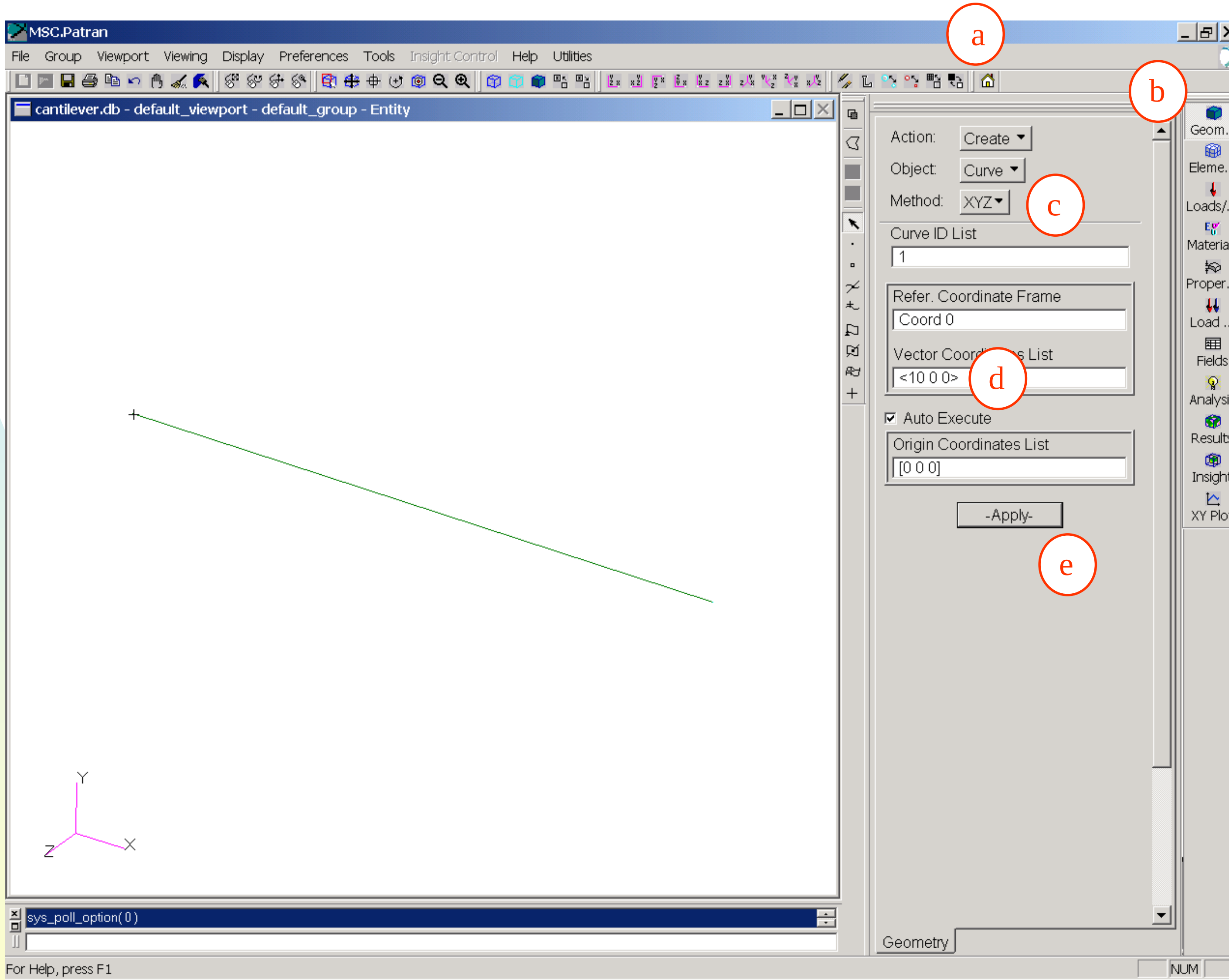
Create a new database called satellite.db.

a. File / New.
b. Enter **cantilever** as the file name.
c. Click **OK**.
d. Choose **Default** Tolerance.
e. Select **MSC.Nastran** as the Analysis Code.
f. Select **Structural** as the Analysis Type.
g. Click **OK**.

Create Geometry

Create Curve

a. Click on the Black and White Reverse Button to change screen background
b. Click on the Geometry
c. Create / Curve / XYZ
d. Change vector to <10, 0, 0>
e. Click on Apply

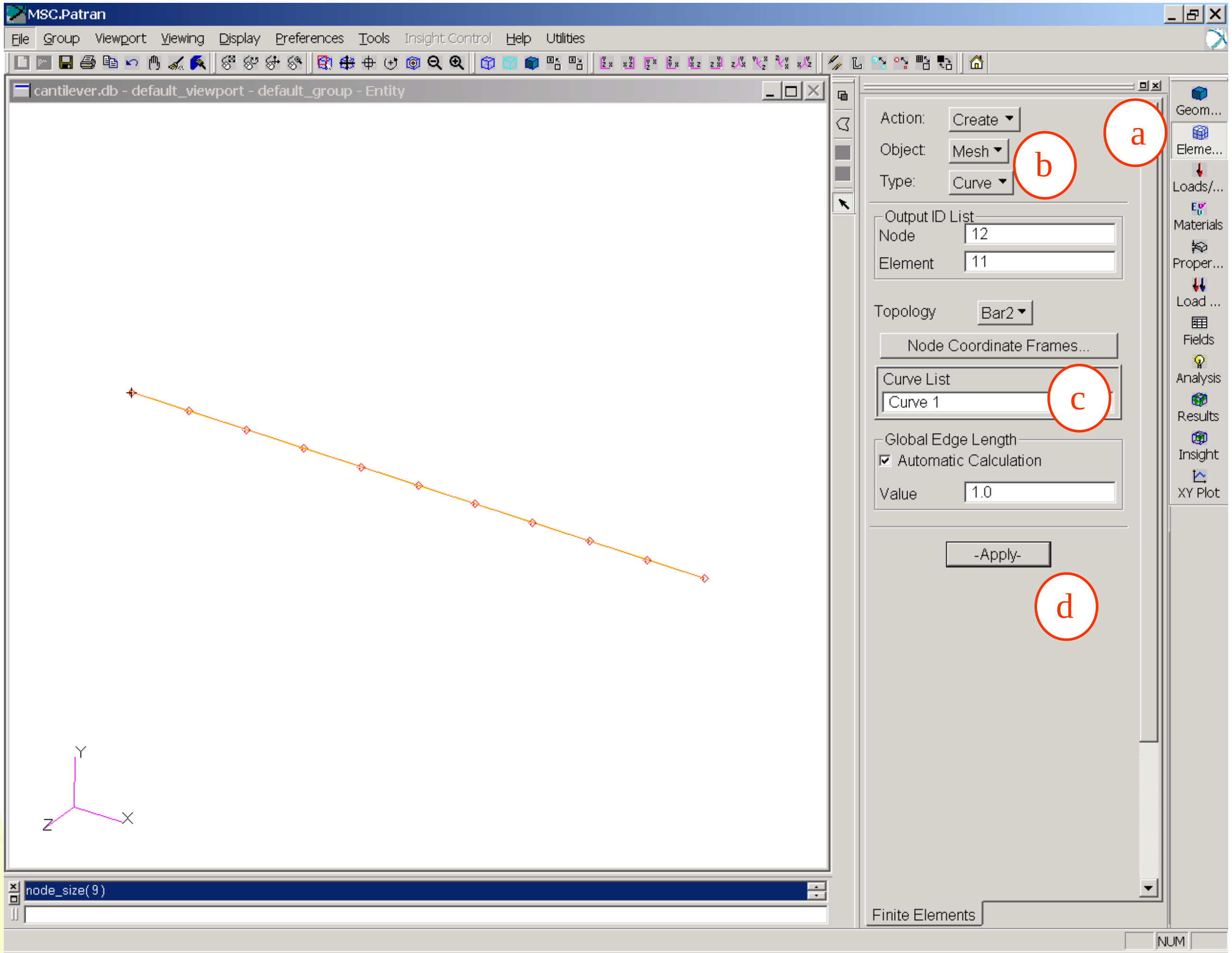


MSC SOFTWARE SIMULATING REALITY

Create the Mesh

Create Mesh on Curve

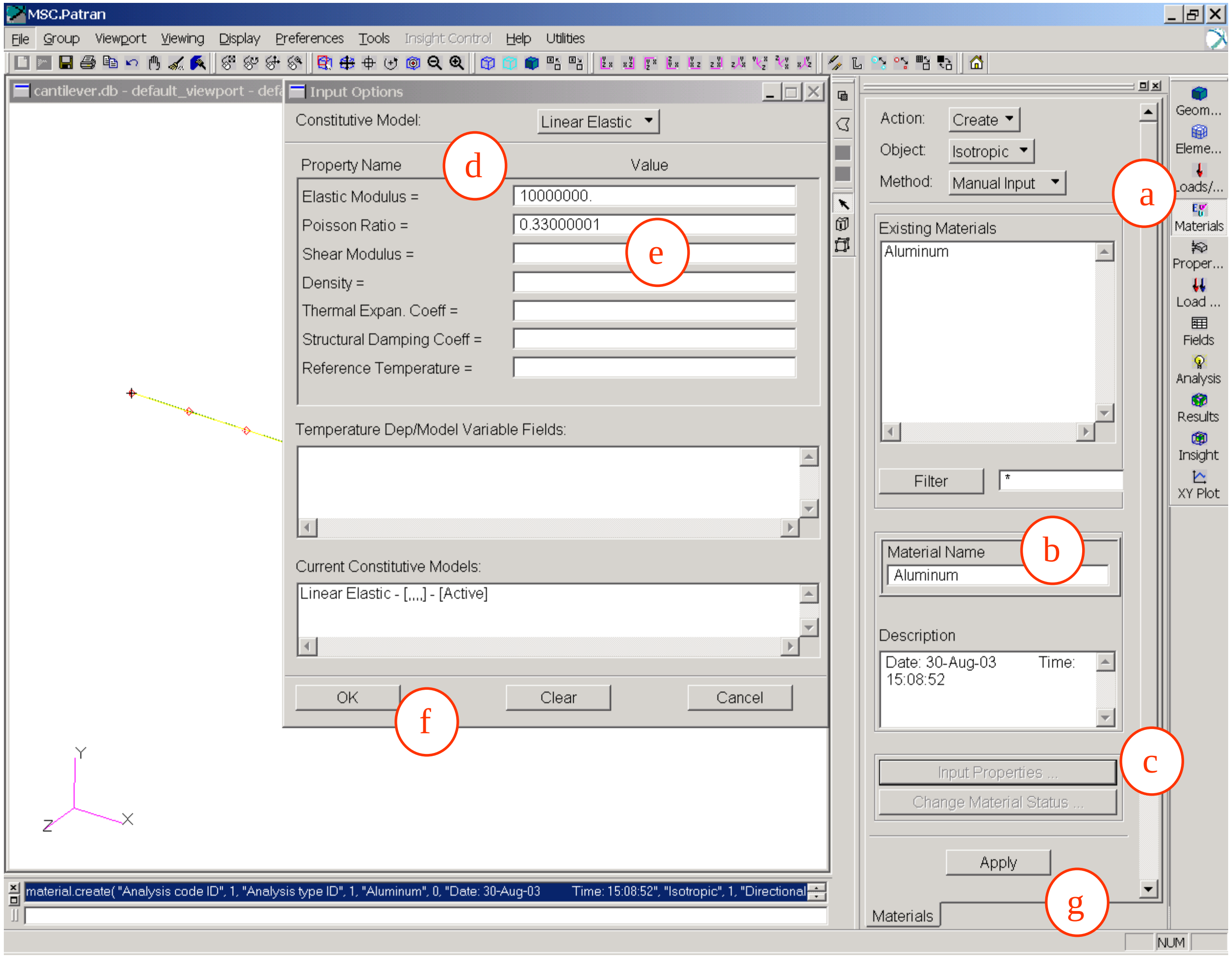
a. Click On Element Form
b. Create / Mesh / Curve
c. Click on Curve 1
d. Apply



Create Material Properties

Create Aluminum Material Property

a. Click on Materials Form
b. Material Name: Aluminum
c. Input Properties…
d. Elastic Modulus: 10e6 psi
e. Poisson's Ratio: 0.33
f. OK
g. Apply



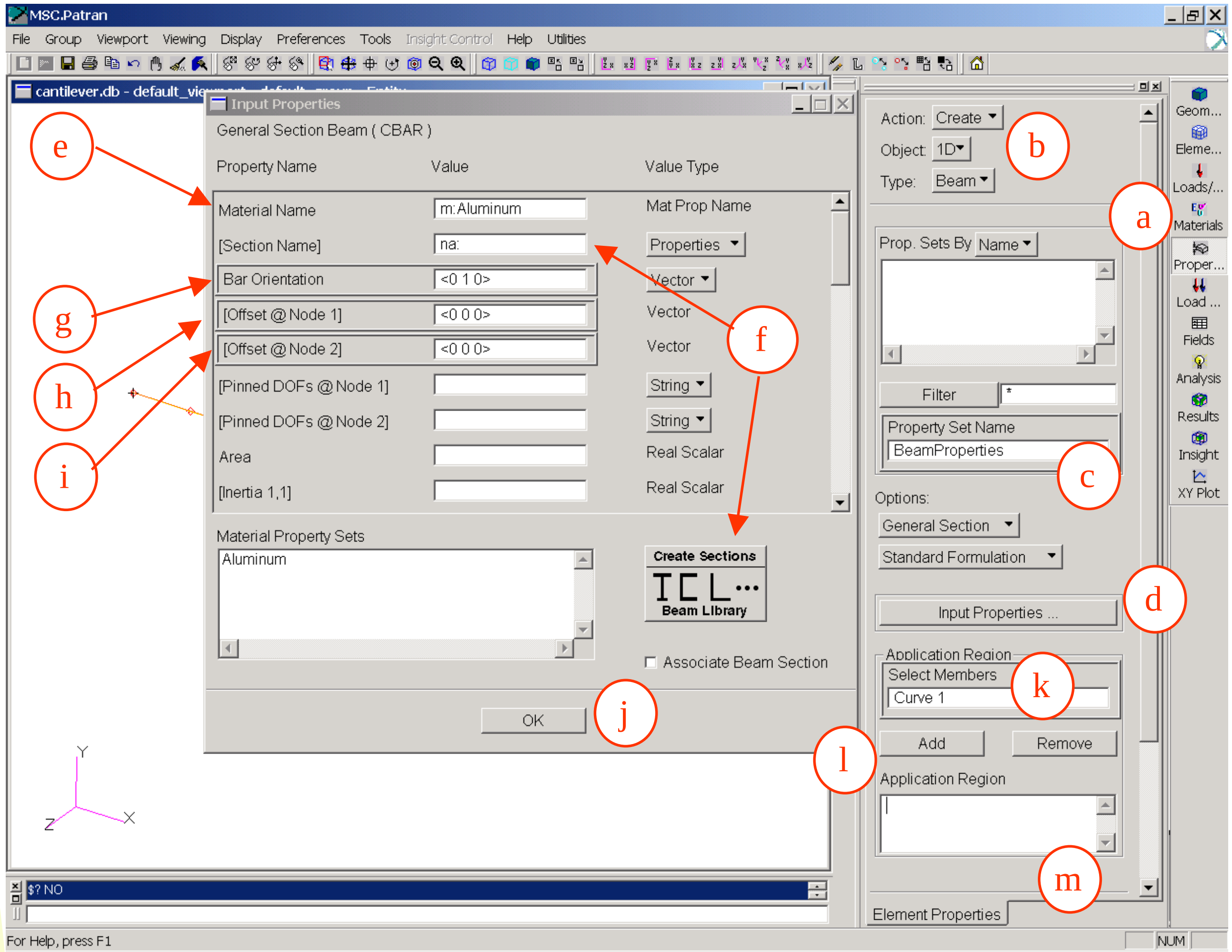
Assign Properties To Curve

Assign Properties to Curve

a. Click on the Properties Form
b. Create / 1D / Beam
c. Property Set Name: BeamProperties
d. Input Properties…

Switch to Input Properties Form

e. Material Name: Aluminum (Select from Material Property Set box)
f. For Section, Click on Create Sections Button, then see the Next Slide (4f.)
g. Bar Orientation: <0 1 0>
h. Offset @ Node 1: <0 0 0>
i. Offset @ Node 2: <0 0 0>
j. OK
k. Select Members: Click on Curve 1
l. Add (To add Curve 1 to Application Region)
m. Apply (Need to scroll form to see Apply button).



Create Beam Section

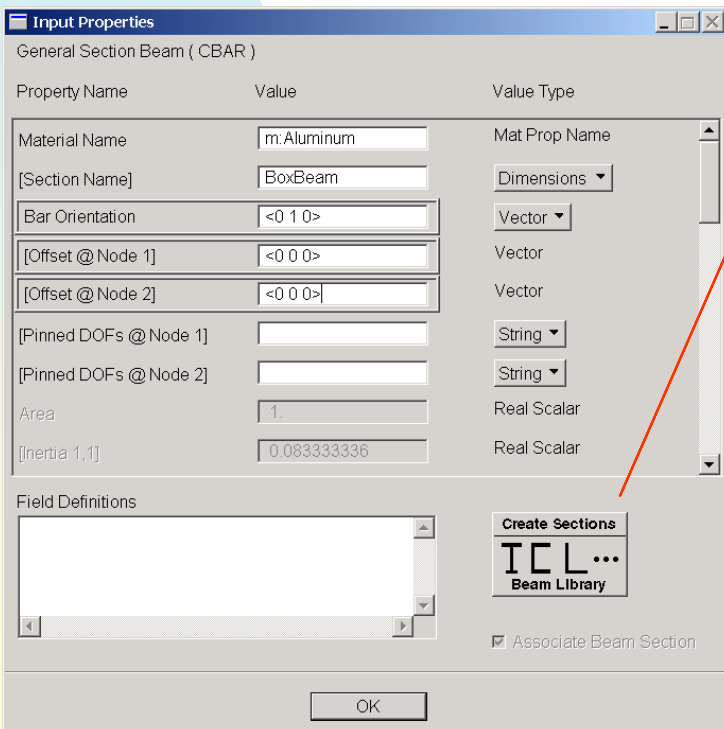
Step 4f (Continued from Previous Page)

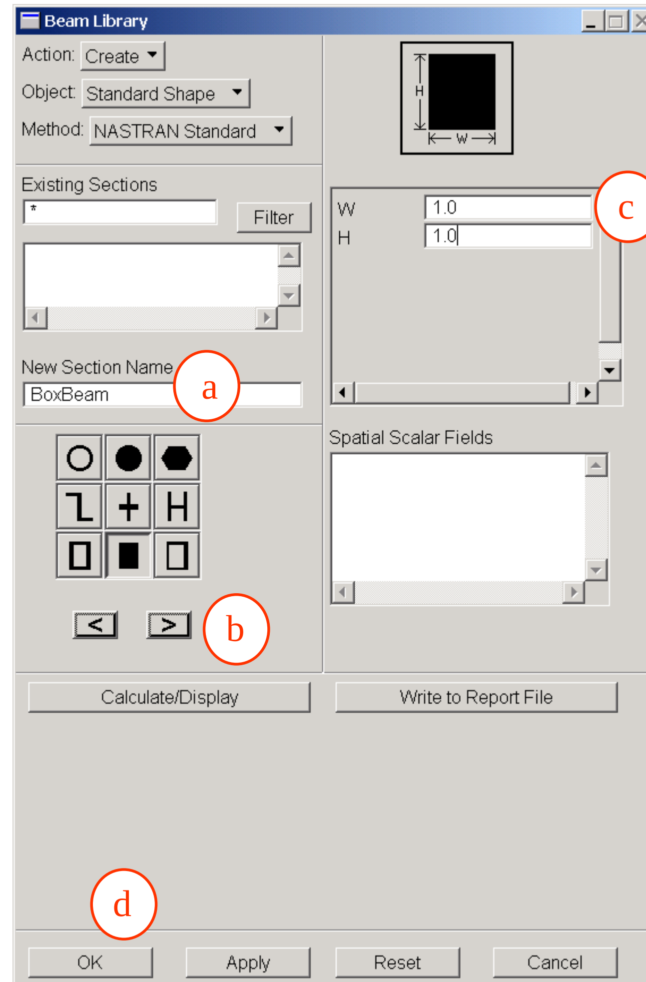
a. New Section Name: BoxBeam

b. Click Arrows Until you can select the Solid Rectangle Cross-Section

c. W: 1.0, H 1.0

d. OK - Return to previous slide and form.



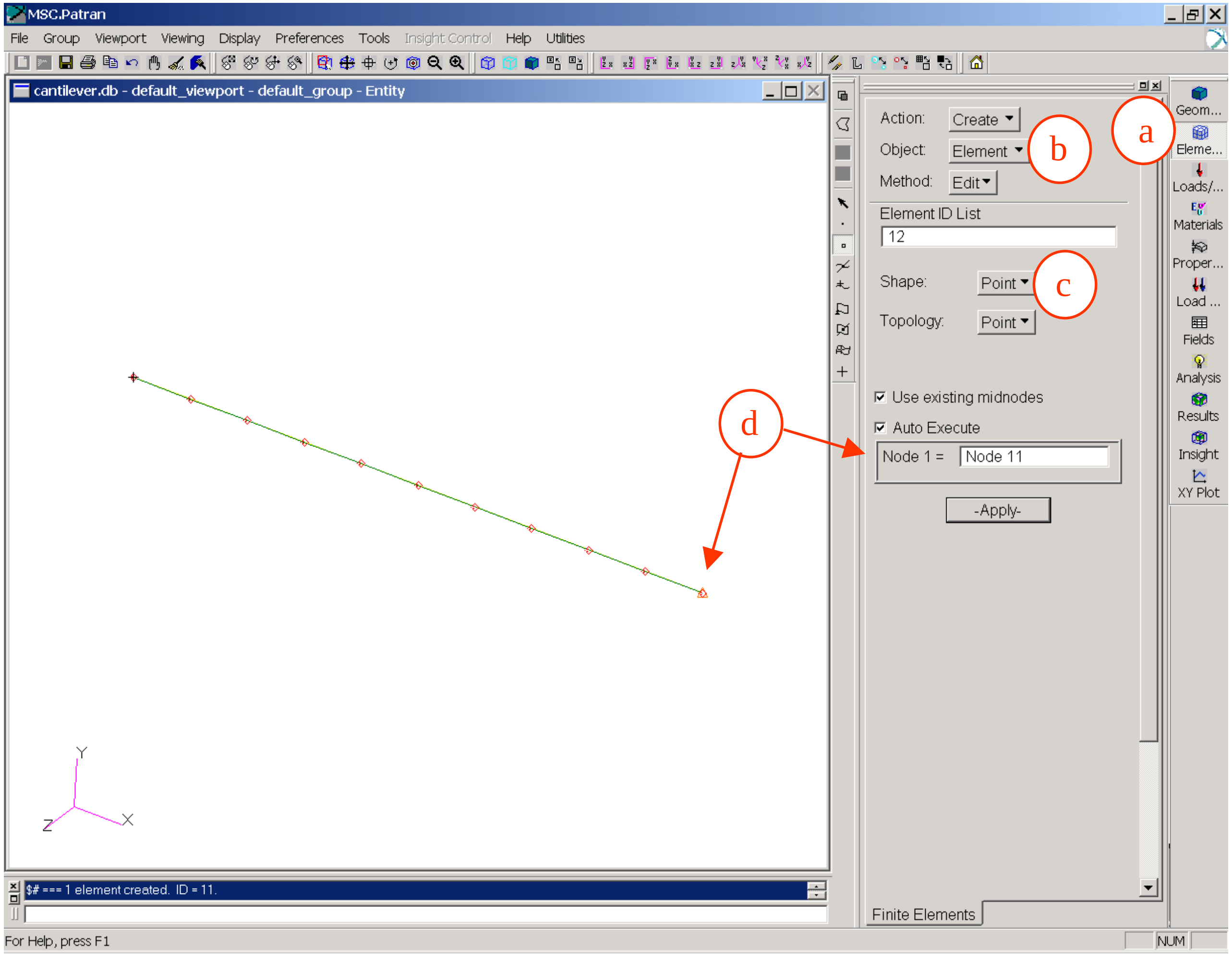


Create Small Mass Element

Create Small Mass Element

a. Click On Element Form
b. Create / Element / Edit
c. Shape: Point
d. Click on Node 11

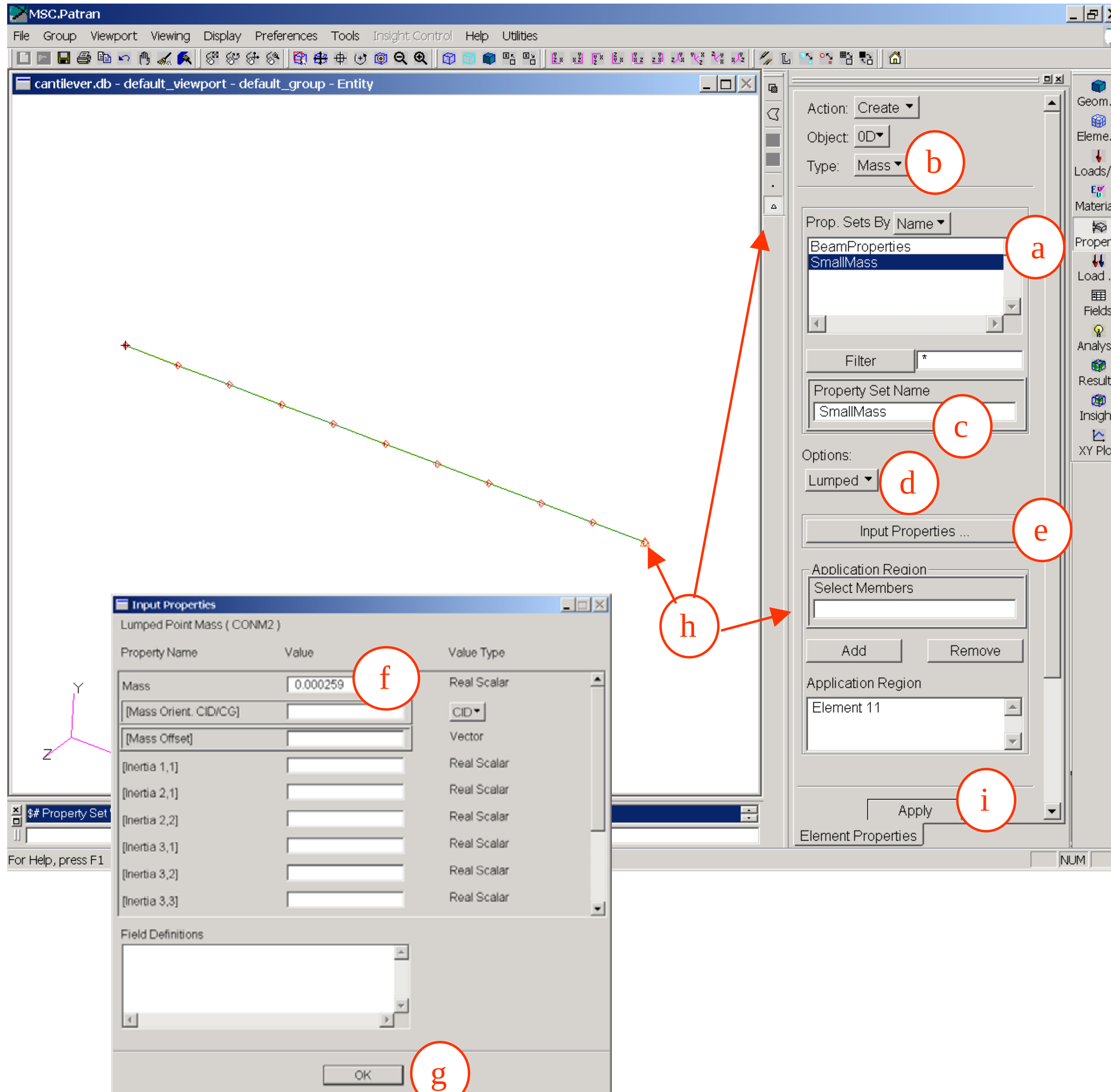
Since Auto Execute is on you do not need to hit Apply to create the mass.



Assign Material Property To Small Mass

Assign Material Property To Small Mass

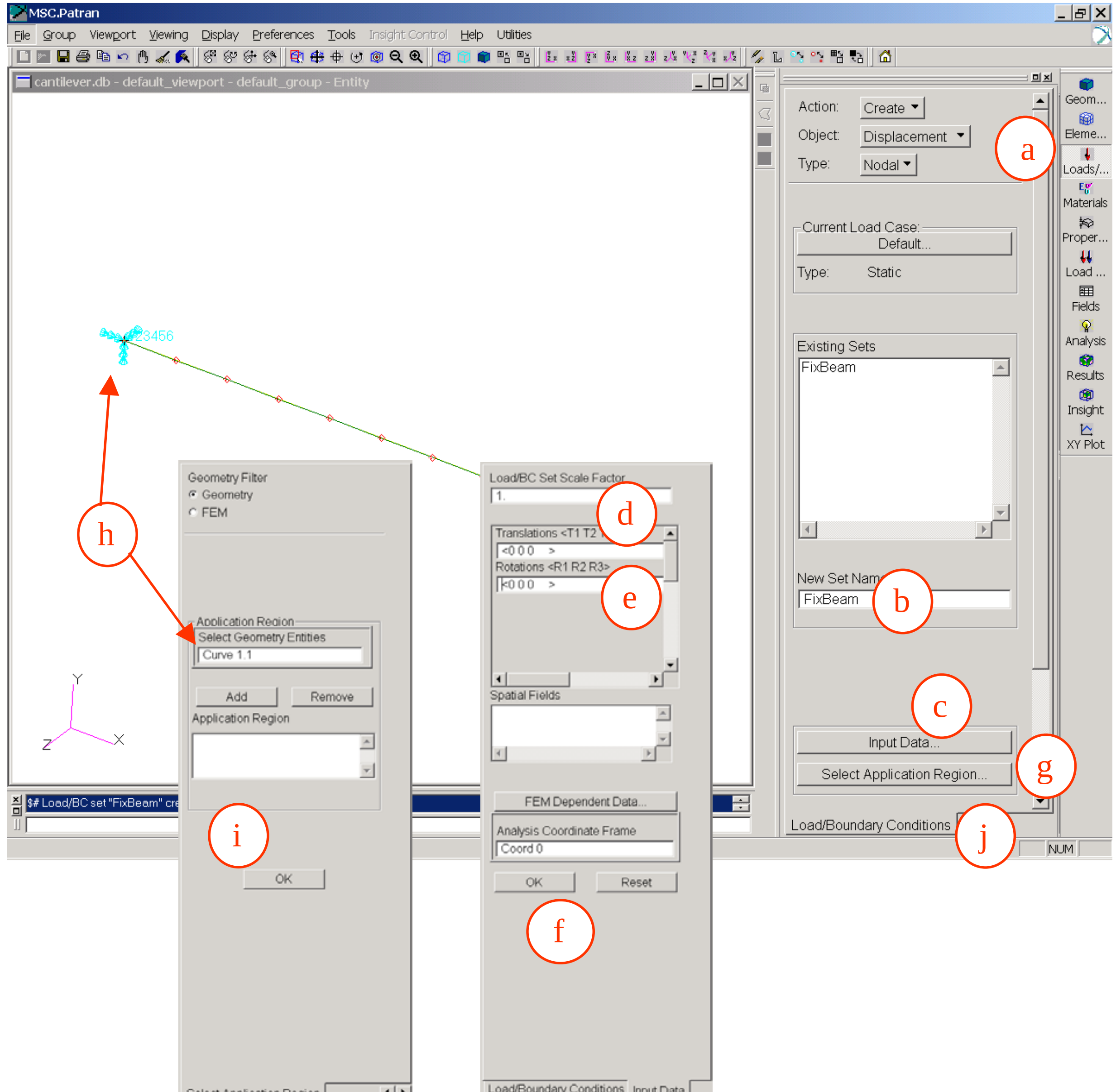
a. Click on Properties Form
b. Create / 0D / Mass
c. Property Set Name: SmallMass
d. Options – Lumped
e. Input Properties…
f. Mass: 0.000259 lb_m
g. OK
h. Select Members: Click on Point Element in Selection Tookbar (The Triangle), Then Select the Element 11 (The Point Mass Element)
i. Apply



Constrain The Beam

Constrain the End Of The Beam

a. Click On the Loads Form
b. New Set Name: FixBeam
c. Input Data…
d. Translational: <0,0,0>
e. Rotational: <0,0,0>
f. OK
g. Select Application Region…
h. Click On Point In the Selection Toolbar, then Click on The Point at the end of the Beam (opposite Small Mass – Curve 1.1), Add to Application Region.
i. OK
j. Apply (Scroll Down Form To See)

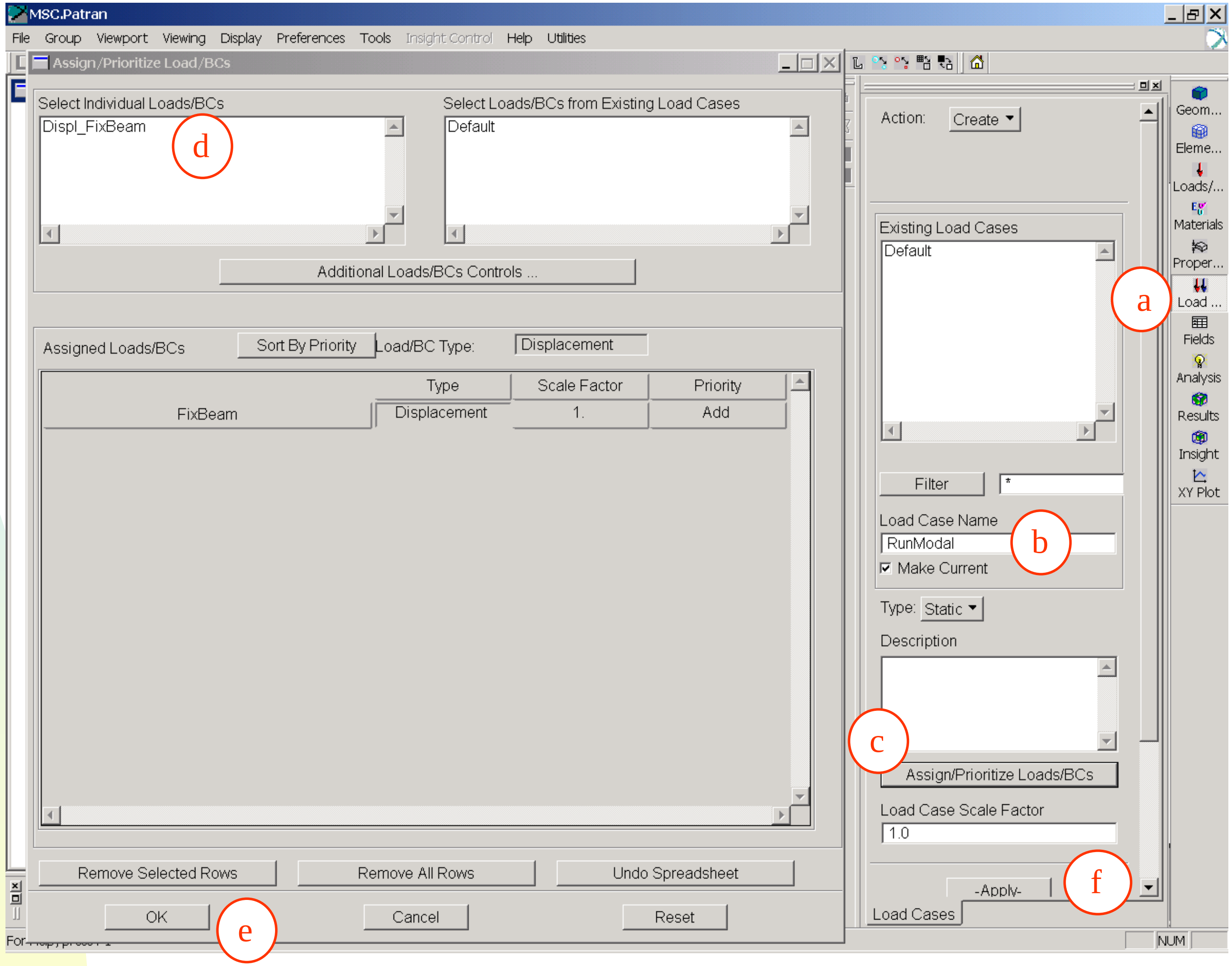


MSC SOFTWARE
SIMULATING REALITY

Create a LoadCase

Create a Load Case Called RunModal

a. Click On LoadCase Form
b. Load Case Name: RunModal
c. Assign/Prioritize Loads/BC's…
d. Select the Displ_FixBeam Constraint
e. OK
f. Apply



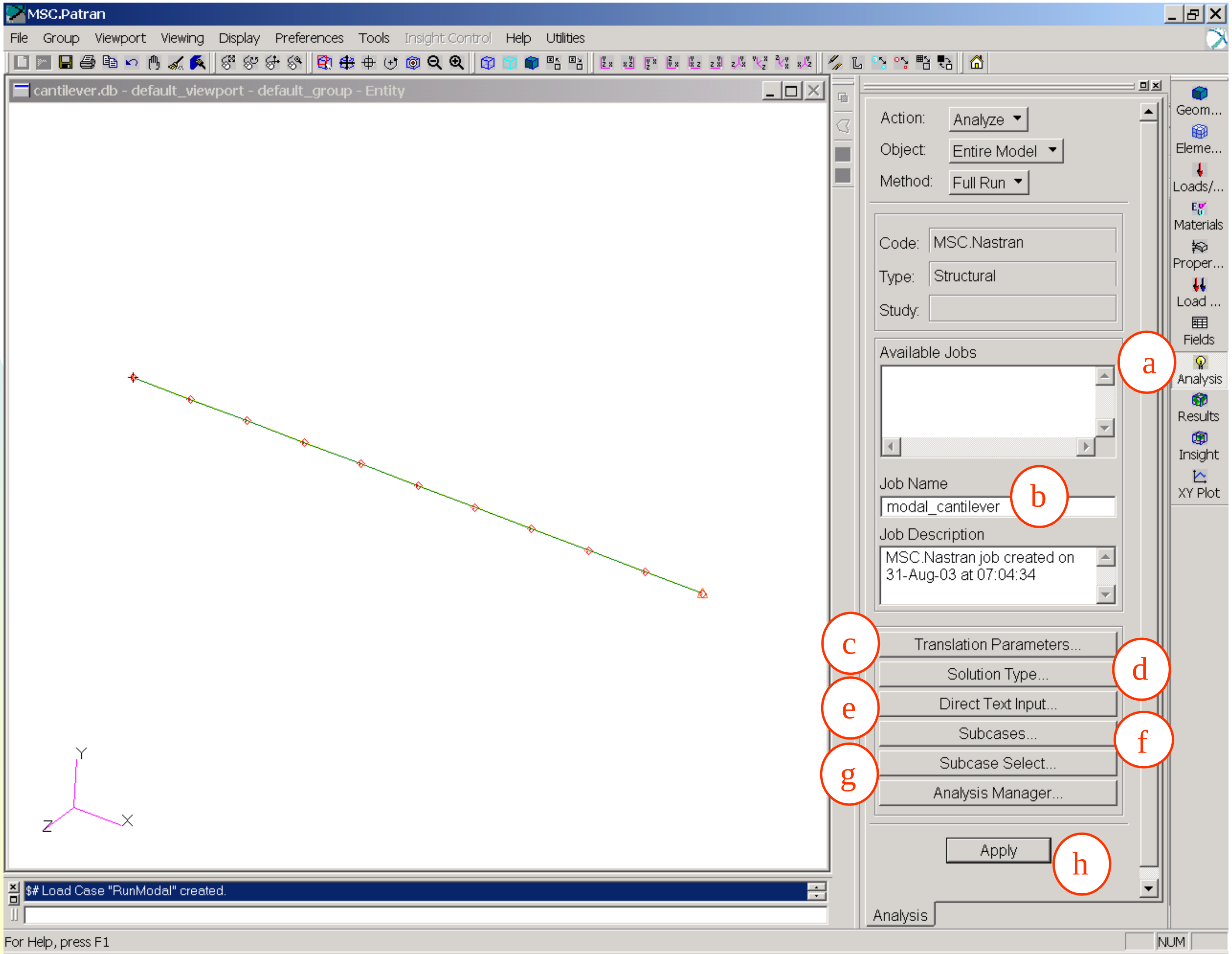
Set Up Modal Analysis

Setup The Modal Analysis

a. Click on the Analysis Form
b. Job Name: modal_cantilever

See Upcoming Slides for these

c. Translation Parameter…
d. Solution Type…
e. Direct Text Input…
f. Subcases…
g. Subcase Select…
h. Apply (To Start Job – after you do the next few slides covering parts c – g)

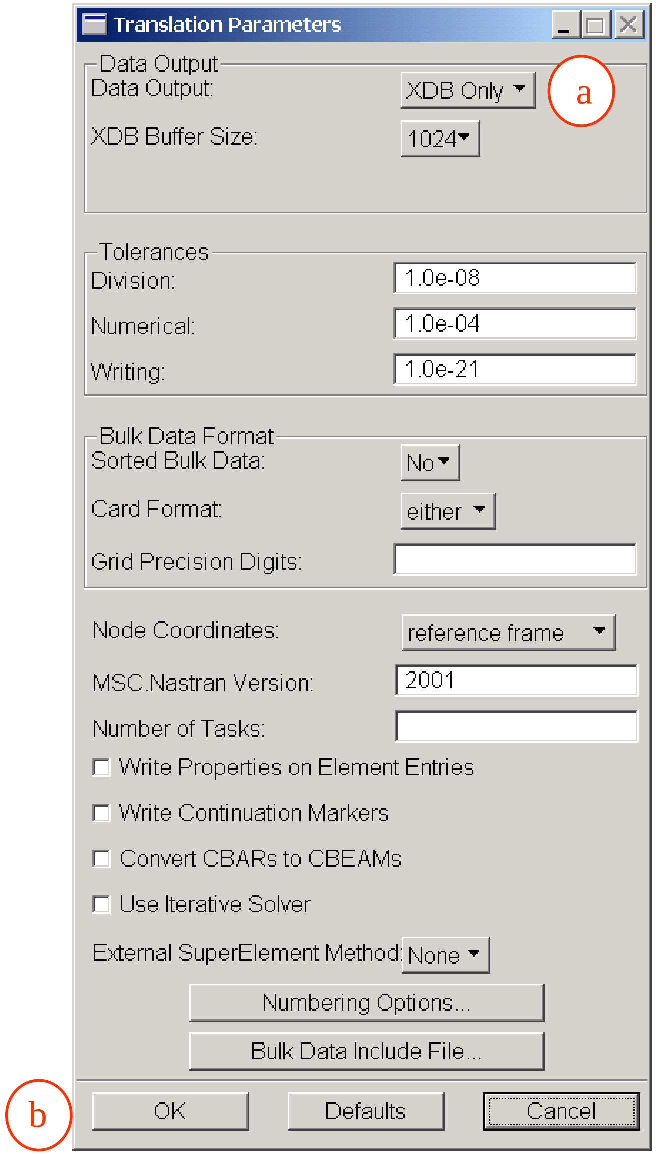


Analysis – Translational Parameters

Translational Parameters…

a. Data Output: XDB Only

b. OK

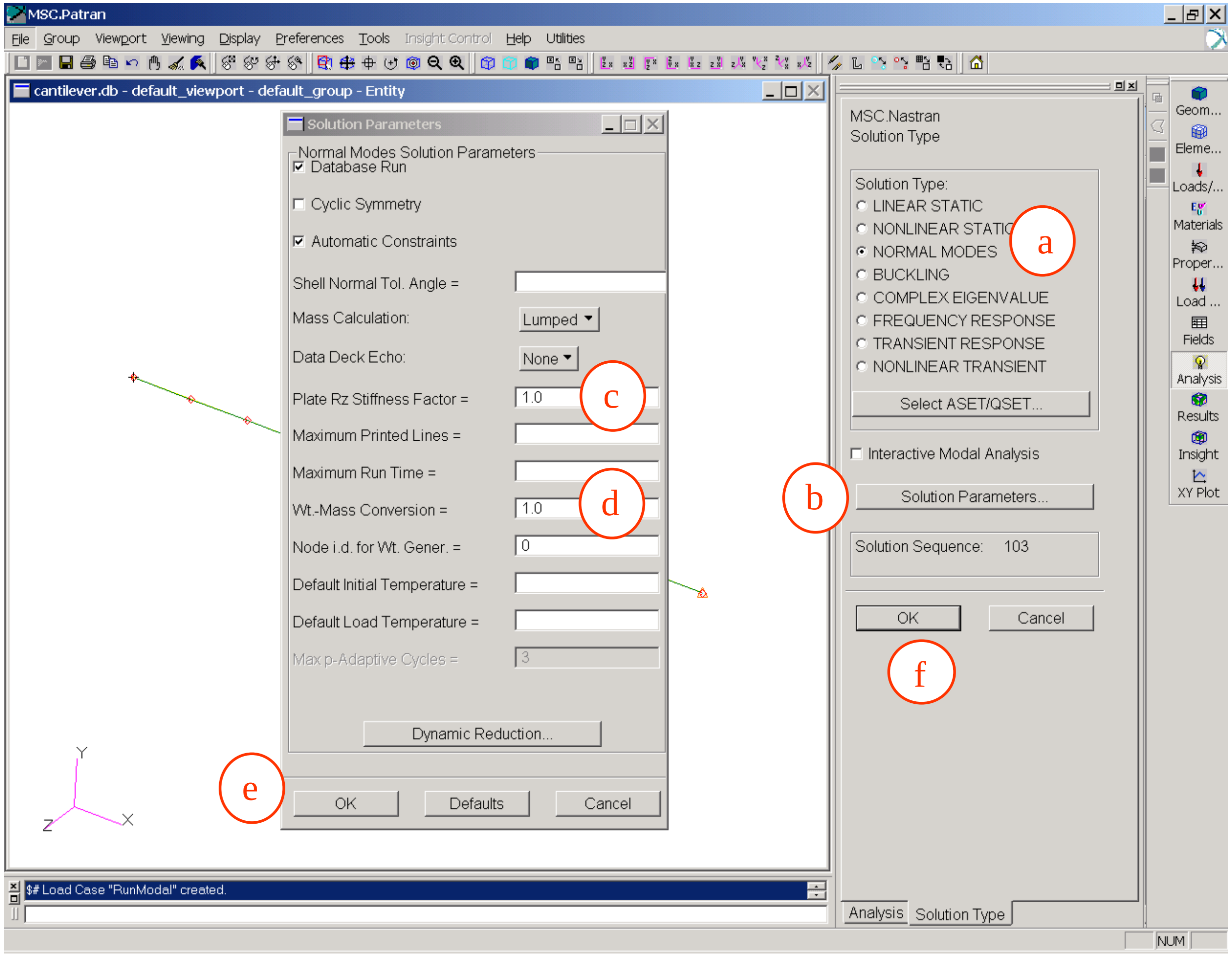


Analysis – Solution Type

Solution Type…

a. Normal Modes (103)
b. Solution Parameters…
c. Plate Rz Stiffness Factor = 1.0
d. Node i.d. for Wt. Gener. = 0
e. OK
f. OK

By Setting Node ID for Weight Generation to 0 (GRDPNT=0), we are asking Nastran to calculate moments about the global coordinate system.

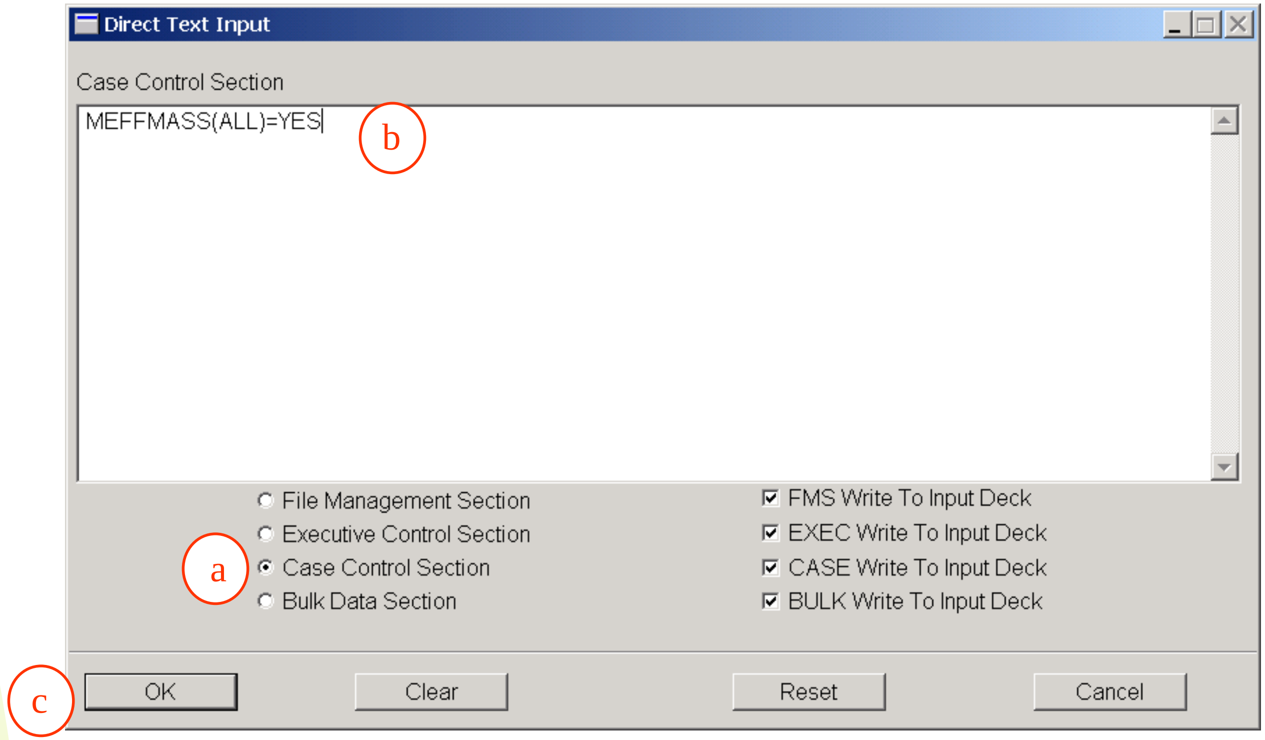


Analysis – Direct Text Input

Direct Text Input…

a. Click on Case Control Section
b. Type in MEFFMASS(ALL)=YES
c. OK

This will create a Modal Effective Mass Table for your Model. This table is a very effective way to determine which modes will most likely contribute the most damage to your part.



Analysis - Subcases

Subcases…

a. Click on RunModal Subcase from the Available Subcases Choices

b. Subcase Parameter…

   1. Lower = -0.1

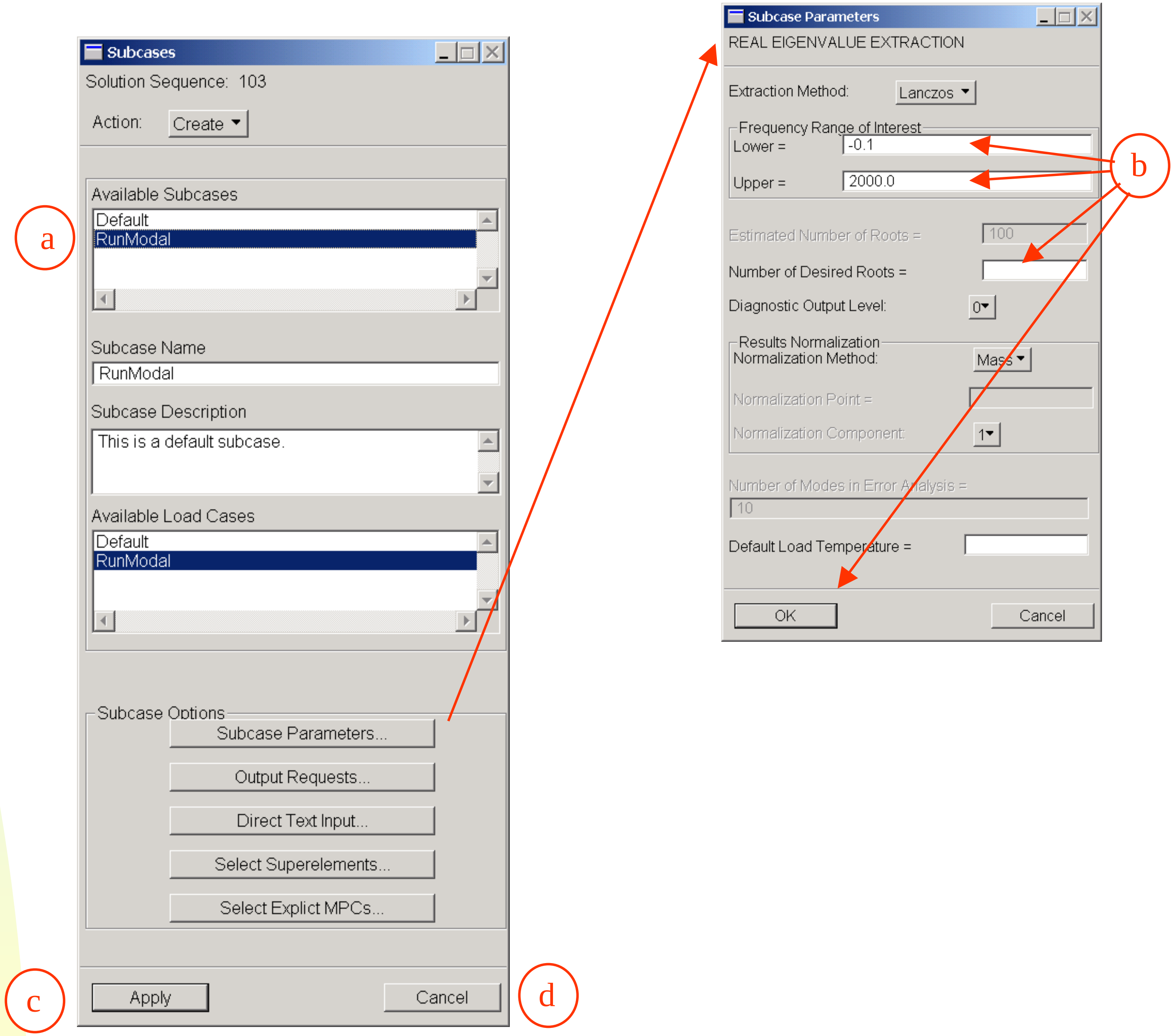
   2. Upper = 2000.0

   3. Clear the 10 from Number of Desired Roots

   4. OK

c. Apply

d. Cancel

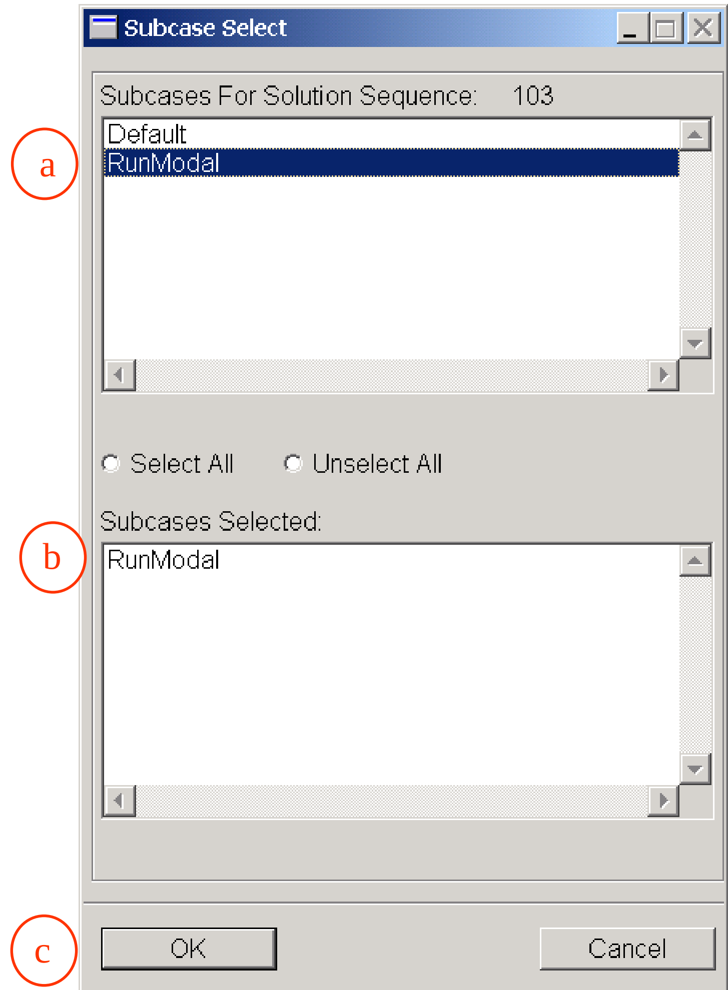


Analysis – Subcase Select

Subcase Select…

a. Click on RunModal in Top Box to add to Lower Box (Subcases Selected)
b. Click on Default in Lower Box to remove it from the Subcases Selected
c. OK

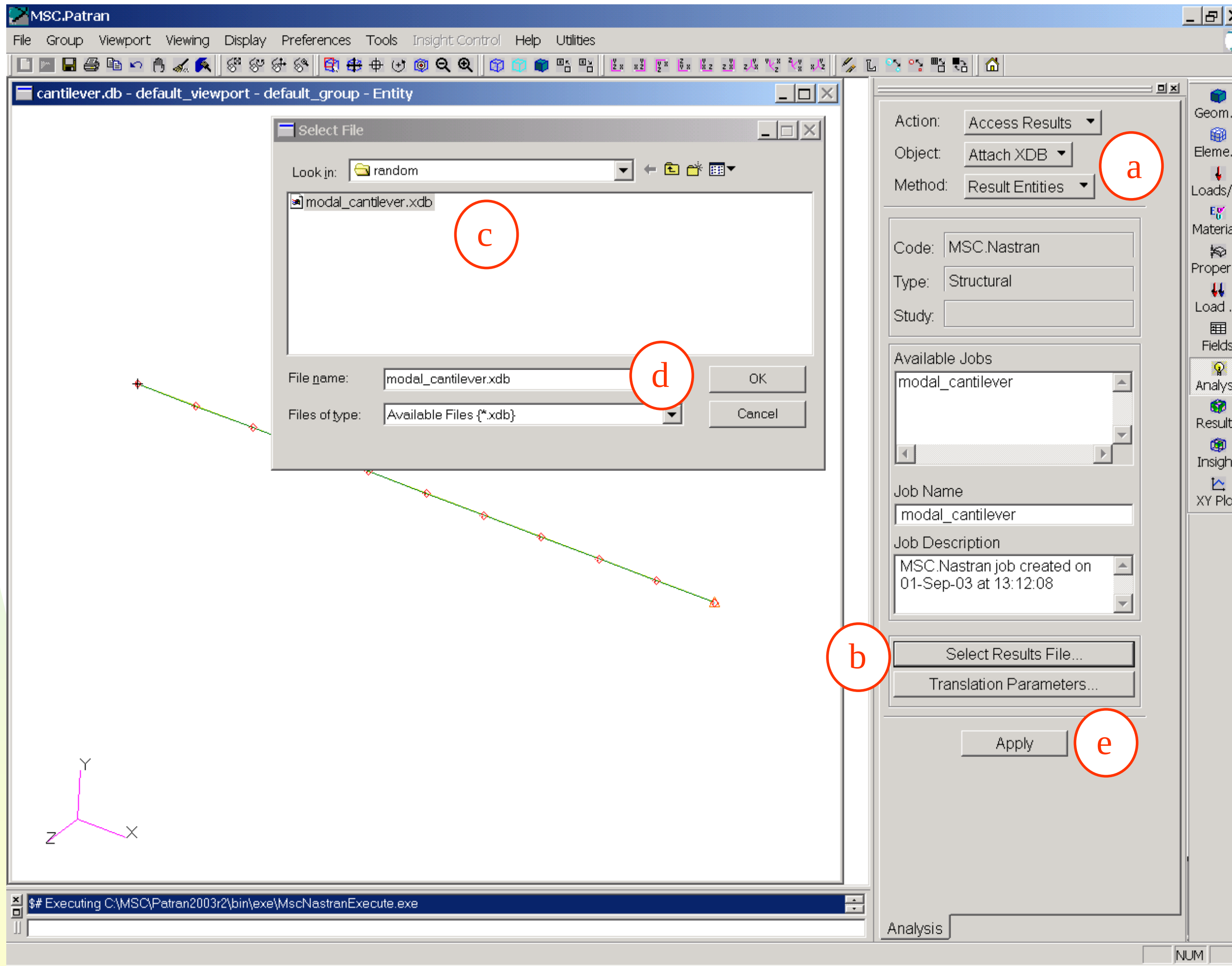
Once you hit OK, you will then return to the main analysis form. Hit Apply and your modal job will begin in Nastran. This job should only take less then 1 minute to solve.



Analysis – Read Results

Read Results (From the Analysis Form)

a. Access Results / Attach XDB / Result Entities
b. Select Results File…
c. Click on modal_cantilever.xdb
d. OK
e. Apply



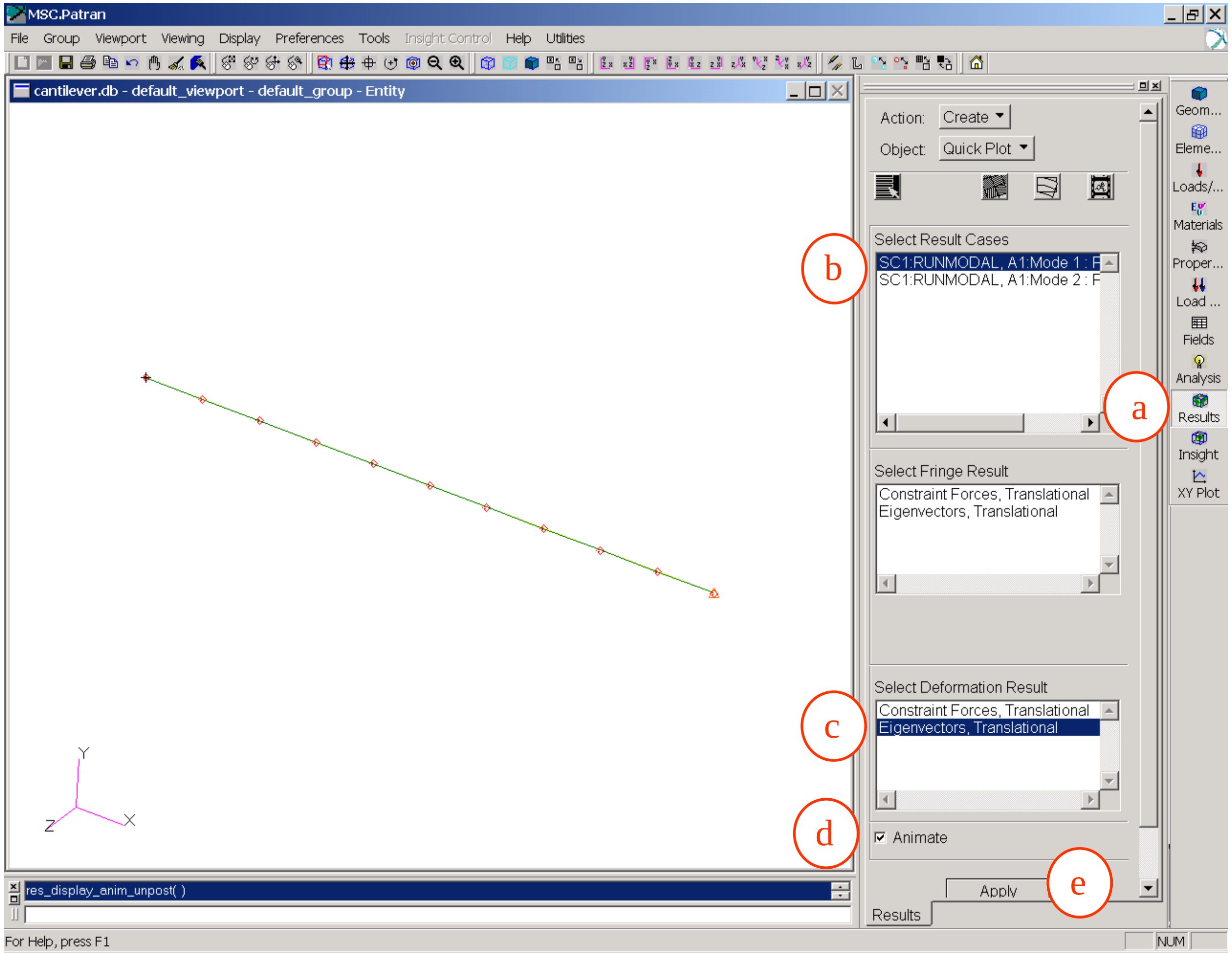
MSC SOFTWARE
SIMULATING REALITY

Results – View Animation

View the Modal Animation

a. Click On The Results Form
b. Click on Mode 1 in Result Case window (492.5 Hz)
c. Click on Eigenvectors, Translational
d. Animate
e. Apply

Note that in this model, there are only 2 modes between – 0.1 to 2000.0 Hz. In a large model with lots of degrees of freedom, this table would be pretty full. In these cases, you will want to check the Modal Participation Table in the *.f06 file to determine which mode and direction is likely to contribute the most to your model. In most cases, it is the mode with the highest mass participation in the direction that the model is being driven.

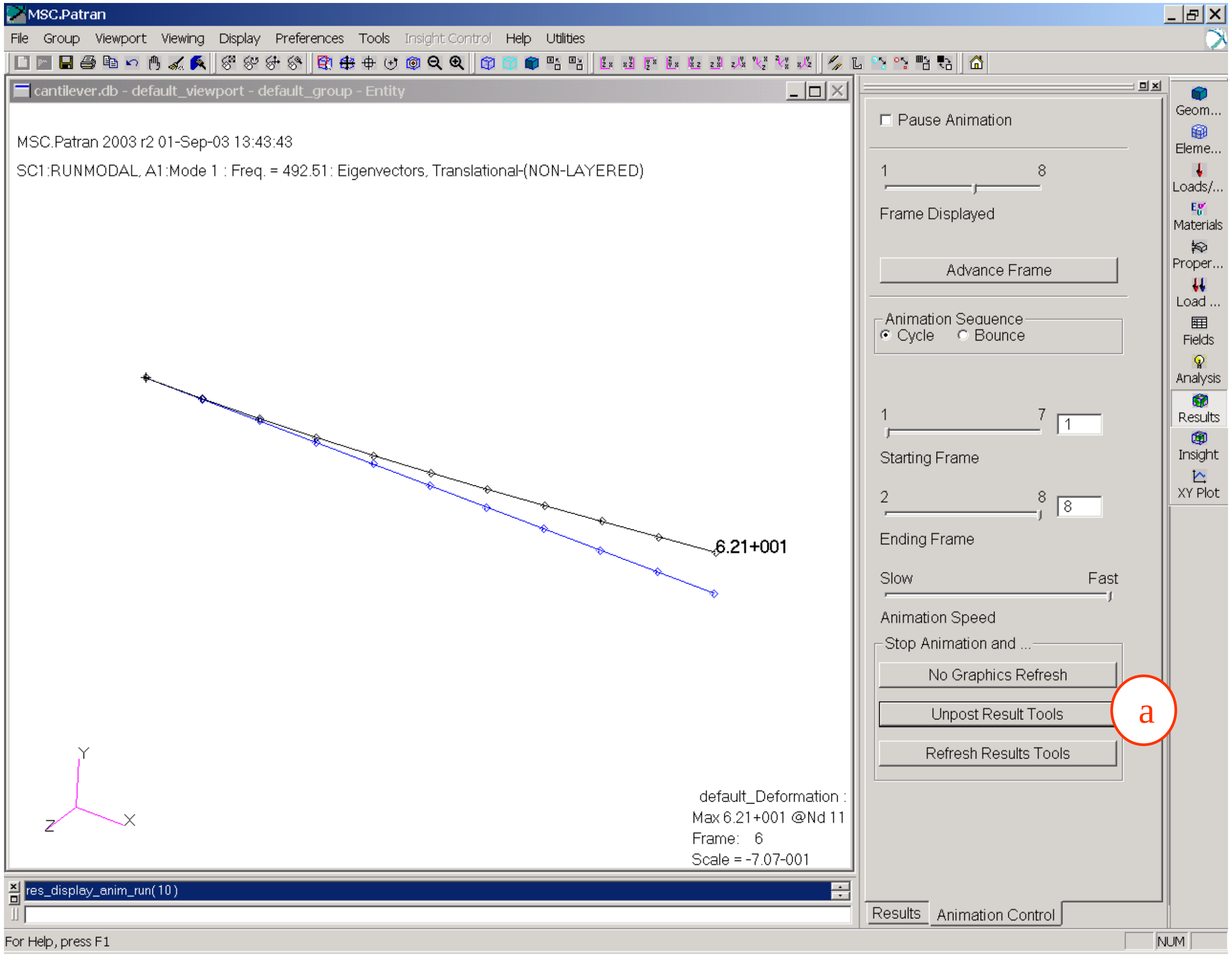


Results Animation

Once the Animation has began, you have many options to display.

a. Click on Unpost Result Tools to Stop and Return to the Main Results Form.

Please note that the animation is scaled against the overall size of the model, and does not reflect actual displacement. This is because at this point there is no load driving the model.



MSC SOFTWARE
SIMULATING REALITY

Results – Comparison Against Hand Analysis

Inputs to Frequency Problem

Youngs Modulus	$E = 10 \cdot 10^{6} \cdot psi$		
Possions Ratio	$\nu = 0.33$		
Shear Modulus	$G = \frac{E}{2 \cdot (1+\nu)}$	$G = 3.759 \times 10^{6} psi$	
Shear Factor	$K_{sf} = \frac{5}{6}$	$K_{sf} = 0.833$	
Length of Beam	$L = 10 \cdot in$		
Width and Height of Beam	$b = 1in$	$h = 1in$	
Area	$A = b \cdot h$	$A = 1\,in^{2}$	
Weight of Small Mass	$W = 0.1lbf$	$M = \frac{W}{g}$	$P = W$
Moment of Inertia	$I = \frac{1}{12} \cdot b \cdot h^{3}$	$I = 0.083\,in^{4}$	

MSC SOFTWARE
SIMULATING REALITY

Results – Comparison Against Hand Analysis

Hand Calculation of 1st Mode Frequency of Cantilever Beam to check FEA

Results are exact with FEA = 492.5 Hz vs. Hand Analysis = 492.5 Hz, 0.0% Error

Bending Stiffness

$$K_b = \frac{3 \cdot E \cdot I}{L^3} \qquad K_b = 2.500 \times 10^3 \frac{lbf}{in}$$

Shear Stiffness

$$K_s = \frac{K_{sf} \cdot A \cdot G}{L} \qquad K_s = 3.133 \times 10^5 \frac{lbf}{in}$$

Total Stiffness
(Springs In Series)

$$K_T = K_b \cdot \frac{K_s}{K_s + K_b} \qquad K_T = 2.48 \times 10^3 \frac{lbf}{in}$$

Calculated Frequency

$$f = \frac{1}{2 \cdot \pi} \cdot \sqrt{\frac{K_T}{M}} \qquad f = 492.502\,Hz$$

Begin Random Vibration Analysis

Create A Relative Displacement MPC

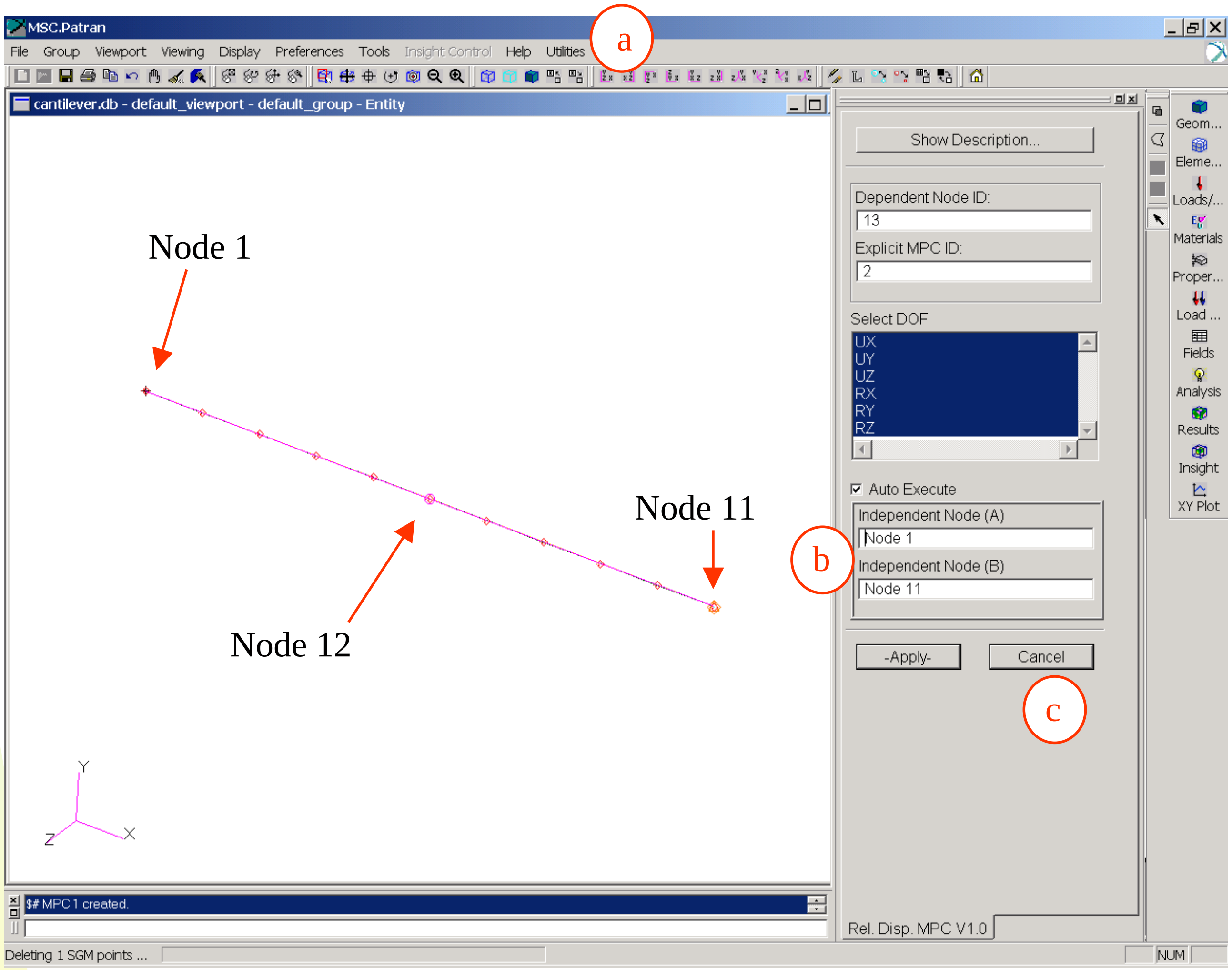
a. Utilities / FEM-General / Relative Displacement MPC

b. Click on Node 1 and Node 11

Auto Execute will automatically create an Explicit MPC between Node 1, 11 and a newly created Node 12, ½ the distance between node 1 and 11.

The Relative Displacement MPC will come into play when it is time to view displacement between to points of concern. An example where this helpful is to determine the displacement between the center and edge of an electronic board that is part of a larger box. This will subtract out the large mass displacement (more to come).

c. Cancel to close form



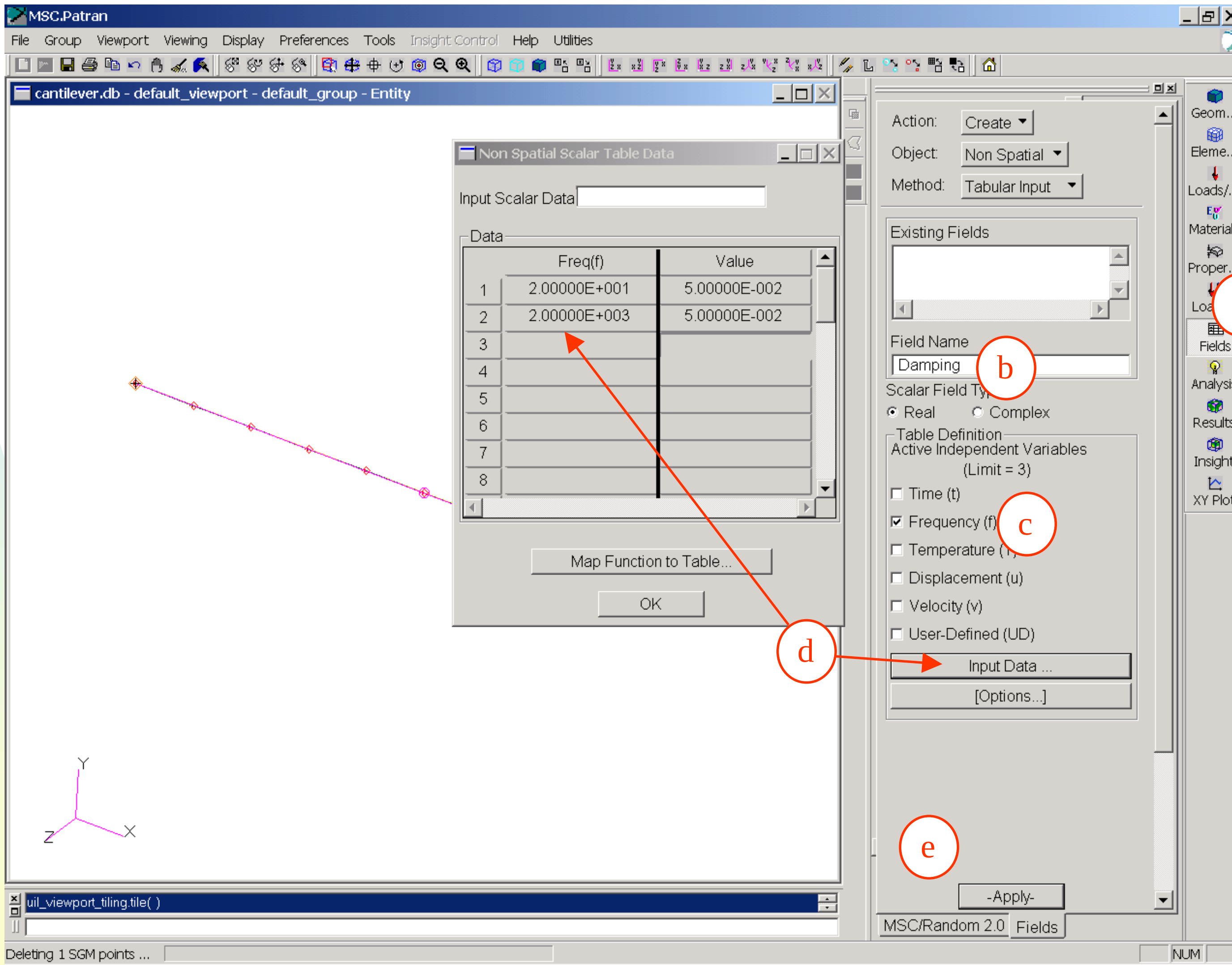
Create a Non-Spatial Damping Field

Create a Damping Field

a. Select the Field Form
b. Name: Damping
c. Change to Frequency Table Definition
d. Input Data…
   1. Fill in the table (click in box on table, then enter number in Input Scalar Data field)
   2. OK
e. Apply

Frequency [Hz]	Damping
1.0	0.05
2000.0	0.05

$$Q = \frac{1}{2\zeta}$$



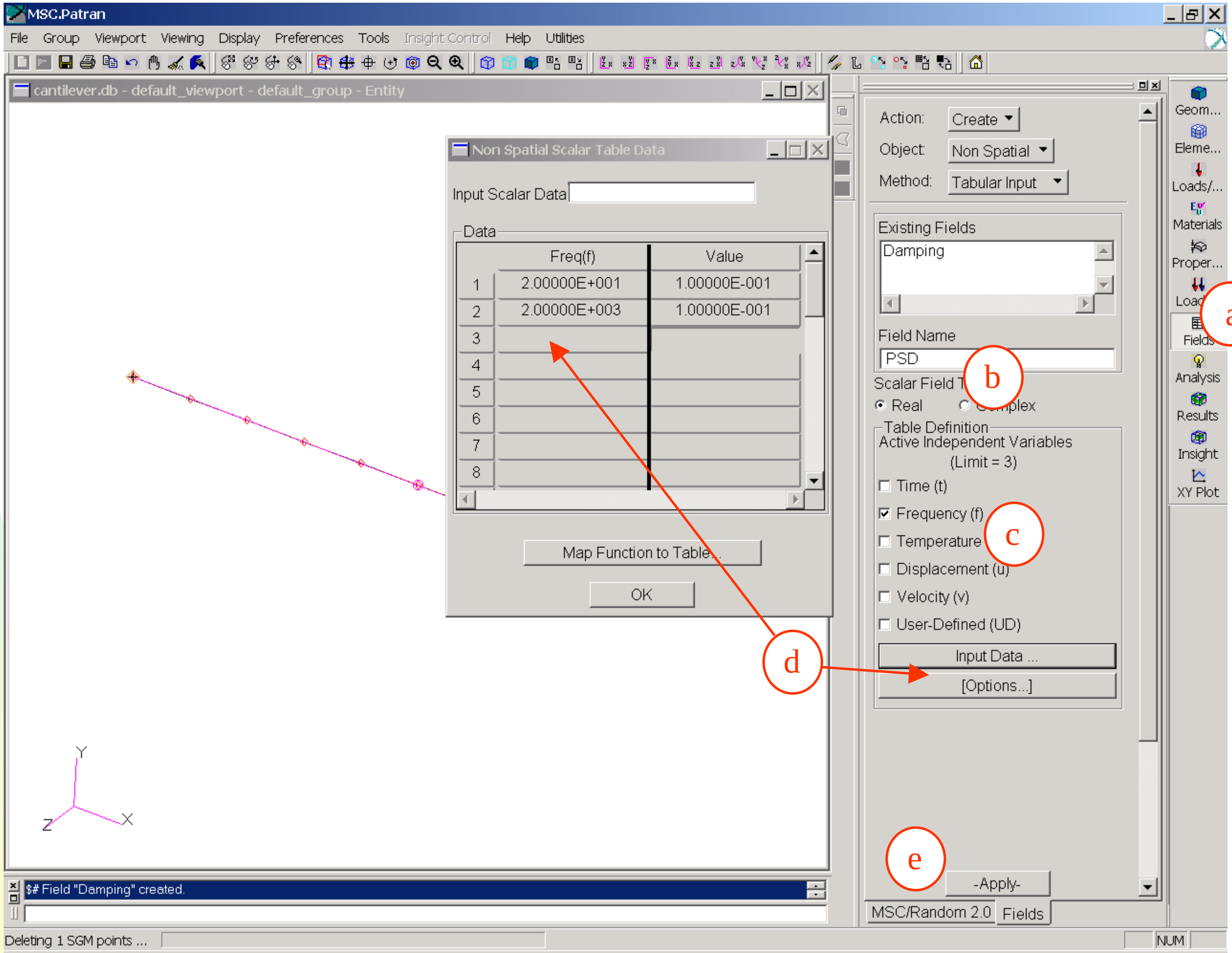
5% Damping (ζ) corresponds to an amplification (Q) of 10

5. Create A Non-Spatial PSD Input

Create a Power Density Spectrum (non-spatial) Field

a. Select the Field Form
b. Name: PSD
c. Change to Frequency Table Definition
d. Input Data…
    a. Fill in the table (click in box on table, then enter number in Input Scalar Data field)
    b. OK
e. Apply

Frequency [Hz]	PSD [G^2/Hz]
20.0	0.10
2000.0	0.10

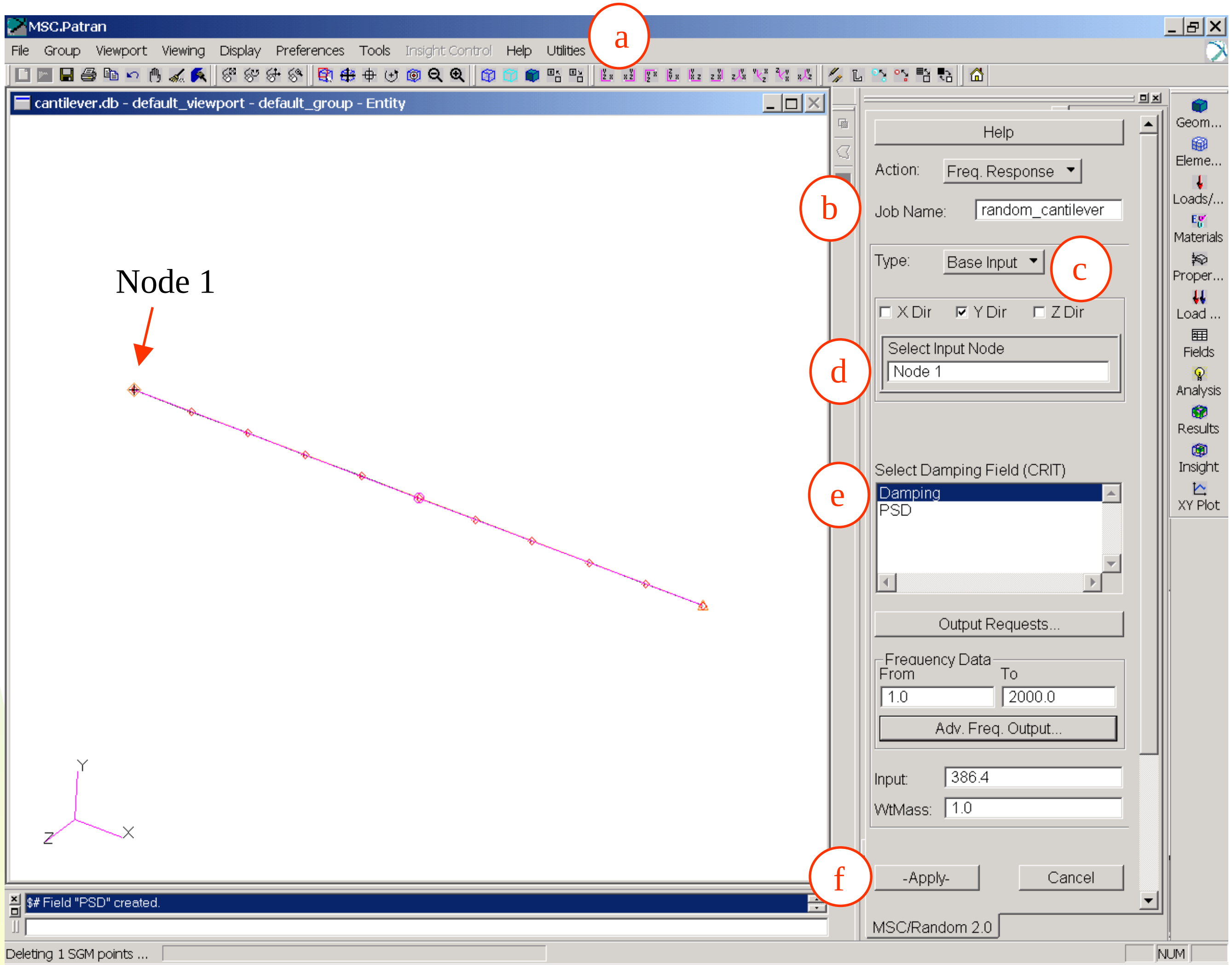


MSC SOFTWARE SIMULATING REALITY

MSC.Random

Create the Random Run

a. Click on Utilities / Applications / MSC.Random…
b. Job Name: random_cantilever
c. Check Only Y-Direction for this Analysis
d. Select Input Node: Node 1 (this the node where the large mass will be placed).
e. Select Damping Field (CRIT): Damping
f. Apply



Note: In cases of a models with multiple points of constraint, a node away from the model should be created, with an RBE2 created between the constraint points and the large mass node. Since this model is simple, the large mass can be placed at the end of the beam.

Output Requests: Allows the users to put elements or nodes into groups to reduce the amount of output.

Adv. Freq. Output: Allows users to select additional ways methods of creating output points for the frequency response (Corresponds to the various FREQ cards in Nastran)

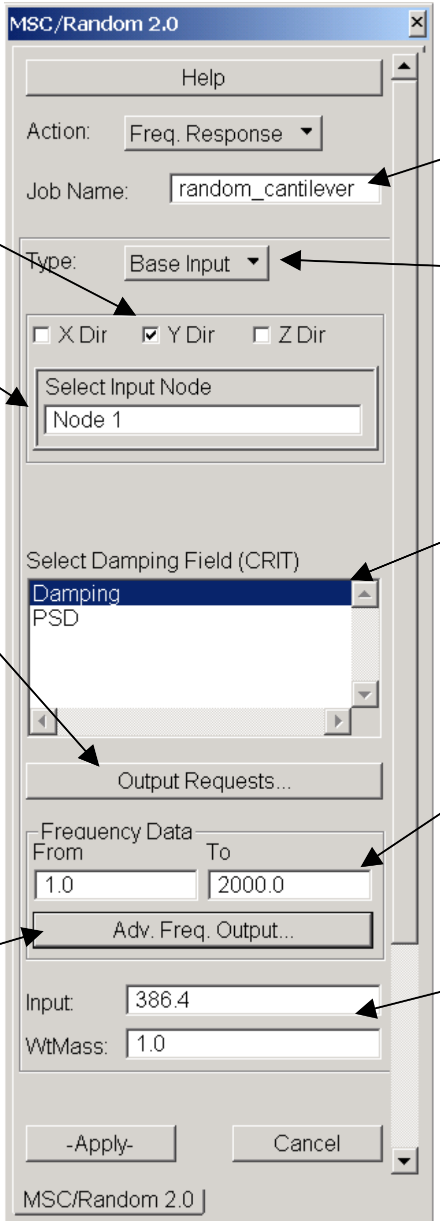
Closer Look At MSC.Random Form

Directions to drive the model. This will create subcases based on selections.

Node to Drive with Input

Allows the user to define groups to calculate outputs on. Default is ALL; however due to disk space, a user may want to limit the calculations to a set of elements and nodes.

Output requests allow a user to describe how much definition should be included in the output PSD (FREQ cards). Generally, you should just accept the defaults.



Jobname

Base Input (Large Mass)

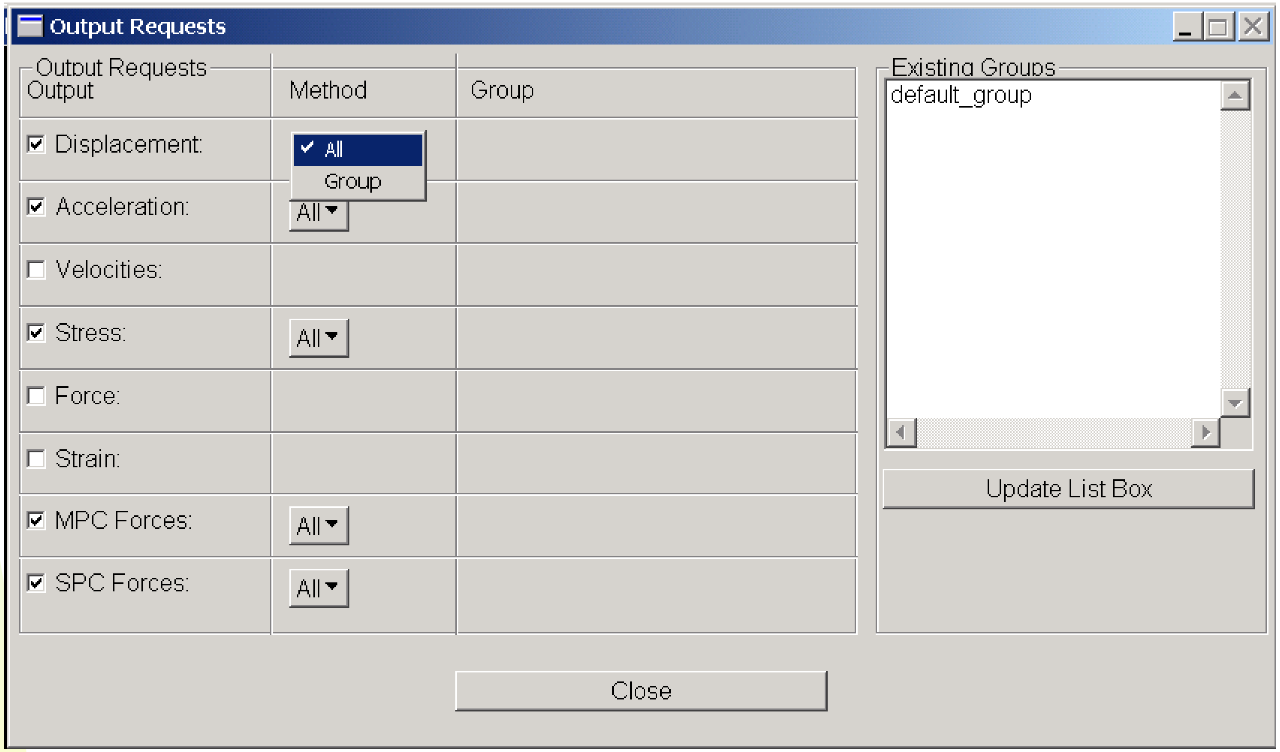
Damping Field to use. Damping is created as a Non-Spatial Tabular Field, using frequency.

Range to perform the transfer function (harmonic frequency response).

Input and Mass default to using mass density. If you use weight density, you will need to modify both values.

Closer Look At The Output Requests Form

- This Form allows the user to define specific groups of nodes and elements for various output
- Use this form if your model is very large and you do not have the resources (disk space or time) to calculate values for all nodes and elements in your model.



Closer Look at the Advanced Frequency Output Form

FREQ: Allows the user to specify explicit frequencies to calculate output

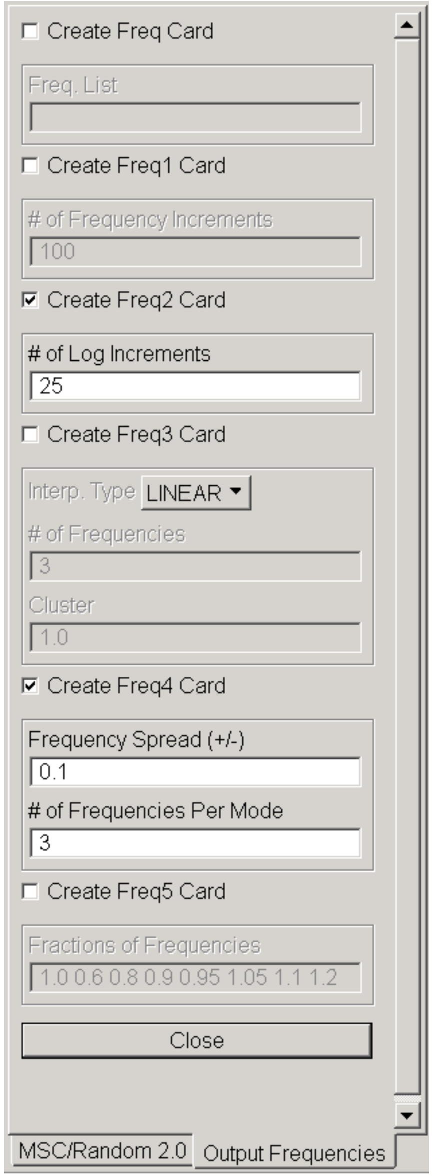
FREQ1: Allows the user to specify output frequencies based on increments (linear)

FREQ2: Allows the user to specify output frequencies based on increments (logarithmic) – default = 25.

FREQ3: Allows the user to specify the number of output frequencies between two modes

FREQ4: Allows the use to define the “Spread” (number of recovery points) around a natural frequency – default is 3 modes at +/- 0.1 around a mode.

FREQ5: Allows the user to define a frequency range and fractions of the natural frequencies within that range.



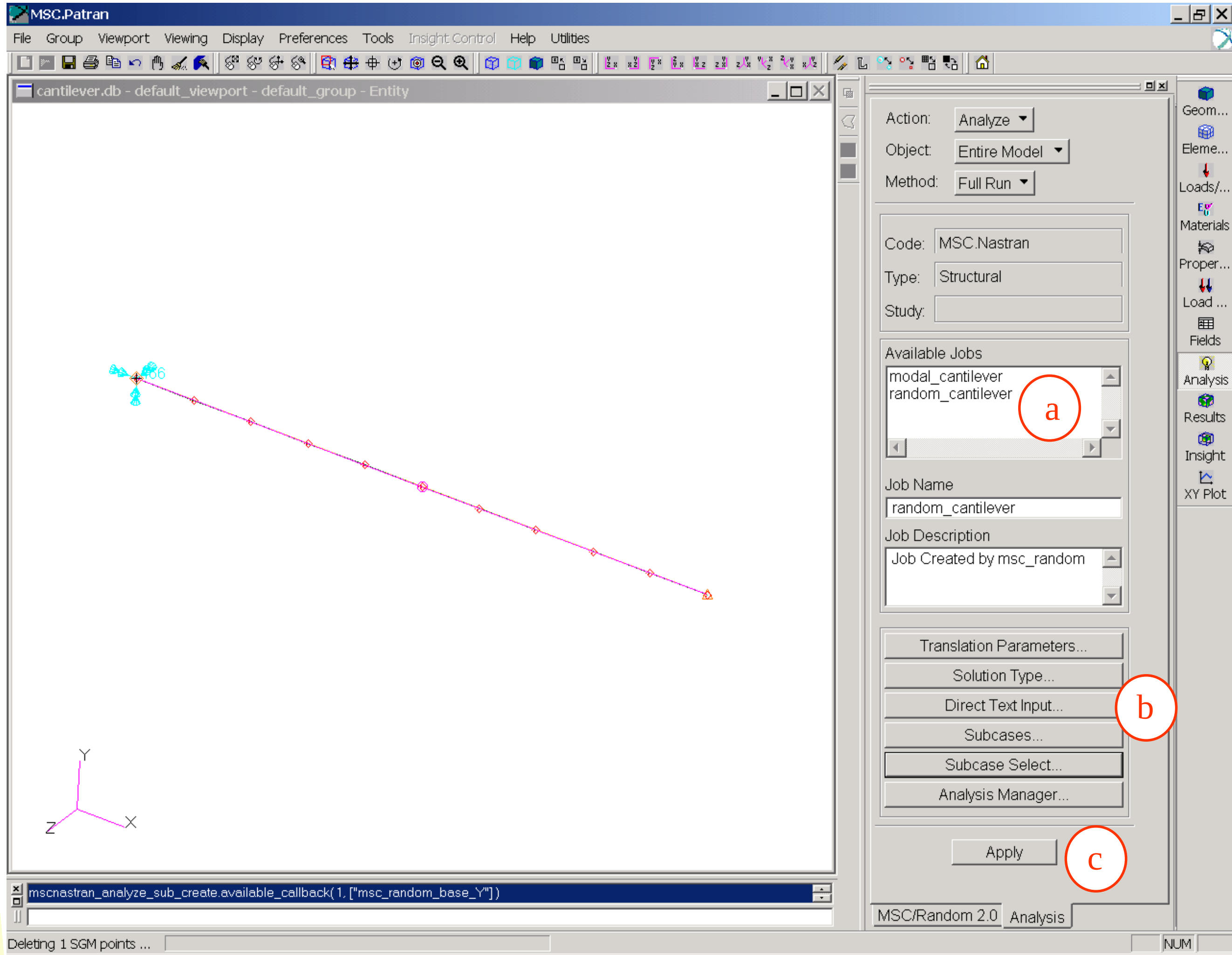
Launch Random Analysis Job

After Hitting Apply in MSC.Random, the Analysis From will Open.

a. Click on the Available Job: random_cantilever
b. Direct Test Input… (See next Page)
c. Click Apply to start the job

(Hit Yes when asked about writing over the Analysis job)

At this point you have just run the frequency response portion of the analysis to develop transfer functions. It is the equivalent of ringing your system with a unit load and recording the response.



Direct Text Input…

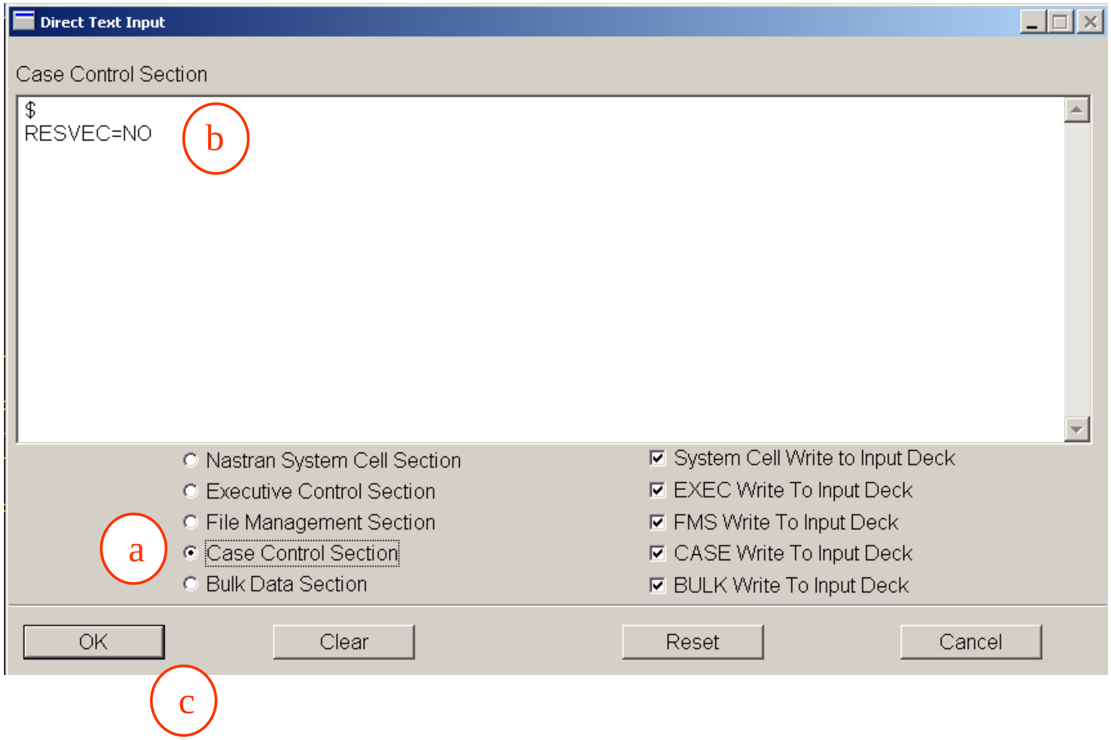
A. Case Control
B. RESVEC=NO
C. OK

NOTE: In the majority of models you will want to allow residual vector calculations (default in Nastran 2004 and higher) as they add to the accuracy of the modal solution.

Therefore, if you are using this slide as an example, in most models, and on non-NT machines this slide should be ignored.

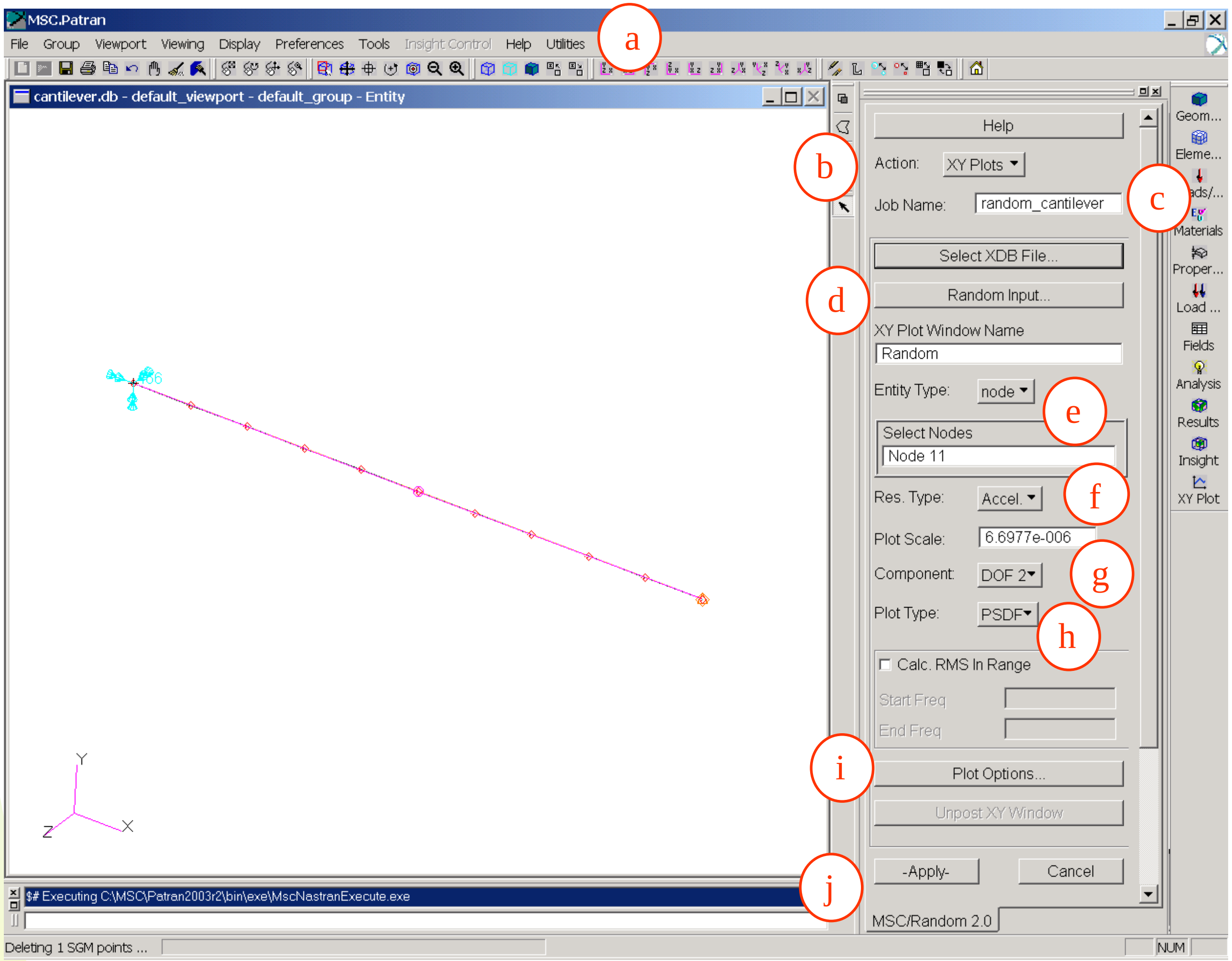
This error with residual vectors only shows on the NT version and will be corrected in future versions.

Also note that the RESVEC error does not appear with the modal solution, only the frequency response solution.



Post-Process With the PSD Input

a. Return to MSC.Random (Utilities / Applications / MSC Random
b. Action: XY Plots
c. Select XDB File… Select the random_rantilever.xdb file
d. Random Input… (See next Slide)
e. Select Node 11
f. Res. Type: Accel
g. DOF 2 (Corresponds to Y Direction)
h. PSDF (Power Spectral Density Function (Transfer Function x PSD input)
i. Plot Options… (See next Slide)
j. Apply



Random Analysis Input

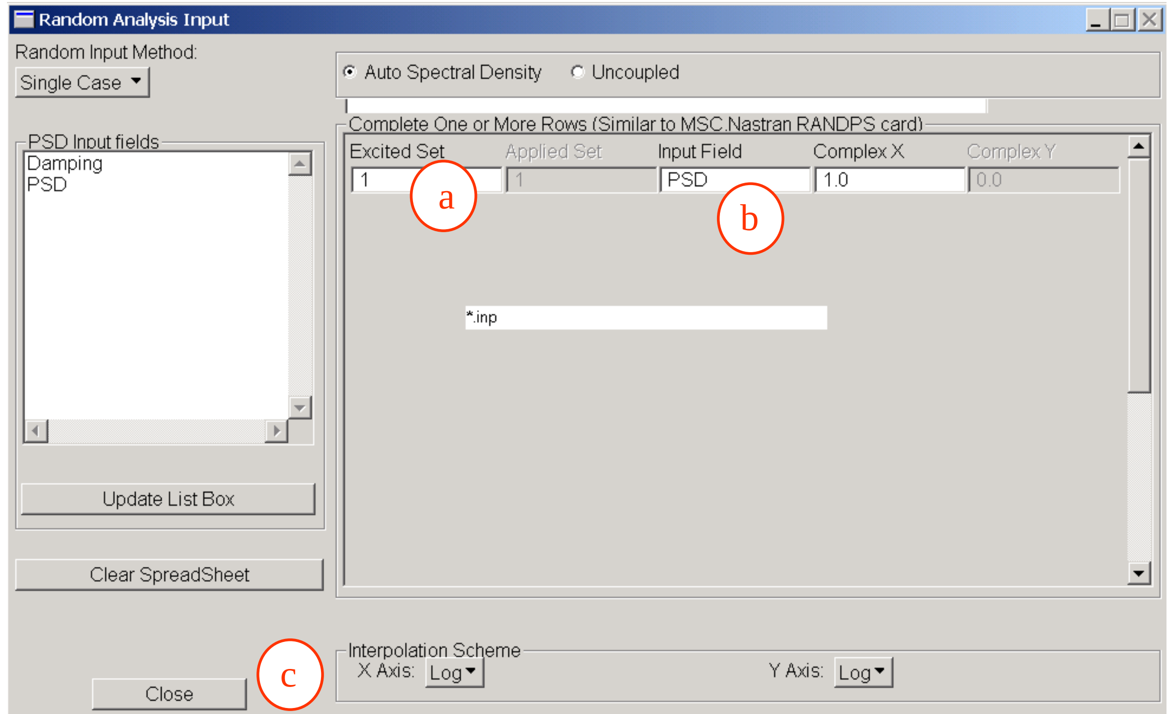
Continuation of Previous Slide

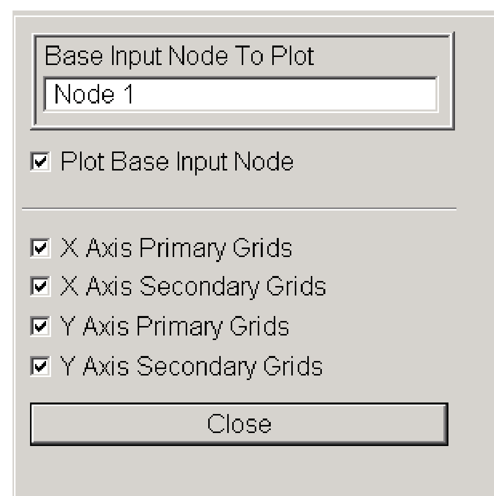
Random Analysis Input…

a. Click on the Excited Set and Choose the MSC_RANDOM_BASE_Y subcase from the Available Subcase list (not shown in picture)
b. Click On Input Field, and Select PSD from the PSD Input Fields
c. Close

Plot Options…

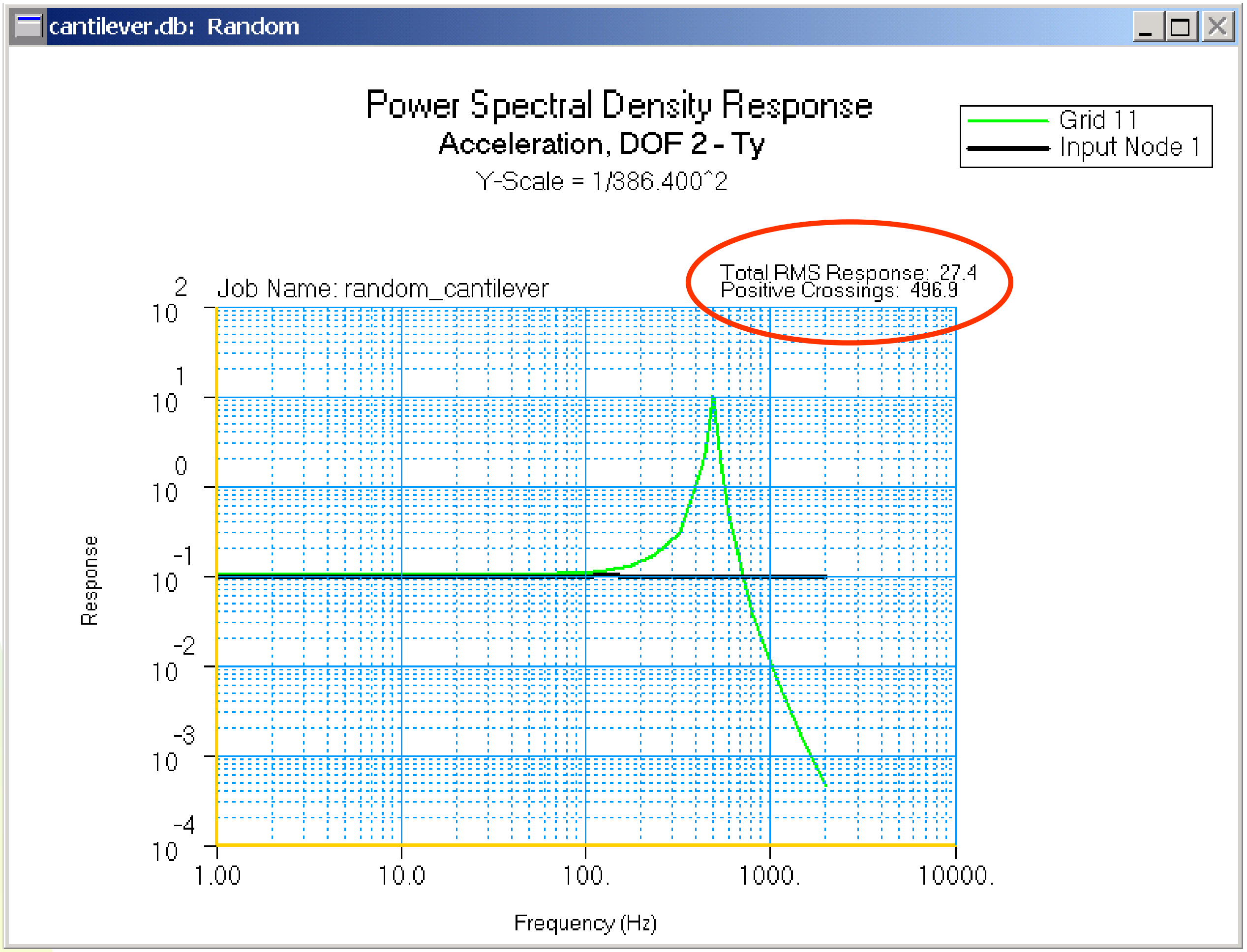
d) Turn on Plot Base Input Node (This will allow the Large Mass Node 1 to be viewed with Node 11)
e) Close





Acceleration Output From Node 11

- The plot on the left contains the large mass (node 11) response, which matches the input (0.1 G^2/Hz input)
- The G_{rms} Response, which corresponds to the area under the response curve (node 11) is 27.4 G_{rms}. This value is often used to characterize a curve.
- In the simple cantilever case, the Positive Crossing number corresponds very closely to the natural frequency of the model.



Miles Equation To Check Acceleration

Note that the G_{rms} calculated value (27.8 G_{rms}) matches very closely to the FEA Calculated Value (27.4 G_{rms}).*

Calculated Frequency $f = \frac{1}{2\cdot\pi}\cdot\sqrt{\frac{K_T}{M}}$ $f = 492.502\,\text{Hz}$

PSD Input $PSD = 0.1\cdot\frac{g^2}{Hz}$ $g = 386.089\frac{in}{sec^2}$

Amplification Factor $Q = 10$ $\zeta = \frac{1}{2\cdot Q}$ $\zeta = 0.05$

Miles Equation $G_{rms} = \frac{\sqrt{\frac{\pi}{2}\cdot f\cdot PSD\cdot Q}}{g}$ $G_{rms} = 27.8$

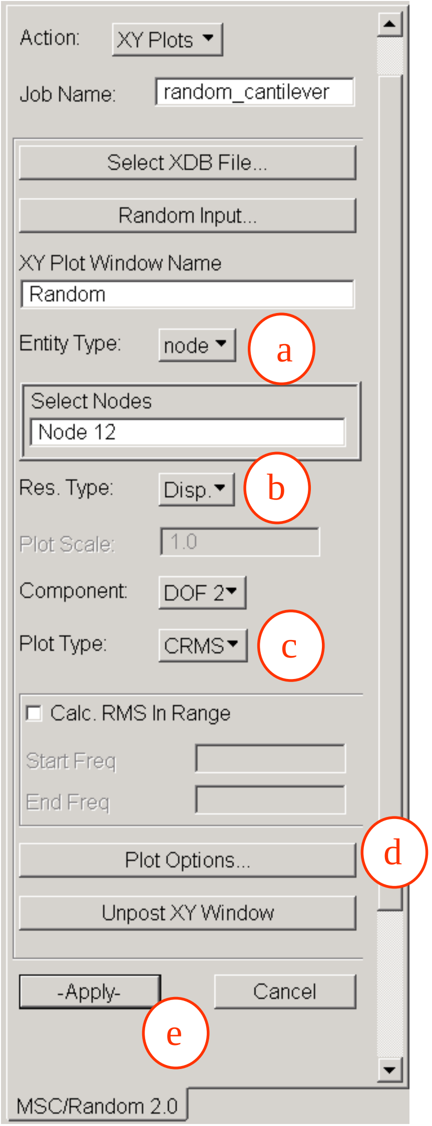
*The G_{rms} is slightly off due to FEA solution being an approximation subjected to the number of points used to describe the response curve. However, in this case, it does not affect the overall accuracy of the solution. By setting the FREQ4 card in Adv. Output Requests in MSC.Random to 5 (default=3), the user will find the solution to be more exact. However, keep in mind that on large models, higher output requests will dramatically increase the size of data files. This is left as an exercise for the user.

MSC Random – CRMS Displacement

Within the MSC.Random Form

a. Change the Select Nodes to Node 12
b. Res. Type: Disp. (Displacement)
c. Plot Type: CRMS (Cumulative RMS)
d. Plot Options (Turn off the Large Mass Node)
e. Apply

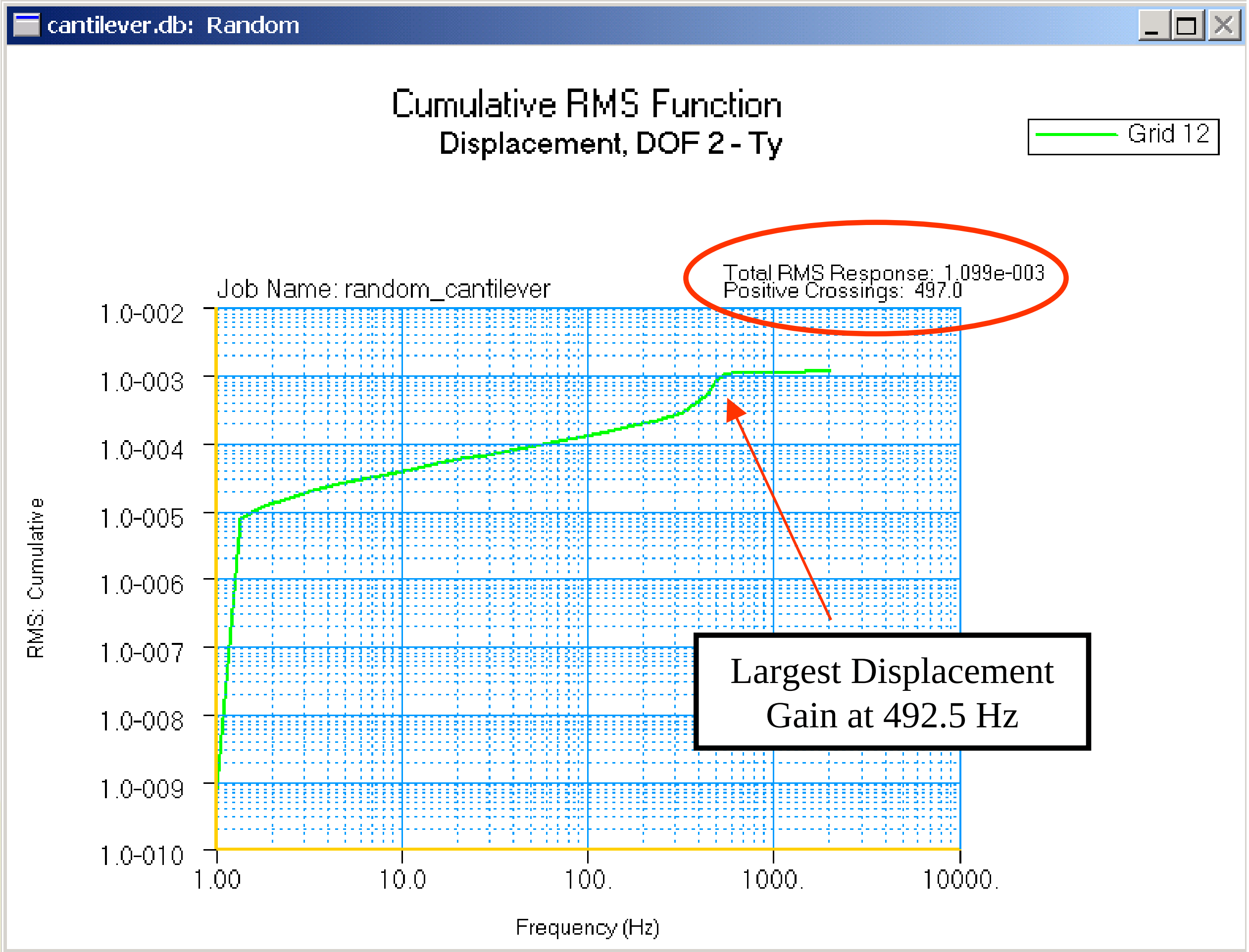
C_{RMS} corresponds to the summation of displacement across the bandwidth. The summation corresponds to the 1σ displacement, while the curve is helpful in determining where the largest influence from all of the modes occurs. You should notice that the largest influence corresponds to the largest acceleration peaks and ultimately to the largest response on the mass participation tables in the modal analysis.



CRMS Displacement

The Total RMS Response (CRMS Response = 0.0011 inch over the bandwidth of 20 – 2000 Hz. This is the 1s displacement of tip of the cantilever beam. Because we are using Node 12, which is part of the explicit MPC between Node 1 (base) and Node 11 (tip), we have removed the large mass displacement of the system.

Node 12 = Node 11 – Node 1



Displacement – Hand Calculated

Using the Displacement calculation for a beam in bending, where P = W = 0.1 lb_f, the 1σ displacement = 0.0011 inch. This corresponds very well with the 0.0011 inch CRMS displacement calculated from MSC.Random.

Miles Equation

$$G_{rms} = \frac{\sqrt{\frac{\pi}{2} \cdot f \cdot PSD \cdot Q}}{g}$$

$$G_{rms} = 27.8$$

Displacement Total
(Bending + Shear)

$$\delta_T = \frac{P \cdot L^3}{3 \cdot E \cdot I} + \frac{P \cdot L}{K_{sf} \cdot A \cdot G}$$

$$\delta_T = 4.032 \times 10^{-5}\,\text{in}$$

Cumulative Displacement

$$\delta_{CRMS} = G_{rms} \cdot \delta_T$$

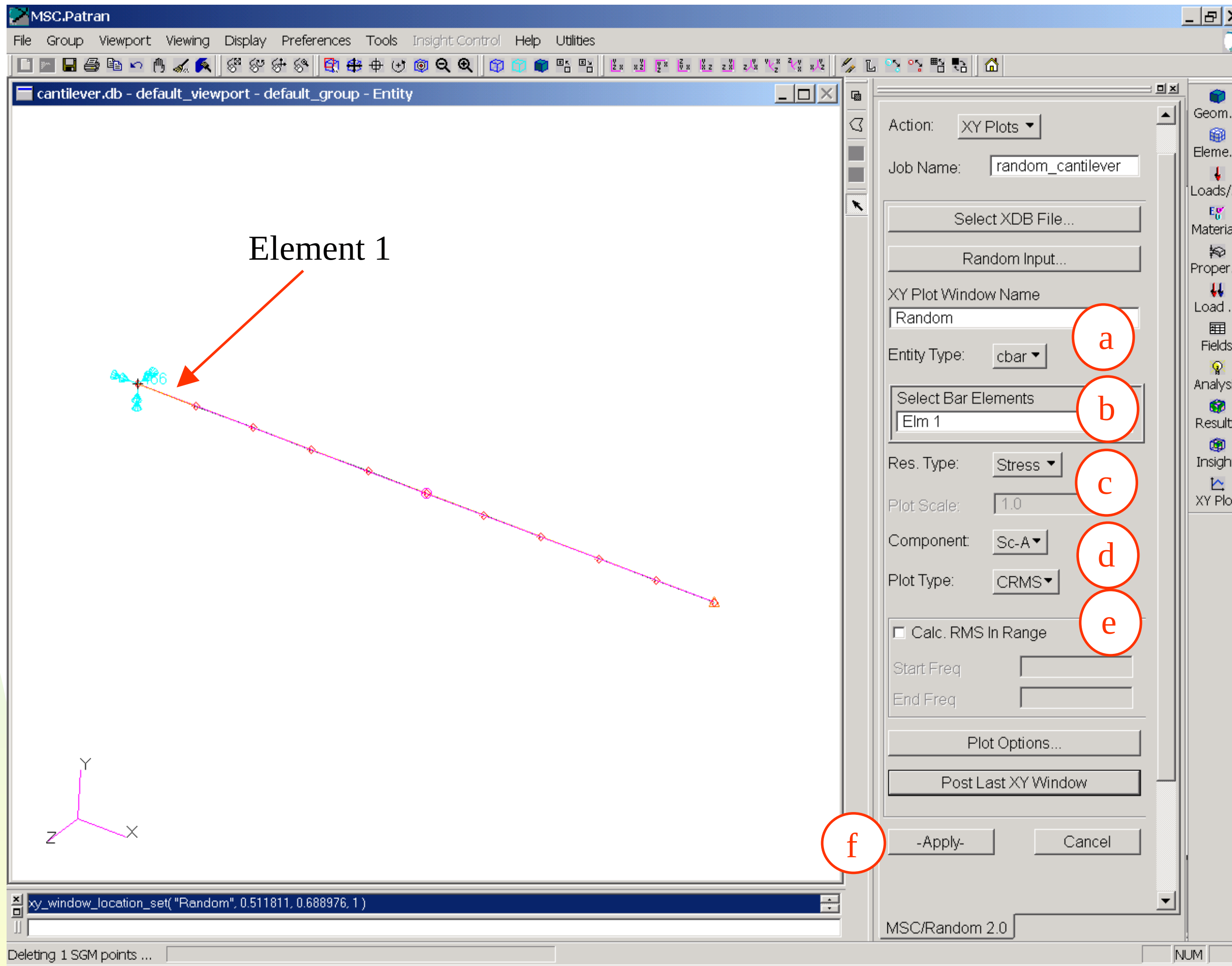
$$\delta_{CRMS} = 0.0011\,\text{in}$$

CRMS Stress

Return to the MSC.Random Form

a. Entity Type: cbar
b. Select Bar Element: Elm 1
c. Res Type: Stress
d. Component: Sc-A
e. CRMS
f. Apply

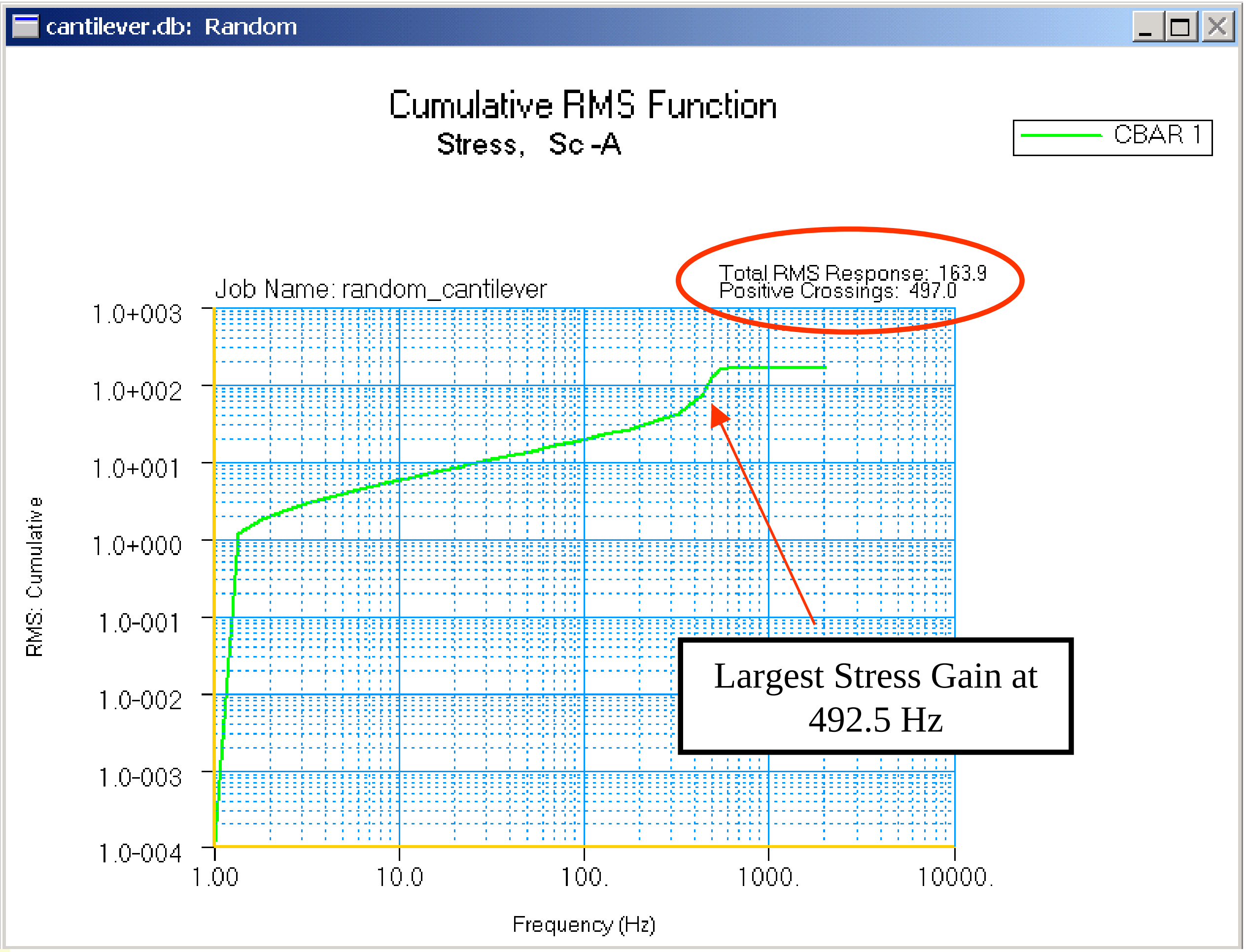
Sc-A = Bending Stress on Beam



CRMS Stress - Results

Cumulative Bending Stress:

1σ Stress = 164 psi



CRMS Stress – Hand Calculated

The Hand-Calculated 1-Sigma Cumulative Bending Stress (167 psi) corresponds with the FEA-Calculated Stress of 164 psi*.

Miles Equation

$$G_{rms} = \frac{\sqrt{\frac{\pi}{2} \cdot f \cdot PSD \cdot Q}}{g}$$

$$G_{rms} = 27.8$$

Moment from Displacement

$$Moment = M \cdot g \cdot L$$

$$Moment = 1\, in \cdot lbf$$

Bending Stress

$$\sigma_b = \frac{Moment \cdot h}{2I}$$

$$\sigma_b = 6\, psi$$

Cummulative Stress

$$\sigma_{bCRMS} = \sigma_b \cdot G_{rms}$$

$$\sigma_{bCRMS} = 167\, psi$$

*The stress is slightly off due to FEA solution being an approximation subjected to the number of points used to describe the response curve. However, in this case, it does not affect the overall accuracy of the solution. By setting the FREQ4 card in Adv. Output Requests in MSC.Random to 5 (default=3), the user will find the solution to be more exact. However, keep in mind that on large models, higher output requests will dramatically increase the size of data files. This is left as an exercise for the user.

Review of Assumptions for Miles Equation

Miles Equation Assumptions

- Single Degree of Freedom System
- Assumes that all energy across the entire bandwidth of interest is the result of one mode (one dominant frequency)
- The input and response are in the same direction (no cross-spectral response with other directions)
- Damping is the same across the entire structure

With Complex Structures (2 or more DOF's), Miles Equation Fails Due to the following:

- A PSD input may have different G^2/Hz values across the entire structure (ie. the curve is not a straight line at one value)
- The complex structure will most likely have more then 1 mode that has significant mass that contributes the response.
- There will most likely be some cross-spectral response (ie. very difficult to isolate specific directions)
- Damping may vary across the structure (especially at joints).

With all that, it is still a very good idea to check your results against Miles Equation, as the majority of response for most structures can be generalized with a single mode. If you squeeze the bandwidth of interest around that mode, then Miles Equation becomes more accurate.