Programma preventivo del Corso integrativo di Elementi finiti (Meccanica delle Vibrazioni AA2012-2013)

-Formulazione dell'equazione del moto mediante il principio di Hamilton.

-**Il metodo di Rayleigh-Ritz**: energia potenziale e cinetica, caratteristiche della funzione di soluzione; accuratezza della soluzione. Esercizio: vibrazioni flessionali di una trave a mensola (prime due frequenze naturali).

-**Vibrazioni flessionali libere della trave mediante il metodo ad elementi finiti:** metodologia di valutazione delle matrici massa e rigidezza, funzione di forma, assemblaggio. Applicazione del metodo per la valutazione delle frequenze naturali di una trave incastrata. Accuratezza della soluzione. Fattori che influenzano l'accuratezza del FEM, Tecniche di riduzione del numero di gradi di libertà, matrice massa "lumped" e "consistent"

-**Software per il calcolo ad elementi finiti: MSC. Nastran e MSC. Patran**. Lettura del file BDF mediante i comandi GRID, MAT1, EIGRL, SOL, PBEAM, CBEAM, CTETRA, SPC1, tipi di analisi dinamiche. Esercitazioni in laboratorio.

-Correlazione numerico-sperimentale

Esercizi da portare all'esame:

ESERCIZIO 1 - Vibrazioni flessionali di una trave a mensola (prime due frequenze naturali) mediante il metodo di Rayleigh-Ritz.
ESERCIZIO 2 – Modello ad elementi finiti di una Trave con Matlab

ESERCIZIO 3 - Trave incastrata in MSC. Nastran-Patran

ESERCIZIO 4 – Porta di automobile in MSC. Nastran-Patran

RIFERIMENTI

G. Diana, F. Cheli, Dinamica e vibrazione dei sistemi, 1993.
C. De Silva, Computer techniques in Vibration, 2007.
S. Rao, Mechanical Vibrations, 1995.
M. Petyt, Introduction to finite element vibration analysis, 1990.
C. Gianini, La progettazione strutturale con il calcolatore, 2003.
L. Meirovitch, Fundamentals of vibrations, 2001.
W. Heylen, S. Lammens, P. Sas, Modal analysis theory and testing, 2004

Tecniche numeriche nell'analisi vibratoria

1. Introduzione [Diana-Cheli,Petyt, Rao,De Silva]

I sistemi reali sono rappresentabili come sistemi continui ad infiniti grafi di libertà (g.d.l.). Utilizzare la teoria dei continui significa avere a che fare con complesse equazioni differenziali alle derivate parziali già per sistemi molto semplici (vibrazioni assiali, torsionali, flessionali della trave). Inoltre i sistemi reali sono spesso di forma complessa, formati da diversi materiali, sottoposti a complesse storie di carico (ad esempio veicoli, organi di macchine, ecc); in queste condizioni è

impossibile ottenere analiticamente le equazioni del moto che soddisfino le condizioni al contorno a cui il sistema è realmente soggetto. Sono state sviluppate pertanto una serie di tecniche approssimate che permettono di risolvere questo problema.
In genere i sistemi continui ad infiniti gradi di libertà vengono discretizzati in un modello approssimato discreto a N gradi di libertà che ne approssimi in maniera sufficientemente adeguata il comportamento. Esistono varie tecniche di discretizzazione, alcune delle quali sono:

- la schematizzazione a parametri concentrati (teoria dei sistemi a 1g.d.l. e N g.d.l.);
- il metodo ad elementi finiti (Finite Element Method);
- il metodo ai contorni finiti (Boundary Element Method);
- il metodo Multibody

Per sistemi a parametri concentrati a N g.d.l. è possibile ottenere frequenze naturali e forme modali in forma esatta uguagliando il determinante dell'equazione caratteristica a zero e risolvendo il sistema di equazioni accoppiate così ottenuto. Per valori elevati di N, la soluzione del sistema può essere lunga; in questi casi è possibile ricorrere al metodo modale per ottenere un sistema dello stesso ordine di grandezze, ma disaccoppiato, oppure utilizzare metodi numeri o analitici per ottenere le frequenze naturali e/o le forme modali. Alcuni di questi metodi sono: la formula di Dunkerley, il quoziente di Rayleigh, il metodo di Holzer, metodi di iterazione matriciale, metodo di Jacobi. In particolare, la formula di Dunkerley permette di stimare il quadrato della prima pulsazione naturale ω_1^2:

$$\frac{1}{\omega_1^2} \cong \sum_{i=1}^{N} \delta_{ii} \cdot m_i$$

dove δ_{ii} e m_i sono rispettivamente i termini della matrice cedevolezza e massa sulla diagonale principale.
Il quoziente di Rayleigh permette anch'esso di stimare il quadrato della pulsazione naturale ω_j^2:

$$\omega_j^2 = \frac{\{\Phi\}_j^T [K] \{\Phi\}_j}{\{\Phi\}_j^T [M] \{\Phi\}_j}$$

dove $\{\Phi\}_j$ rappresenta la deformata del j-esimo modo di vibrare, mentre $[K]$ ed $[M]$ sono le matrici rigidezza e massa, rispettivamente. In genere il quoziente di Rayleigh viene utilizzato per stimare la prima frequenza naturale. Infatti la deformata del primo modo ha in genere una forma semplice e quindi è facilmente ottenibile, inoltre in genere il comportamento dinamico delle strutture è prevalentemente governato dai primi modi di vibrare, di fondamentale importanza è quindi la stima della prima frequenza naturale. E' da notare che la stima della frequenza naturale è tanto più precisa quanto la deformata del primo modo ipotizzato è vicina alla realtà.

Esistono ancora metodi approssimati numerici che permettono di calcolare la soluzione completa per sistemi a N g.d.l. o per sistemi continui e non solo alcune frequenze naturali e/o modi di vibrare. Tutti i metodi di integrazione numerica delle equazioni differenziali del moto hanno due caratteristiche comuni: a) non soddisfano le equazioni del moto ad ogni istante *t*, ma solo a intervalli di tempo discreti (Δt), b) assumono un possibile tipo di variazione per spostamento, velocità e accelerazione, in genere utilizzando la scomposizione in serie di Taylor (Metodo alle differenze finite, metodo di Runge-Kutta, Metodo di Houbolt, Metodo di Wilson, Metodo di Newmark,) oppure ipotizzando arbitrariamente possibili deformate (metodo di Ritz-Rayleigh, metodo ad elementi finiti).

2. Formulazione delle equazioni del moto mediante approccio energetico (principio di Hamilton)

Il primo passo per analizzare il comportamento vibratorio di un sistema meccanico è quello di scrivere correttamente le equazioni del moto. A tal proposito esistono metodi più o meno automatizzati per farlo:

- il metodo basato sulla legge di Newton
- il metodo basato sul principio dei lavori virtuali
- il metodo basato sul principio di D'Alembert
- il metodo delle cedevolezze
- il metodo utilizzato dalle tecniche Multibody
- il metodo dei coefficienti di influenza
- il metodo basato sul principio di Hamilton

Di seguito se ne descriverà uno che sta alla base del metodo ad elementi finiti: un metodo basato su un approccio energetico e che utilizza il principio di Hamilton. Il principio di Hamilton afferma che "l'integrale fra t_1 e t_2 della variazione di energia cinetica δT e del lavoro δW svolto dalle forze interne ed esterne è uguale a zero."

$$\int_{t_1}^{t_2} (\delta T + \delta W) dt = 0 \tag{1}$$

dove δW è la somma delle variazioni dei lavori dovuti alle forze conservative e non conservative:

$$\delta W = \delta W_C + \delta W_{NC} \tag{2}$$

In particolare, δW_{NC} è dovuto alle forze dissipative (e.g. smorzamento) o alle forze che portano energia nel sistema (e.g. forze esterne), mentre δW_C è definito come l'inverso della variazione di energia potenziale elastica:

$$\delta W_C = -\delta U = -(U(t_2) - U(t_1)) \tag{3}$$

Sostituendo la (2)(3) nella (1):

$$\int_{t_1}^{t_2} (\delta T + \delta W_C + \delta W_{NC}) dt = 0 \rightarrow \int_{t_1}^{t_2} \left[\delta (T - U) + \delta W_{NC} \right] dt = 0 \tag{4}$$

Il vantaggio di questa formulazione è che si usano termini energetici e pertanto scalari e non vettoriali che danno inutili complicanze di segno.

Ad esempio, per il sistema ad 1 g.d.l. di Figura 1, l'energia cinetica T, l'energia potenziale elastica U e δW_{NC} sono:

$$T = \frac{1}{2} m \dot{u}^2 ;\ U = \frac{1}{2} k u^2 ;\ \delta W_{NC} = f_e \delta u - c \dot{u} \delta u \tag{5}$$

Sostituendo le espressioni in (5) nella (4) si ottiene l'equazione del moto nella forma standard:

$m\ddot{u} + c\dot{u} + ku = f_e$.

I passaggi matematici sono forniti in Petyt pp. 9-10.

Il principio di Hamilton (4)applicato ad un sistema discreto a *N* g.d.l. può essere scritto mediante le equazioni di Lagrange (alcuni passaggi matematici sono mostrati in Petyt pag. 10):

$$\begin{cases} \dfrac{d}{dt}\left(\dfrac{\partial T}{\partial \dot{q}_1}\right) + \left(\dfrac{\partial D}{\partial \dot{q}_1}\right) + \left(\dfrac{\partial U}{\partial q_1}\right) = Q_1 \\ \dots \\ \dfrac{d}{dt}\left(\dfrac{\partial T}{\partial \dot{q}_N}\right) + \left(\dfrac{\partial D}{\partial \dot{q}_N}\right) + \left(\dfrac{\partial U}{\partial q_N}\right) = Q_N \end{cases} \tag{6}$$

dove:

- $q_1 \dots q_N$ coordinate generalizzate indipendenti del sistema a *N* g.d.l.;

- $T = T(\dot{q}_1 \ldots \dot{q}_N)$ energia cinetica funzione delle velocità $\dot{q}_j$;
- $U = U(q_1 \ldots q_N)$ energia potenziale elastica funzione degli spostamenti q_j;
- $\delta W_{NC} = \sum_{j=1}^{N} \left(Q_j - \frac{\partial D}{\partial \dot{q}_j} \right) \delta q_j$ variazione del lavoro svolto dalle forze non conservative;
- $D = D(\dot{q}_1 \ldots \dot{q}_N)$ funzione dissipazione che dipende delle velocità $\dot{q}_j$;
-

In generale, T,D,U possono essere calcolate sistematicamente usando le seguenti espressioni:

$$T = \frac{1}{2} \{\dot{q}\}^T [M] \{\dot{q}\} \tag{7}$$

$$D = \frac{1}{2} \{\dot{q}\}^T [C] \{\dot{q}\} \tag{8}$$

$$U = \frac{1}{2} \{q\}^T [K] \{q\} \tag{9}$$

dove :

$[M],[C],[K]$ sono le matrici massa, smorzamento e rigidezza del sistema N g.d.l., mentre il vettore colonna $\{q\}$ è il vettore delle coordinate generalizzate:

$$\{q\} = \begin{Bmatrix} q_1 \\ \vdots \\ q_N \end{Bmatrix}$$

E' da osservare che nelle equazioni di Lagrange (6), il primo addendo è il termine di energia cinetica che dimostro essere uguale a $[M]\{\ddot{q}\}$:

$$\begin{Bmatrix} \frac{d}{dt}\left(\frac{\partial T}{\partial \dot{q}_1}\right) \\ \cdots \\ \frac{d}{dt}\left(\frac{\partial T}{\partial \dot{q}_N}\right) \end{Bmatrix} = [M]\{\ddot{q}\} \tag{10}$$

infatti, per un sistema ad 1 .g.d.l., l'energia cinetica T è definita in (5) e svolgendo i calcoli:

$$\frac{d}{dt}\left(\frac{\partial T}{\partial \dot{u}}\right) = m\ddot{u}$$

In analogia si dimostra che il secondo termine nelle equazioni di Lagrange è il termine dissipativo e il terzo termine è il termine legato alla matrice rigidezza:

$$\begin{Bmatrix} \left(\frac{\partial D}{\partial \dot{q}_1}\right) \\ \cdots \\ \left(\frac{\partial D}{\partial \dot{q}_N}\right) \end{Bmatrix} = [C]\{\dot{q}\} \tag{11}$$

$$\begin{Bmatrix} \left(\frac{\partial U}{\partial q_1}\right) \\ \cdots \\ \left(\frac{\partial U}{\partial q_N}\right) \end{Bmatrix} = [K]\{q\} \tag{12}$$

Pertanto le equazioni di Lagrange (6) possono essere ricondotte all'equazione del moto in forma standard per sistemi a N g.d.l.:

$$[M]\{\ddot{q}\} + [C]\{\dot{q}\} + [K]\{q\} = \{Q\} \tag{13}$$

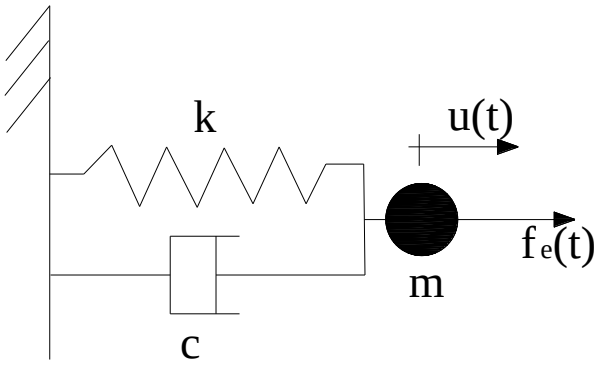


Figura 1. Sistema ad 1 g.d.l.

3. Il metodo di Ritz-Rayleigh

Per sistemi reali la soluzione dell'equazione di Hamilton è complicata, e la soluzione in forma chiusa può non esistere. Pertanto alcuni metodi approssimati sono nati. Uno di questi è il metodo di Ritz-Rayleigh (R.R.) che approssima la soluzione con una espansione della forma

$$v(x,t)=\sum_{k=1}^{m} d_k(x) q_k(t) = \{d(x)\}^T \{q(t)\} = \begin{Bmatrix} d_1(x) \\ \dots \\ d_k(x) \\ \dots \\ d_m(x) \end{Bmatrix} \{q_1(t) \quad \dots \quad q_k(t) \quad \dots \quad q_m(t)\} \tag{14}$$

Dove $q_k(t)$ sono le funzioni incognite dipendenti dal tempo che saranno ricavate risolvendo un problema agli auto vettori-autovalori, mentre $d_k(x)$ sono le funzioni prescritte conosciute a priori e rappresentano una possibile deformata del sistema (funzioni di forma di R.R.), e *m* è l'ordine considerato, cioè il numero di termini presi in considerazione nella sommatoria.

Il metodo di R.R. per lo studio di vibrazioni libere (forze esterne nulle) e per sistemi non smorzati prevede:

1. scegliere una funzione (che chiamo *v(x,t)*)che sia soluzione approssimata dell' equazione di Hamilton (4) e che soddisfi le condizioni al contorno del problema;
2. Calcolare l' energia potenziale elastica;
3. Calcolare l' energia cinetica;
4. utilizzare le equazioni di Lagrange (6) per ricavare l'equazione del moto;
5. risolvere un problema agli autovalori-autovettori. In particolare risolvendo l'equazione caratteristica $\left([K]-\omega_j^2[M]\right)\{\Phi\}_j=\{0\}$) si ottengono le incognite $\{\Phi\}_j$ (autovettori) e le pulsazioni naturali ω_j (autovalori) (j=1…m). Gli autovettori permettono di ricavare le funzioni incognite $\{q(t)\}$ della (14) mediante l'espressione:

   $\begin{Bmatrix} q_1(t) \\ \dots \\ q_k(t) \\ \dots \\ q_m(t) \end{Bmatrix} = \{\Phi\}_j \sin\omega_j t = \begin{Bmatrix} \Phi_{1j} \\ \dots \\ \Phi_{kj} \\ \dots \\ \Phi_{mj} \end{Bmatrix} \sin\omega_j t$. Le funzioni incognite sono definite alla pulsazione naturale ω_j. Le funzioni incognite sono tante quante sono le pulsazioni naturali ω_j. Allora la forma del modo di vibrare si ottiene con la (14):

   $$v(x)=\{d(x)\}^T \{\Phi\}_j \tag{15}$$

   Il moto libero alla pulsazione naturale ω_j è pertanto:

   $$v(x,t)=\{d(x)\}^T \{q(t)\} = \{d(x)\}^T \{\Phi\}_j \sin\omega_j t$$

Nel seguito studieremo solo le vibrazioni flessionali della trave (Figura 2) pertanto le espressioni da usare per T e U saranno:

$$U=\frac{1}{2}\int_0^L EI_z \left(\frac{\partial^2 v(x,t)}{\partial x^2}\right)^2 dx \tag{16}$$

$$T = \frac{1}{2}\int_0^L \rho A\left(\frac{\partial v(x,t)}{\partial t}\right)^2 dx \qquad (17)$$

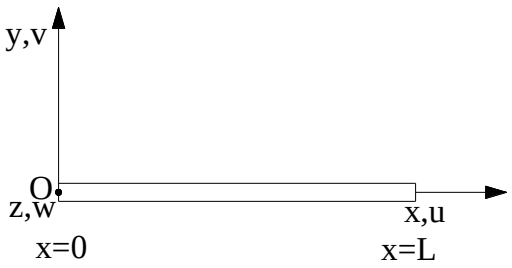


Figura 2. Sistema di riferimento per la trave soggetta a flessione.

La funzione *v(x,t)* è particolare. E' una espansione formata da due funzioni, una dipendente solo dal tempo e una solo dallo spazio. Le funzioni di forma $d_k(x)$ sono scelte arbitrariamente, tuttavia devono soddisfare le seguenti specifiche:

1. devono essere linearmente indipendenti;
2. devono essere *p* volte differenziabili, dove *p* è l'ordine maggiore che compare nell'espressione dell'energia potenziale elastica (p=2 nel nostro caso);
3. devono soddisfare le condizioni al contorno;
4. devono formare una serie completa, cioè l'errore medio quadratico deve essere al limite nullo:

$$\lim_{m\to\infty}\int_0^L\left(v(x,t)-\sum_{k=1}^{m}d_k(x)q_k(t)\right)^2dx=0$$

In genere le funzioni di forma $d_k(x)$ sono:

1. funzioni polinomiali: $d(x)=\alpha_1+\alpha_2x+\alpha_3x^2+\ldots;$
2. funzioni trigonometriche: $d(x)=\alpha_1e^{i\beta_1x}+\alpha_2e^{i\beta_2x}+\ldots;$
3. polinomi di Legendre, di Jacobi, etc;

ESEMPIO 1. Vibrazioni flessionali di una trave a mensola. Si determinino le prime due frequenze naturali una volta noti modulo di Young (E), densità (ρ), area e momento di inerzia della sezione (A, I_z) e lunghezza (l).

Sia $\{d(x)\}=\begin{Bmatrix}d_1(x)\\d_2(x)\end{Bmatrix}=\begin{Bmatrix}\left(\frac{x}{l}\right)^2\\\left(\frac{x}{l}\right)^3\end{Bmatrix}$ la funzione di forma prescritta e pertanto la deformata diventa:

$$v(x,t)=\sum_{k=1}^{m}\{d(x)\}^T\{q(t)\}=\{d_1(x)\quad d_2(x)\}\begin{Bmatrix}q_1(t)\\q_2(t)\end{Bmatrix}=\begin{Bmatrix}\left(\frac{x}{l}\right)^2&\left(\frac{x}{l}\right)^3\end{Bmatrix}\begin{Bmatrix}q_1(t)\\q_2(t)\end{Bmatrix}=\left(\frac{x}{l}\right)^2q_1(t)+\left(\frac{x}{l}\right)^3q_2(t)$$

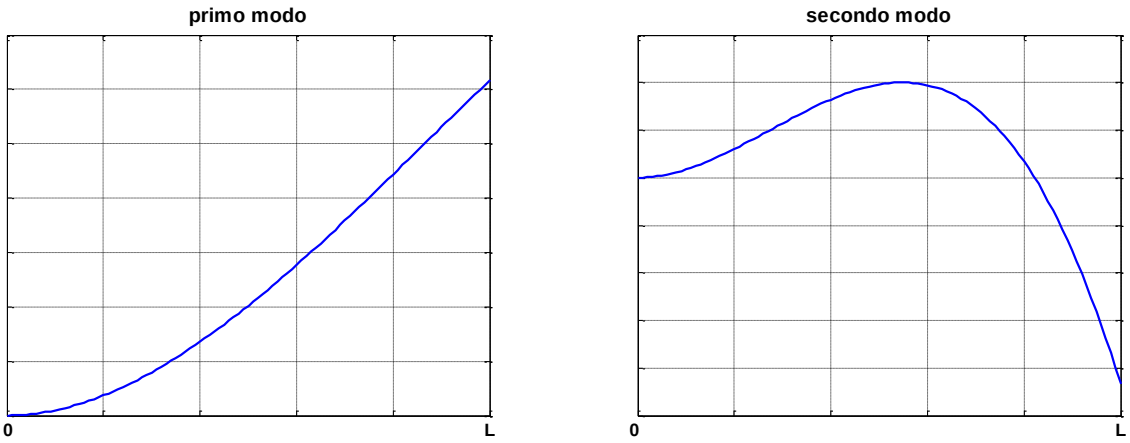
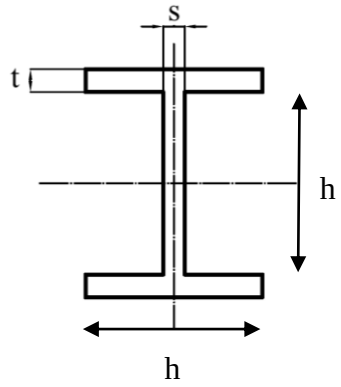


Figura 3. Primo e seconda forma modale ottenute con la funzione di R.R a due termini.

Pertanto aumentando il numero di termini considerati nell'espansione di R.R., aumenta la precisione con cui le pulsazioni naturali vengono stimate e aumenta il numero delle pulsazioni naturali che possono essere stimate.

Esercizio 1 (da portare in forma scritta all'esame). Vibrazioni flessionali di una trave a mensola in acciaio. Si determinino le prime due frequenze naturali e le relative forme modali (con prima componente di ogni modo uguale ad 1) della trave di lunghezza 3 metri e dimensioni della sezione h=4cm; t=3 mm; s=3mm.

Sia $\{d(x)\} = \begin{Bmatrix} d_1(x) \\ d_2(x) \end{Bmatrix} = \begin{Bmatrix} \left(4\frac{x}{L}\right)^2 \\ \left(2\frac{x}{L}\right)^3 \end{Bmatrix}$



Si risolva l'esercizio per iscritto ricavando i modi e le frequenze naturali per via analitica e successivamente risolvere l'esercizio in ambiente Matlab mostrando il grafico delle deformate modali

Traccia di soluzione in ambiente matlab

1) Definizione della matrice M e K

```
M = […….;
….
```

2) Calcolo di frequenze e modi

```
[fi,omegaq] = eig(K,M);
for i =1:2,
    finorm(:,i) = fi(:,i)./fi(1,i);
end;
omega=sqrt(omegaq)
%%%modi
x=0:0.01:L;
v1=4*(x/L).^2+finorm(2,1).*2*(x/L).^3;
v2=4*(x/L).^2+finorm(2,2).*2*(x/L).^3;
```

3) Grafico delle deformate modali

```
figure
plot(x,v1)
```

4. Il metodo ad elementi finiti

Il metodo di R.R. è essenzialmente una tecnica di discretizzazione per derivare soluzioni approssimate dell'equazione del moto del sistema quando lo spostamento *v(x,t)* è ottenuto come combinazione lineare di funzioni prescritte $\{d(x)\}$ moltiplicate per le funzioni incognite $\{q(t)\}$. Queste ultime sono ottenute risolvendo un problema agli autovalori. E' però necessario chiedersi quanto la soluzione del problema agli autovalori (*v(x,t)*) approssimi correttamente l'equazione

differenziale del moto del sistema continuo. Il grado di correttezza del metodo di R.R. risiede principalmente nella bontà delle funzioni di forma prescritte e nel loro numero, come visto nell'esempio precedente. Tuttavia per sistemi complessi non è semplice definire a priori una possibile funzione di forma sull'intero dominio. Infatti, nel metodo di R.R. la funzione di forma deve essere definita sull'intero dominio della struttura (nell'esempio precedente fra 0 ed L). Il metodo ad elementi finiti permette di superare questo problema, infatti le funzioni di forma sono definite in piccoli sottodomini del sistema completo, chiamati elementi finiti. L'insieme degli elementi finiti si chiama mesh. In genere queste funzioni sono polinomi di basso ordine e sono le stesse per ogni elemento finito. La metodologia seguita dai metodi ad elementi finiti è la seguente:

1. dividere la struttura in un numero di elementi di dimensione finita. Gli elementi sono uniti l'una all'altro mediante nodi;
2. associare ad ogni nodo un dato numero di g.d.l.;
3. costruire un set di funzioni (funzioni di forma) in modo tale che ognuna abbia valore unitario in un grado di libertà e zero in tutti gli altri;
4. sostituire le funzioni di forma di un elemento nell'espressione dell'energia cinetica e dell'energia potenziale per ottenere le matrici massa e rigidezza di ogni elemento finito
5. sommare le energie cinetiche e potenziali di elemento per ottenere le energie del sistema completo (assemblaggio delle matrici massa e rigidezza)
6. imporre le condizioni al contorno;
7. risolvere il problema (ad esempio un problema agli autovettori-autovalori per ottenere modi e frequenze naturali).

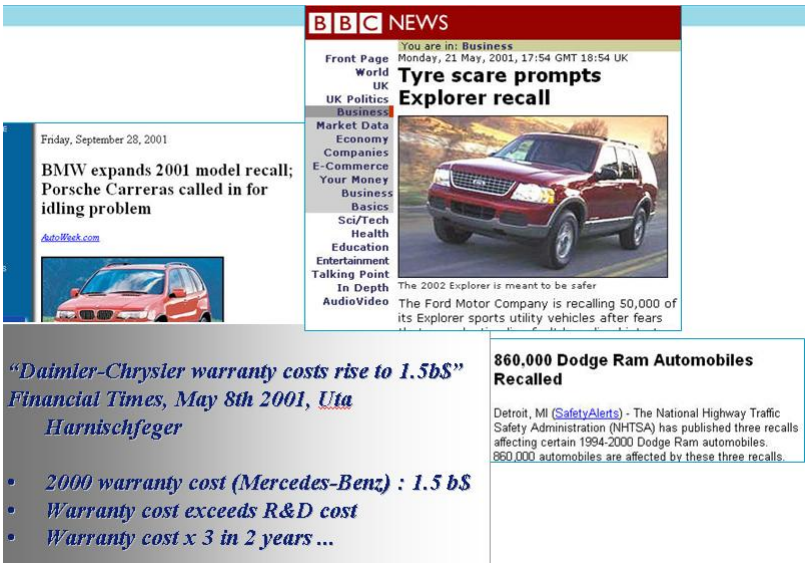


Figura 4. Importanza del CAE nella progettazione.

- Origin of the Finite Element Method :
  - driven by aircraft industry
  - in the early fifties
  - to describe and analyze complex geometry
- Basic idea :
  - break the geometry down in a set small simple 'elements'
  - approximate exact solution at element level
  - connect the elements together again and solve the assembled system

Deterministic method

Figura 5. Origine del FEM

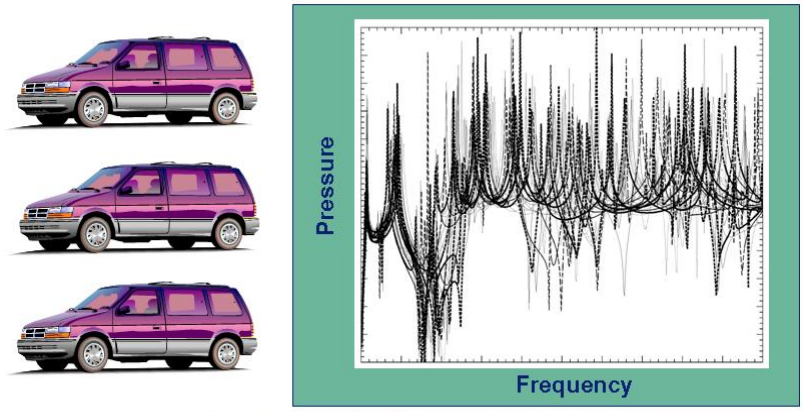
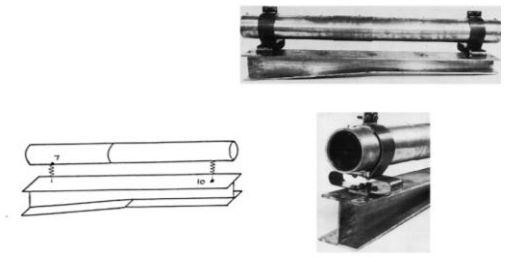


Figura 6. Variabilità delle strutture reali



Time Spent on Analysis

Participant	I	II	III	IV	V	VI	VII	VIII	IX	X	XI	XII
Man hours	50	80	75	300	200	100	25	200	80	110	NS	38

Natural frequencies (Hz)

Participant	S1	S2	S3	B1	B2	B3	C1	C2	C3	NOM
I	54	74	—	503	1464	2552	630	1670	—	12
II	45	61	—	448	1130	2110	518	1380	—	42
III	54	73	67	512	1380	2385	561	1553	2503	12
IV	57	35	80	434	1080	1645	544	1436	—	22
V	59	—	188	644	1608	2313	566	1565	2464	14
VI	54	74	84	440	1125	1920	530	1355	2384	15
VII	54	73	—	370	908	1580	553	1126	—	11
VIII	55	75	—	570	1574	2573	617	1588	—	8
IX	55	75	96	424	1052	1571	534	1435	—	24
X	55	75	110	428	1048	1559	545	1478	—	23
XI	55	75	123	445	1137	1935	537	1365	2379	18
XII	38	51	74	461	1140	1708	499	1358	2389	27
Min	38	35	67	370	908	1571	499	1126	2379	8
Max	59	75	188	644	1608	2573	630	1670	2503	42
Mean	54	68	103	473	1220	1998	553	1442	2420	19
Measured	59	80	118	426	1072	2002	532	1470		

Figura 7. Gara fra prestigiose università negli anni ottanta.

Alcuni siti interessanti
http://www.youtube.com/watch?v=U9swU5J3gLI&NR=1
http://www.youtube.com/watch?v=L3x5iq3oT9U&feature=related
http://www.youtube.com/watch?v=_NHqXWohW2g&feature=related
http://www.youtube.com/watch?v=0NG9v3JdfEs&feature=related

4.1 Trattazione teorica per la trave (vedi Petyt,)

Nel seguito si mostrerà il metodo agli elementi finiti nel caso di vibrazioni flessionali libere di una trave (vibrazioni non forzate). Si descriverà in dettaglio la metodologia descritta nel paragrafo precedente.

1. **dividere la struttura in un numero di elementi di dimensione finita. Gli elementi sono uniti l'una all'altro mediante nodi.**

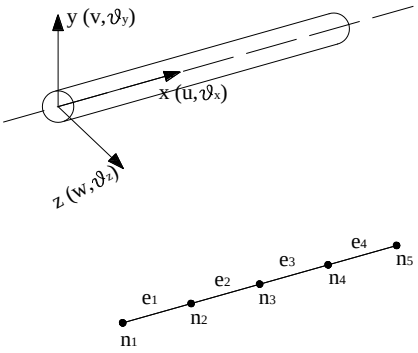


Figura 8. Discretizzazione in 4 elementi di una trave a sezione circolare uniforme.

2. **associare ad ogni nodo un dato numero di g.d.l..**

Per lo studio delle vibrazioni flessionali della trave si è associato ad ogni nodo uno spostamento lungo *y* (*v*) e una rotazione attorno all'asse z (θ_z).

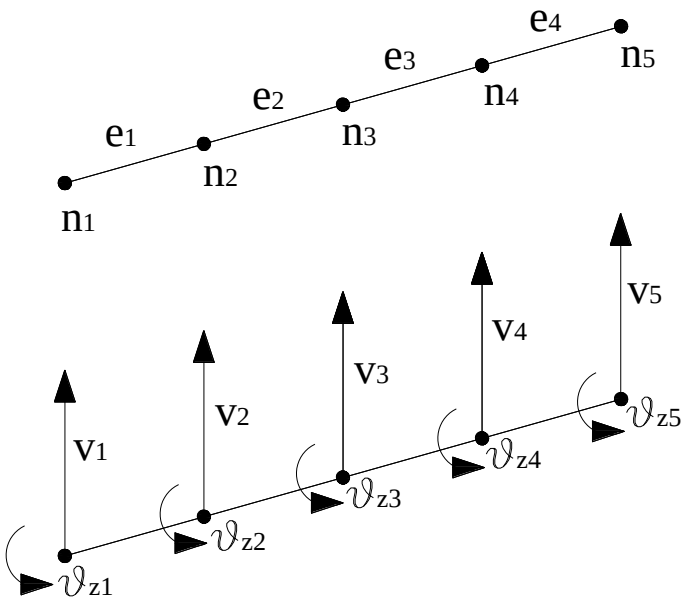


Figura 9. Gradi di libertà flessionali e rotazioni dei 5 nodi.

3. **Costruire un set di funzioni (funzioni di forma) in modo tale che ognuna abbia valore unitario in un grado di libera e zero in tutti gli altri (Figura 10).**

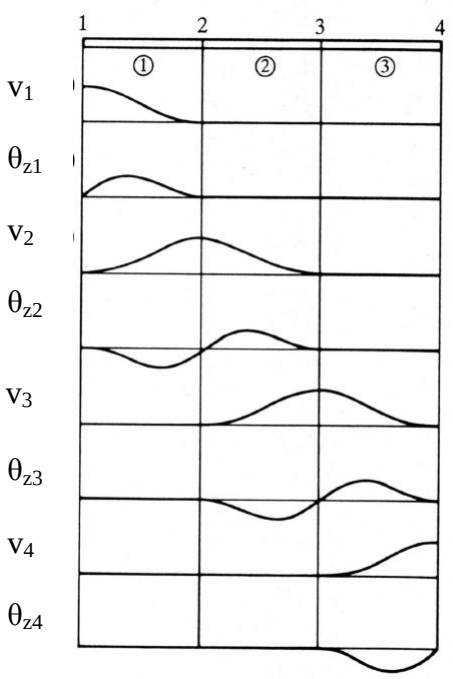


Figura 10. Funzioni di forma di elemento per vibrazioni flessionali della trave.

Le funzioni di forma in analogia con quanto mostrato per il metodo di R.R. devono soddisfare le seguenti condizioni:

- Essere linearmente indipendenti;
- Essere funzioni continue e *p* volte differenziabili all'interno dell'elemento, dove *p* è il massimo ordine di derivazione che appare nell'espressione dell'energia potenziale;
- Se le funzioni di forma sono polinomi, devono essere polinomi complete di ordine almeno *p* oppure se hanno ordine superiore possono essere anche incompleti;
- Soddisfare le condizioni al contorno;

Per lo studio delle vibrazioni flessionali della trave si è scelto un polinomio completo di ordine 3, visto che il massimo ordine di derivazione che appare nella (16) è uguale a 2.

Considero ora un elemento finito trave di densità ρ, modulo di Young E, inerzia e area della sezione I_z e A (Figura 11) in cui gli effetti di taglio e l'inerzia rotazionale sono trascurati.

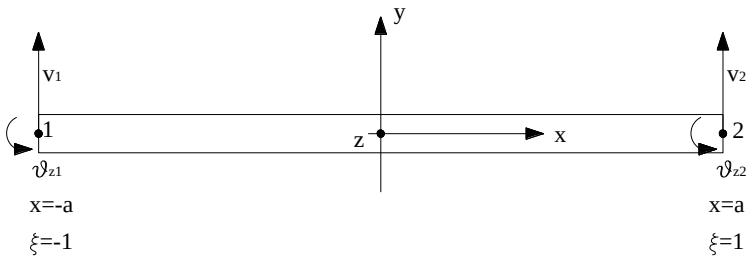


Figura 11. Notazione per lo studio delle vibrazioni flessionali di un elemento trave.

Il polinomio del terzo ordine che descrive la deformata flessionale dell'elemento finito scritto in funzione della coordinata adimensionale $\xi = x/a$ è:

$$v(\xi,t)=\alpha_1+\alpha_2\xi+\alpha_3\xi^2+\alpha_4\xi^3 \quad (18)$$

Che può essere riscritto in forma matriciale:

$$v(\xi,t)=\begin{Bmatrix}1 & \xi & \xi^2 & \xi^3\end{Bmatrix}\begin{Bmatrix}\alpha_1\\ \alpha_2\\ \alpha_3\\ \alpha_4\end{Bmatrix}=\{p(\xi)\}\{\alpha(t)\} \quad (19)$$

Per ottenere un'espressione simile alla (18), ma riferita al grado di libertà rotazionale, è necessario ricordare che $\theta_z=\dfrac{\partial v}{\partial x}$ e differenziare la (18):

$$\theta_z(\xi,t)=\frac{\partial v}{\partial x}=\frac{\partial v}{\partial \xi}\frac{d\xi}{dx}=\left(\alpha_2+2\alpha_3\xi+3\alpha_4\xi^2\right)\frac{1}{a} \quad (20)$$

E moltiplicando da ambo i membri per *a*:

$$a\theta_z(\xi,t)=\left(\alpha_2+2\alpha_3\xi+3\alpha_4\xi^2\right)=\begin{Bmatrix}0 & 1 & 2\xi & 3\xi^2\end{Bmatrix}\begin{Bmatrix}\alpha_1\\ \alpha_2\\ \alpha_3\\ \alpha_4\end{Bmatrix} \quad (21)$$

La (19) e (21) rappresentano la deformata flessionale e rotazionale dell'elemento finito. Valutando queste espressioni agli estremi ($\xi=\pm1$)si ottiene:

$$\begin{Bmatrix}v_1\\ a\theta_{z1}\\ v_2\\ a\theta_{z2}\end{Bmatrix}=\begin{bmatrix}1 & -1 & 1 & -1\\ 0 & 1 & -2 & 3\\ 1 & 1 & 1 & 1\\ 0 & 1 & 2 & 3\end{bmatrix}\begin{Bmatrix}\alpha_1\\ \alpha_2\\ \alpha_3\\ \alpha_4\end{Bmatrix} \quad (22)$$

In forma compatta la (22) può essere scritta nella forma:

$$\{\bar{v}_e\}=[A_e]\{\alpha(t)\} \quad (23)$$

Risolvendo per $\{\alpha(t)\}$ si ottiene:

$$\{\alpha(t)\}=[A_e]^{-1}\{\bar{v}_e\} \quad (24)$$

dove $[A_e]^{-1}=\dfrac{1}{4}\begin{bmatrix}2 & 1 & 2 & -1\\ -3 & -1 & 3 & -1\\ 0 & -1 & 0 & 1\\ 1 & 1 & -1 & 1\end{bmatrix}$ (25)

L'equazione (24) può essere scritta in una forma alternativa:

$$\{\alpha(t)\} = [C_e]\{v_e\} \quad (26)$$

dove $[C_e] = \frac{1}{4}\begin{bmatrix} 2 & a & 2 & -a \\ -3 & -a & 3 & -a \\ 0 & -a & 0 & a \\ 1 & a & -1 & a \end{bmatrix}$ e $\{v_e\} = \begin{Bmatrix} v_1 \\ \theta_{z1} \\ v_2 \\ \theta_{z2} \end{Bmatrix}$

Sostituendo la (26) nella (19) si ottiene:

$$v(\xi,t) = \{p(\xi)\}\{\alpha(t)\} = \{p(\xi)\}[C_e]\{v_e\} = \{n(\xi)\}\{v_e(t)\} \quad (27)$$

Dove

$$\{n(\xi)\} = \{N_1(\xi) \quad aN_2(\xi) \quad N_3(\xi) \quad aN_4(\xi)\} \quad (28)$$

Con:

$$N_1(\xi) = \frac{1}{4}(2-3\xi+\xi^3);\ N_2(\xi) = \frac{1}{4}(1-\xi-\xi^2+\xi^3);\ N_3(\xi) = \frac{1}{4}(2+3\xi-\xi^3);\ N_4(\xi) = \frac{1}{4}(-1-\xi+\xi^2+\xi^3)$$

Pertanto considerando la (27), le $\{n(\xi)\}$ forniscono il valore di flessione o rotazione in corrispondenza di ogni g.d.l. del sistema. La differenza fra la (27) e la (18) è che la (27) è specificata per i g.d.l. dell'elemento finito.

4. Sostituire le funzioni di forma di un elemento nell'espressione dell'energia cinetica e dell'energia potenziale per ottenere le matrici massa e rigidezza di ogni elemento finito.

In base alla (17) l'energia cinetica di elemento è definita come:

$$T_e = \frac{1}{2}\int_{-a}^{a} \rho A\dot{v}^2(x,t)dx = \frac{1}{2}\int_{-1}^{1} a\rho A\dot{v}^2(\xi,t)d\xi \quad (29)$$

Una volta calcolato $\dot{v}^2(\xi,t) = \{\dot{v}_e(t)\}^T \{n(\xi)\}^T \{n(\xi)\}\{\dot{v}_e(t)\}$, si sostituisca nella (29):

$$T_e = \frac{1}{2}\{\dot{v}_e(t)\}^T \left[a\rho A\int_{-1}^{1} \{n(\xi)\}^T \{n(\xi)\} d\xi \right] \{\dot{v}_e(t)\} \quad (30)$$

L'energia cinetica di elemento può anche essere definita in base alla (7):

$$T_e = \frac{1}{2}\{\dot{v}_e(t)\}^T [M_e]\{\dot{v}_e(t)\} \quad (31)$$

Pertanto, confrontando la (30) con la (31), la matrice massa di elemento diventa:

$$\left[M_e\right] = a\rho A\int_{-1}^{1}\left\{n(\xi)\right\}^T\left\{n(\xi)\right\}d\xi = \frac{a\rho A}{105}\begin{bmatrix} 78 & 22a & 27 & -13a \\ 22a & 8a^2 & 13a & -6a^2 \\ 27 & 13a & 78 & -22a \\ -13a & -6a^2 & -22a & 8a^2 \end{bmatrix} (32)$$

Si ripeta ora il procedimento per il calcolo della matrice rigidezza di elemento utilizzando la energia potenziale elastica di elemento ottenuta in base alla (16):

$$U_e = \frac{1}{2}\int_{-a}^{a} EI_z\left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx = \frac{1}{2}\int_{-1}^{1} EI_z\frac{1}{a^4}\left(\frac{\partial^2 v}{\partial \xi^2}\right)^2 a d\xi \,(33)$$

Una volta calcolato $\frac{\partial^2 v(\xi,t)}{\partial \xi^2} = \left\{\frac{d^2 n(\xi)}{d\xi^2}\right\}\left\{v_e\right\}$ e $\left(\frac{\partial^2 v(\xi,t)}{\partial \xi^2}\right)^2 = \left\{v_e\right\}^T\left\{\frac{d^2 n(\xi)}{d\xi^2}\right\}^T\left\{\frac{d^2 n(\xi)}{d\xi^2}\right\}\left\{v_e\right\}$ si sostituisca nella (33):

$$U_e = \frac{1}{2}\left\{v_e\right\}^T\left[EI_z\frac{1}{a^3}\int_{-1}^{1}\left\{\frac{d^2 n(\xi)}{d\xi^2}\right\}^T\left\{\frac{d^2 n(\xi)}{d\xi^2}\right\}d\xi\right]\left\{v_e\right\} (34)$$

L'energia potenziale elastica di elemento può anche essere definita in base alla (9):

$$U_e = \frac{1}{2}\left\{v_e\right\}^T\left[K_e\right]\left\{v_e\right\} (35)$$

Pertanto, confrontando la (34) e la (35), la matrice rigidezza di elemento diventa:

$$\left[K_e\right] = EI_z\frac{1}{a^3}\int_{-1}^{1}\left\{\frac{d^2 n(\xi)}{d\xi^2}\right\}^T\left\{\frac{d^2 n(\xi)}{d\xi^2}\right\}d\xi = \frac{EI_z}{2a^3}\begin{bmatrix} 3 & 3a & -3 & 3a \\ 3a & 4a^2 & -3a & 2a^2 \\ -3 & -3a & 3 & -3a \\ 3a & 2a^2 & -3a & 4a^2 \end{bmatrix} (36)$$

5. sommare le energie cinetiche e potenziali di elemento per ottenere le energie del sistema completo (assemblaggio delle matrici massa e rigidezza)

Sia $\left\{v\right\}$ il vettore contenente tutti i g.d.l. della trave a 4 elementi considerata:

$$\left\{v\right\}^T = \left\{v_1 \quad \theta_{z1} \quad v_2 \quad \theta_{z2} \quad v_3 \quad \theta_{z3} \quad v_4 \quad \theta_{z4} \quad v_5 \quad \theta_{z5}\right\} (37)$$

Che può essere relazionata al vettore contenente i g.d.l. di ogni singolo elemento finito mediante la matrice $\left[a_e\right]$:

$$\left\{v_e\right\} = \left[a_e\right]\left\{v\right\} (38)$$

dove, per esempio la matrice di trasformazione $\left[a_1\right]$ per il primo elemento è:

$$\left[a_1\right] = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} (39)$$

L'energia cinetica totale è data dalla somma delle singole energie cinetiche di elemento, secondo la relazione:

$$T_{tot} = \sum_{e=1}^{4} T_e = \frac{1}{2}\left\{\dot{v}\right\}^T\sum_{e=1}^{4}\left(\left[a_e\right]^T\left[M_e\right]\left[a_e\right]\right)\left\{\dot{v}\right\} = \frac{1}{2}\left\{\dot{v}\right\}^T\left[M\right]\left\{\dot{v}\right\} (40)$$

Esplicitando la (40) fino al secondo elemento si ottiene:

$$T_{tot}^{(2)}=\frac{1}{2}\begin{Bmatrix}\dot{v}_1 & \dot{\theta}_{z1} & \dot{v}_2 & \dot{\theta}_{z2} & \dot{v}_3 & \dot{\theta}_{z3}\end{Bmatrix}\begin{bmatrix} m_{11}^{(1)} & m_{12}^{(1)} & m_{13}^{(1)} & m_{14}^{(1)} & & \\ m_{21}^{(1)} & m_{22}^{(1)} & m_{23}^{(1)} & m_{24}^{(1)} & & \\ m_{31}^{(1)} & m_{32}^{(1)} & m_{33}^{(1)}+m_{11}^{(2)} & m_{34}^{(1)}+m_{12}^{(2)} & m_{13}^{(2)} & m_{14}^{(2)} \\ m_{41}^{(1)} & m_{42}^{(1)} & m_{43}^{(1)}+m_{21}^{(2)} & m_{44}^{(1)}+m_{22}^{(2)} & m_{23}^{(2)} & m_{24}^{(2)} \\ & & m_{31}^{(2)} & m_{32}^{(2)} & m_{33}^{(2)} & m_{34}^{(2)} \\ & & m_{41}^{(2)} & m_{42}^{(2)} & m_{43}^{(2)} & m_{44}^{(2)} \end{bmatrix}\begin{Bmatrix}\dot{v}_1 \\ \dot{\theta}_{z1} \\ \dot{v}_2 \\ \dot{\theta}_{z2} \\ \dot{v}_3 \\ \dot{\theta}_{z3}\end{Bmatrix} \quad (41)$$

Un ragionamento simile può essere fatto relativamente all'energia potenziale e alla matrice rigidezza:

$$U_{tot}=\sum_{e=1}^{4}U_e=\frac{1}{2}\{v\}^T\sum_{e=1}^{4}\left(\left[a_e\right]^T\left[K_e\right]\left[a_e\right]\right)\{v\}=\frac{1}{2}\{v\}^T\left[K\right]\{v\} \quad (42)$$

$$U_{tot}^{(2)}=\frac{1}{2}\begin{Bmatrix}v_1 & \theta_{z1} & v_2 & \theta_{z2} & v_3 & \theta_{z3}\end{Bmatrix}\begin{bmatrix} k_{11}^{(1)} & k_{12}^{(1)} & k_{13}^{(1)} & k_{14}^{(1)} & & \\ k_{21}^{(1)} & k_{22}^{(1)} & k_{23}^{(1)} & k_{24}^{(1)} & & \\ k_{31}^{(1)} & k_{32}^{(1)} & k_{33}^{(1)}+k_{11}^{(2)} & m_{34}^{(1)}+k_{12}^{(2)} & k_{13}^{(2)} & k_{14}^{(2)} \\ k_{41}^{(1)} & k_{42}^{(1)} & k_{43}^{(1)}+k_{21}^{(2)} & k_{44}^{(1)}+k_{22}^{(2)} & k_{23}^{(2)} & k_{24}^{(2)} \\ & & k_{31}^{(2)} & k_{32}^{(2)} & k_{33}^{(2)} & k_{34}^{(2)} \\ & & k_{41}^{(2)} & k_{42}^{(2)} & k_{43}^{(2)} & k_{44}^{(2)} \end{bmatrix}\begin{Bmatrix}v_1 \\ \theta_{z1} \\ v_2 \\ \theta_{z2} \\ v_3 \\ \theta_{z3}\end{Bmatrix} \quad (43)$$

6. **Imporre le condizioni al contorno.**

Le condizioni al contorno geometriche possono essere imposte, nel caso di incastri, bloccando i g.d.l. coinvolti. Per esempio se la trave è incastrata ad una estremità, allora v_1 e θ_{z1} saranno zero. Questo si ripercuote nelle matrici massa e rigidezza eliminando le righe e le colonne relative a quei g.d.l.; ad esempio, eliminando le parti di matrici evidenziate:

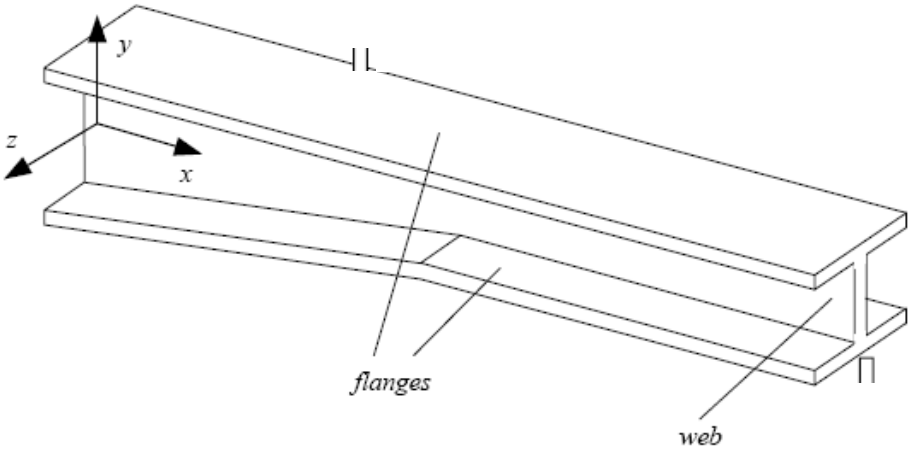
$$\left[M\right]=\begin{bmatrix} m_{11} & m_{12} & m_{13} & \cdots & m_{110} \\ m_{21} & m_{22} & m_{23} & \cdots & m_{210} \\ m_{31} & m_{32} & m_{33} & \cdots & m_{310} \\ \vdots & \vdots & \vdots & \ddots & \vdots \\ m_{101} & m_{102} & m_{103} & \cdots & m_{1010} \end{bmatrix};\left[k\right]=\begin{bmatrix} k_{11} & k_{12} & k_{13} & \cdots & k_{110} \\ k_{21} & k_{22} & k_{23} & \cdots & k_{210} \\ k_{31} & k_{32} & k_{33} & \cdots & k_{310} \\ \vdots & \vdots & \vdots & \ddots & \vdots \\ k_{101} & k_{102} & k_{103} & \cdots & k_{1010} \end{bmatrix}$$

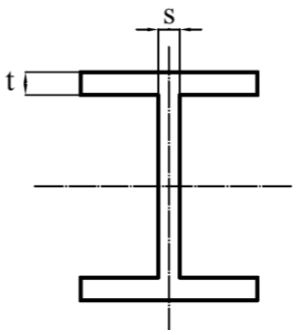
7. **risolvere il problema (ad esempio un problema agli autovettori-autovalori per ottenere modi e frequenze naturali).**

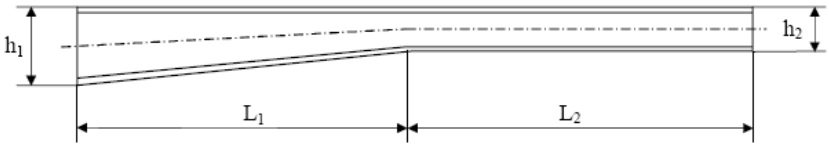
Nel metodo di R.R. l'accuratezza della soluzione si otteneva aumentando i termini delle funzioni prescritte. Nel metodo ad elementi finiti, per aumentare il numero di funzioni prescritte (funzioni di forma) occorre aumentare il numero di nodi ed elementi, pertanto infittire la mesh. Si può anche aumentare il grado del polinomio della funzione di forma, scegliendo una serie di ordine superiore.

Esercizio 2 (da portare in forma scritta all'esame). Implementazione del metodo ad elementi finiti in Matlab di Trave incastrata

Implementare in ambiente matlab il metodo agli elementi finiti per la trave incastrata di figura. Modellare la trave con 4 elementi. Mostrare le prime 8 frequenze naturali e le relative forme modali. Stampare il listato matlab.







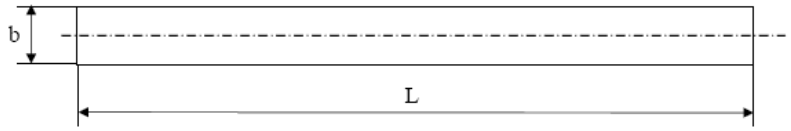


Table 1: Physical properties (Aluminum)

Parameter	Value
Density	$\rho = 2700\ \mathrm{Kg/m^3}$
Young's modulus of elasticity	$E = 7.1\times10^{10}\ \mathrm{N/m^2}$
Poisson's ratio	$\nu = 0.33$

Table 3 Boundary conditions

Position	Value
x = 0	Clamped
x = L	Free

Sia L=0.8m; L_1=L_2=0.4 m; h_1=4cm; h_2=2cm; b=3cm; t=3 mm; s=2.5mm.

Traccia di soluzione

1) Inserimento dati

% Dati:
n = 4; % Numero degli elementi
dati = zeros(9,n);

dati(1,:) = [0.03 0.03 0.03 0.03]; % b Larghezza della sezione dell'elemento

2) Definizione delle matrici di Massa e Rigidezza dell'elemento

Introdurre le formule di matrice massa e rigidezza di elemento (dare un indice anche all'elemento)

M(i,j,k), dove k è l'indice dell'elemento. i e j sono gli indici della matrice.

3)Assemblaggio
4)condizioni al contorno

% Applicazione delle condizioni al contorno (trave vincolata)
M(1:2,:) = [];
K(1:2,:) = [];
M(:,1:2) = [];
K(:,1:2) = [];

5)Risoluzione del problema agli autovalori

```
[fi,omegaq] = eig(K,M);
freq_nat=sqrt(omegaq)/(2*pi)
```

6) plot modi

```
primo_autovettore=[0 fi(1:2:end,1)']
plot(primo_autovettore),title('primo modo')
```

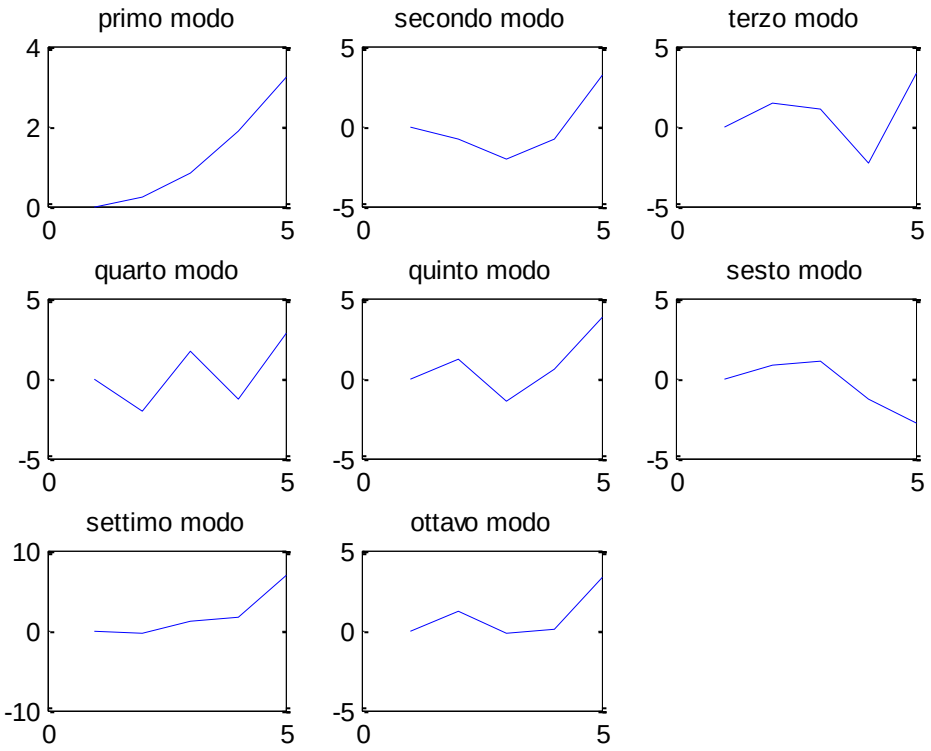


Figura 12. primi 8 modi per la trave incastrata soggetta a flessione modellata con 4 elementi finiti.

4.2 Matrice delle masse "Lumped" e "consistent"

La matrice delle masse ottenuta per la trave in (32) è detta "consistent". E' chiamata così perche si utilizza per ottenerla lo stesso metodo usato per la matrice rigidezza. Spesso però, molti problemi, si risolvono con accuratezza anche utilizzando forme più semplici della matrice massa. La formulazione più semplice della matrice massa è quella denominata "lumped", cioè questa matrice è ottenuta trascurando gli effetti inerziali e concentrando l'intera massa nei nodi dell'elemento. Nel caso dello studio delle vibrazioni flessionali della trave, in cui ogni nodo ha 2 g.d.l., si concentra metà della massa nel g.d.l. traslazionale di sinistra e l'altra metà nel g.d.l. traslazionale di destra. Si ottiene pertanto la matrice massa:

$$[M] = a\rho A \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & & 0 \end{bmatrix} \quad (44)$$

E' evidente che la formulazione "consistent" dia risultati più accurati, infatti viene trascurato l'accoppiamento fra i gradi di libertà così come l'effetto inerziale. Tuttavia, la formulazione "lumped" è molto utilizzata poiché risultando in una matrice massa diagonale determina consistenti riduzioni del tempo computazionale.

In Figura 13 è mostrato l'errore percentuale sulle prime 15 frequenze naturali per una trave a mensola utilizzando al formulazione lumped e consistent. Interessante notare come all'aumentare del numero di elementi finiti le due formulazioni danno risultati sempre più simili.

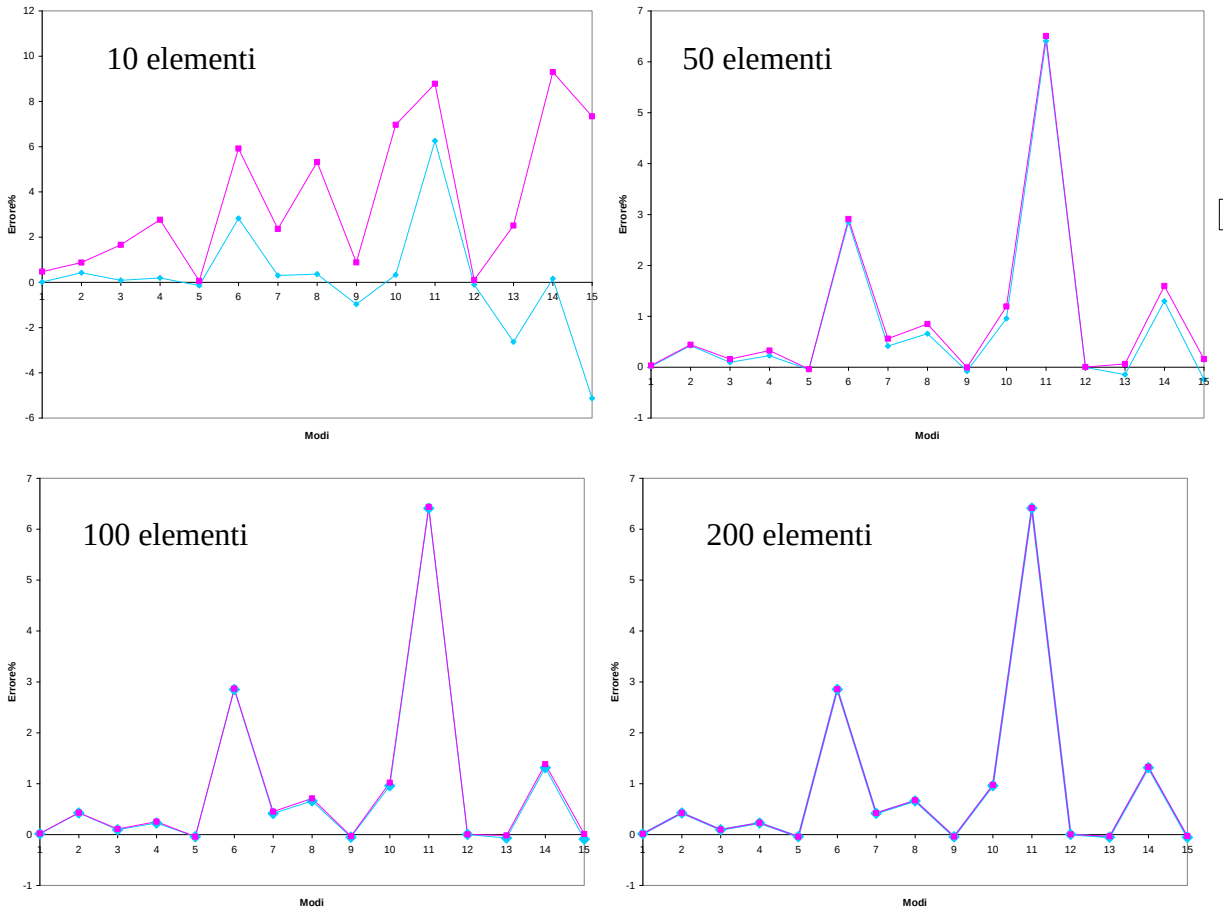


Figura 13. Errore percentuale (rispetto alla soluzione teorica utilizzando i sistemi continui) sulle prime 15 frequenze naturali per una trave incastrata modellata con un numero variabile di elementi (10, 50, 100, 200). In viola utilizzando la formulazione "lumped" ed in azzurro utilizzando la formulazione "consistent".

4.3. Tecniche per diminuire il numero di gradi di libertà (Model reduction)

La creazione della mesh può generare un elevato numero di elementi e pertanto un elevato numero di g.d.l. aumentando pertanto il costo computazionale. In particolare, quando vengono usati schemi automatici di generazione della mesh o quando si usano elementi tridimensionali il numero di g.d.l. aumenta velocemente. Esistono diverse tecniche per ridurre il numero di g.d.l.. nel seguito ne sono chiarite alcune:

- semplificazione del modello. Eliminare dal disegno CAD (prima della mesh) tutti quegli elementi che risultano inutili per l'analisi (bulloni, rivetti, piccoli fori, etc);
- semplificazione per idealizzazione. Utilizzare se possibile elementi trave o piastra piuttosto che elementi tridimensionali (ad esempio per meshare una struttura a forma di trave);
- usare la simmetria del problema.
- Riduzione modale. Sia un problema di dimensione N ($[M]\{\ddot{x}\}+[C]\{\dot{x}\}+[K]\{x\}=\{f\}$), pertanto le matrici massa e rigidezza avranno dimensione NXN e il vettore dei g.d.l. x avrà dimensione NX1. Applicando un cambiamento di coordinate $\{x\}=[\Phi]_r\{q\}_r$ dove $\{q\}_r$ ha dimensione mX1 con $m<N$ e $[\Phi]_r$ rappresenta la matrice (rettangolare) degli autovettori (in cui gli autovettori relativi alle pulsazioni naturali più elevate sono stati eliminati), la taglia del problema si riduce. In particolare, le matrici massa e rigidezza risultano diagonali e di dimensione mXm:
  $[m]=\Phi^T[M]\Phi;[k]=\Phi^T[K]\Phi;[c]=\Phi^T[C]\Phi;\{p\}=\Phi^T\{f\};$ (45)

- Riduzione di Guyan o riduzione statica. In questa procedura di riduzione, i g.d.l. totali del problema vengono divisi in due categorie: i g.d.l. master e gli slave. I g.d.l. slave sono quelli che possono essere rimossi ed espressi in funzione dei g.d.l. master mediante una relazione statica. In particolare, l'equazione del moto per un sistema non smorzato forzato può essere riscritta in funzione dei g.d.l. slave e master nel seguente modo:

$$\begin{pmatrix} M_{mm} & M_{ms} \\ M_{sm} & M_{ss} \end{pmatrix} \begin{Bmatrix} \ddot{u}_m \\ \ddot{u}_s \end{Bmatrix} + \begin{pmatrix} K_{mm} & K_{ms} \\ K_{sm} & K_{ss} \end{pmatrix} \begin{Bmatrix} u_m \\ u_s \end{Bmatrix} = \begin{Bmatrix} f_m \\ f_s \end{Bmatrix} (46)$$

Le due equazioni matriciali in (46) sono pertanto:

$$M_{mm}\ddot{u}_m + M_{ms}\ddot{u}_s + K_{mm}u_m + K_{ms}u_s = f_m (47)$$

$$M_{sm}\ddot{u}_m + M_{ss}\ddot{u}_s + K_{sm}u_m + K_{ss}u_s = f_s (48)$$

Il metodo prevede che la relazione fra i g.d.l. master e slave non venga influenzata dalla massa e inerzia (riduzione statica), pertanto dalle (47) (48) il contributo dei termini inerziali è annullato:

$$K_{mm}u_m + K_{ms}u_s = f_m (49)$$

$$K_{sm}u_m + K_{ss}u_s = f_s (50)$$

Risolvendo la (50) in funzione di u_s si ottiene:

$$u_s = K_{ss}^{-1} f_s - K_{ss}^{-1} K_{sm} u_m (51)$$

Sostituendo la (51) nella (49) si ottiene:

$$K_{mm}u_m + K_{ms}(K_{ss}^{-1} f_s - K_{ss}^{-1} K_{sm} u_m) = f_m \quad \rightarrow \left(K_{mm} - K_{ss}^{-1} K_{sm}\right) u_m = f_m - K_{ms} K_{ss}^{-1} f_s (52)$$

che può essere riscritta come:

$$K_r u_r = f_r (53)$$

dove:

$$u_r = u_m \text{ e } f_r = f_m - K_{ms} K_{ss}^{-1} f_s (54)$$

Ma

$$\{f_r\} = [W]^T \{f\} = [W]^T \begin{Bmatrix} f_m \\ f_s \end{Bmatrix} (55)$$

Confrontando al (54) con la (55) si ottiene la matrice di trasformazione di riduzione di coordinate W:

$$\{f_r\} = \begin{bmatrix} I & -K_{ms} K_{ss}^{-1} \end{bmatrix} \{f\} (56)$$

Abbiamo quindi ricavato la matrice W che riduce il numero di coordinate $u = Wu_r$. La nuova equazione del moto si ottiene poi con le (45).

4.4 Smorzamento strutturale

Una accurata analisi delle strutture reali suggerisce che lo smorzamento viscoso non è rappresentativo per modelli a molti gradi di libertà, come ad esempio i modelli ad elementi finiti. Appare infatti che lo smorzamento, in questi casi, abbia un andamento legato all'inverso della frequenza.

Per un sistema ad un grado di libertà lo smorzamento strutturale assume la forma:

$c = h/\omega,$

pertanto la relativa forza di smorzamento strutturale assume la forma:

$f_{smorz,strut} = -(h/\omega)\dot{x} = -ihx$.

L'equazione del moto per un sistema forzato ad un grado di libertà con smorzamento strutturale diventa:

$m\ddot{x} + ihx + kx = f \rightarrow m\ddot{x} + (k + ih)x = f$

Ponendo $(k + ih) = k(1 + i\eta)$, con η detto loss factor, si ottiene nell'equazione del moto un termine di rigidezza complessa:

$m\ddot{x} + k(1 + i\eta)x = f$

Tipici valori di η sono fra 10^{-5} per alluminio o acciaio fino ad 1 per materiali plastici (gomme)

Per sistemi a molti gradi di libertà, si può generalizzare:

$[M]\{\ddot{x}\} + [K + iH]\{x\} = \{f\}$

Esercizio 3 (da portare in forma scritta all'esame)- Trave incastrata in Nastran-Patran

Consideriamo la trave incastrata (Figura 14, Figura 15) con sezione rettangolare 0,15x0,44 m e lunghezza 10 m. La trave è di alluminio. La trave dovrà essere modellata con elementi CBEAM (1D) e con elementi CTETRA (3D) come in (Figura 14, Figura 15).

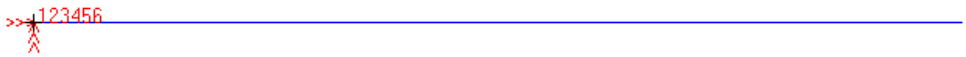


Figura 14: trave incastrata modellata con elementi CBEAM.

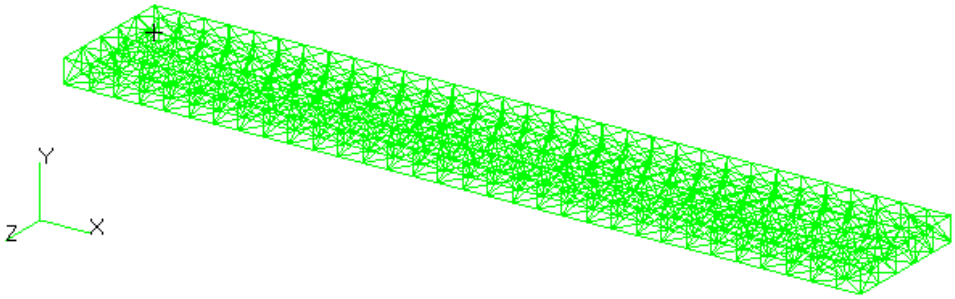


Figura 15: trave modellata con elementi CTETRA.

L'incastro nel caso della trave modellata con elementi CBEAM è stato ottenuto impedendo le tre traslazioni e le tre rotazioni ad una estremità, cioè in un nodo, mentre nel caso della trave modellata con elementi CTETRA è stato ottenuto impedendo questi gradi d libertà ad ogni nodo di un lato del parallelepipedo (Figura 16).

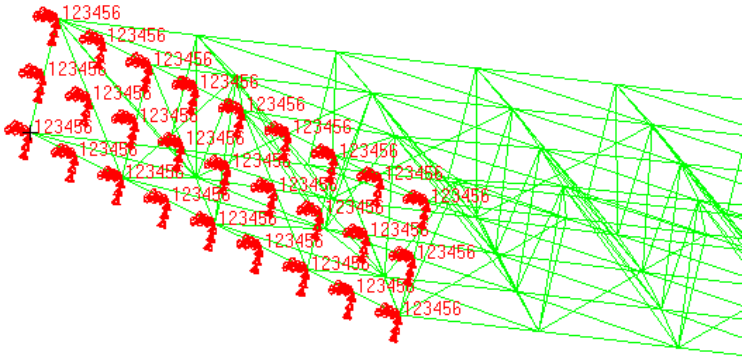


Figura 16: vincoli per la trave modellata con elementi CTETRA.

Si richiede di modellare con MSC Patran la trave considerandola prima come monodimensionale (usando elementi CBEAM) e poi tridimensionale (usando elementi CTETRA) e confrontare le frequenze naturali ottenute (SOL103) con le frequenze naturali teoriche. Inserire poi i dati nella seguente Tabella 1.

Tabella 1 Trave incastrata modellata con 1000 elementi CBEAM e 1000 elementi CTETRA

Modi	Frequenze TRAVE CBEAM [Hz]	Frequenze TRAVE CTETRA [Hz]	Modo (indicare il tipo di modo, flessionale , torsionale, etc)	Frequenze Teoriche [Hz]	Errore% Teorica vs CBEAM	Errore% Teorica vs CTETRA
1						
2						
3						
4						
5						
6						
7						
8						
9						
10						

Traccia di soluzione 103 in MSC.Nastran

L’analisi modale viene eseguita in MSC.Nastran selezionando la soluzione 103.

In particolare nella sezione Analysis di MSC.Patran dopo aver impostato “Analyze-Entire Model-Full Run” si clicca in “Solution Type” per selezionare la soluzione desiderata (Figura 17).

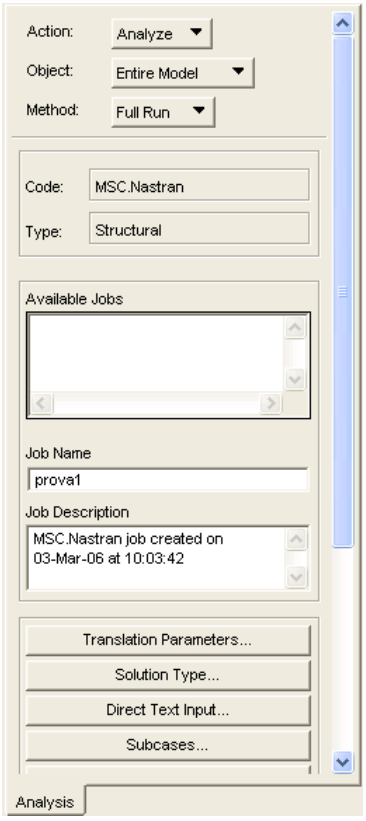


Figura 17: interfaccia grafica in MSC.Patran per impostare la soluzione desiderata.

Compare una finestra in cui si seleziona la soluzione voluta, in questo caso “Normal Modes” (Figura 18).

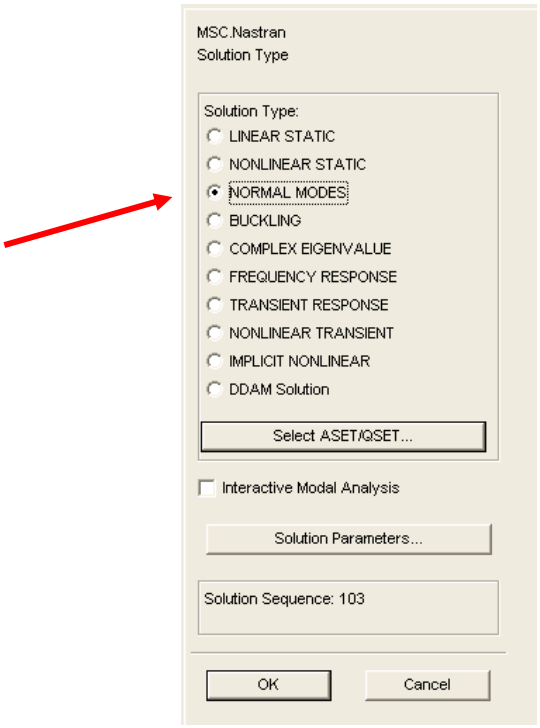


Figura 18: Interfaccia grafica con l'elenco delle soluzioni disponibili.

Successivamente si clicca in "Solution Parameters" (Figura 19) in cui si settano alcuni parametri tra cui il modo con cui considerare la massa (*Lumped* o *Coupled* (cioè *consistent*).

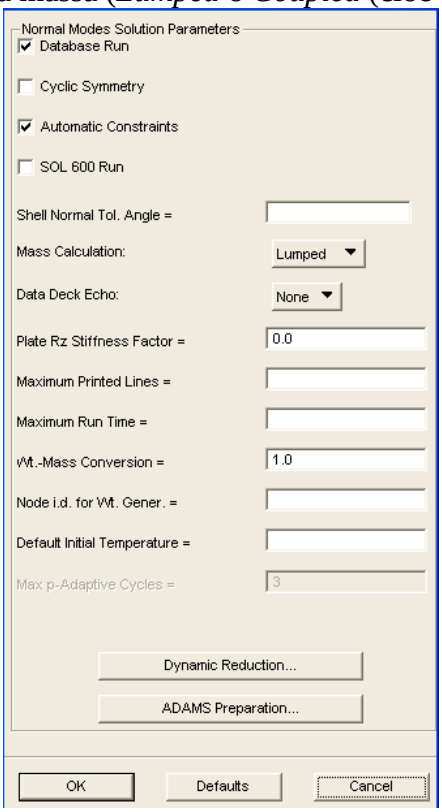


Figura 19: interfaccia grafica per selezionare alcuni paramenti dell'analisi.

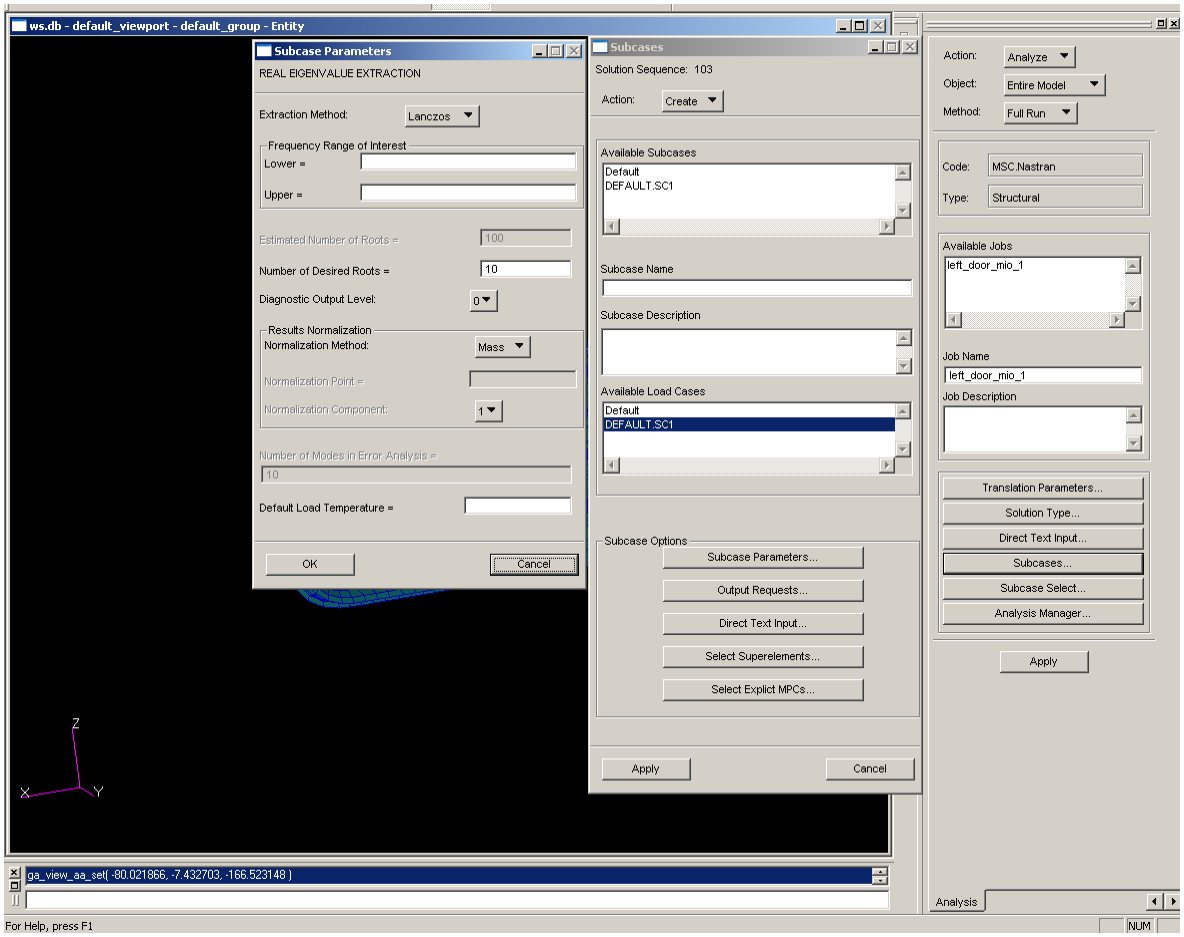


Figura 20: interfaccia grafica per la creazione dei Subcases.

Seguendo il percorso *Subcases/subcase parameters* si possono scegliere il numero di frequenze naturali da calcolare (number of desired roots). Infine si clicca su “Apply” (Figura 17) per far partire la creazione del file .bdf che serve a MSC.Nastran per calcolare le soluzioni desiderate.

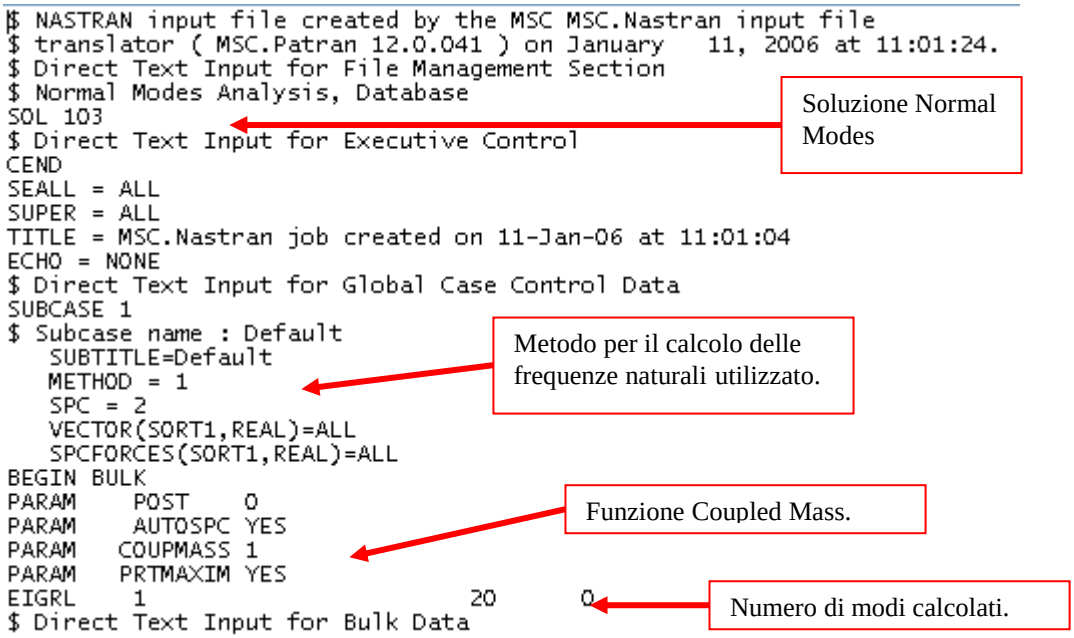


Figura 21: file .bdf per la soluzione 103.

Le cards presenti nel file .bdf per la soluzione Normal Modes sono:

- METHOD: indica con quale metodo sono calcolati gli autovalori (1= metodo di Lanczos).
- EIGRL: definisce i dati per l'elaborazione per ottenere gli autovalori (vibrazioni o deformazioni) nell'analisi con il metodo di Lanczos.

Elementi CBEAM

per creare una trave con elementi CBEAM in Patran., seguire i seguenti suggerimenti (Figura 22):

1) Geometry/create/curve/xyz e inserire la lunghezza della trave in "vector coordinate list"
2) Utilizzare BAR2 come tipologia di mesh
3) Nella definizione delle "properties" utilizzare le impostazioni di figura

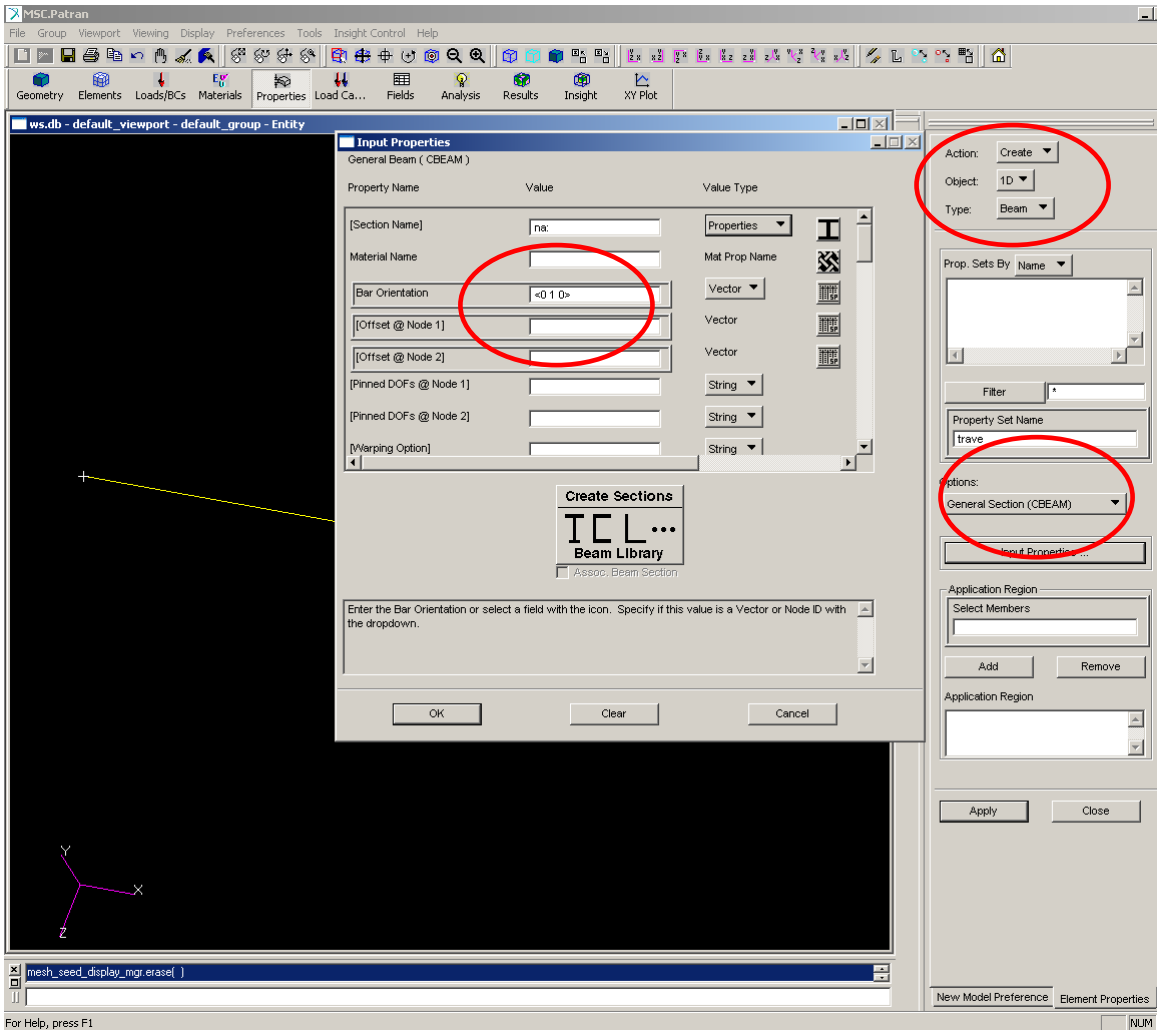


Figura 22.

I gradi di libertà associati all'elemento CBEAM sono tre spostamenti, tre rotazioni e la torsione. L'inserimento delle proprietà dell'elemento BEAM è mostrato in Figura 23.

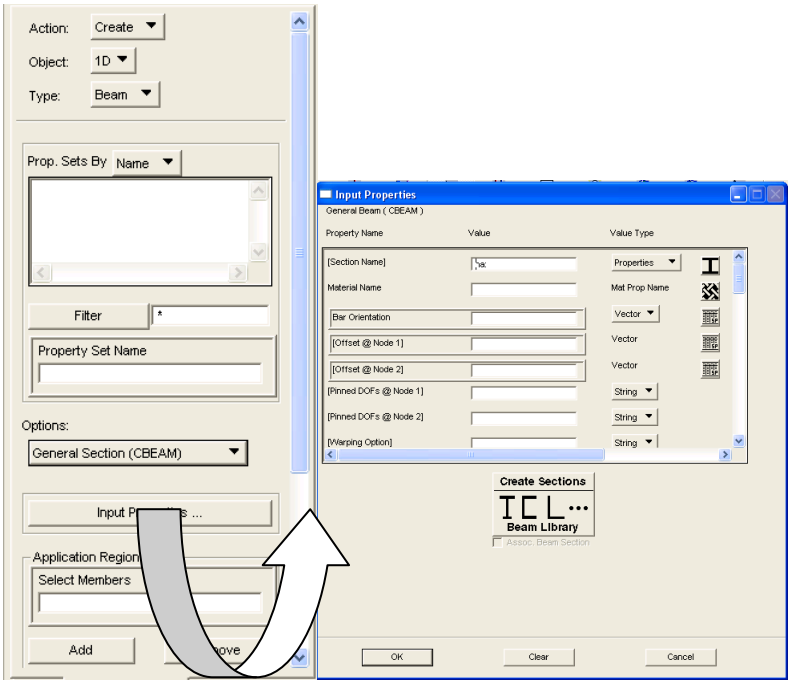


Figura 23: interfaccia grafica di come inserire le proprietà degli elementi CBEAM.

Elementi 3D

Gli elementi solidi di MSC.Nastran includono:

- elementi a quattro facce chiamati CTETRA;
- elementi a cinque facce chiamati CPENTA;
- elementi a sei facce chiamati CHEXA.

Questi elementi sono mostrati in Figura 24.

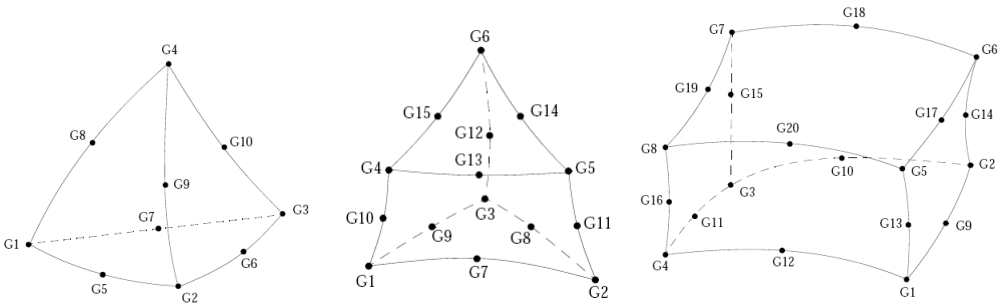


Figura 24: elementi solidi TETRA, PENTA e HEXA.

Gli elementi a quattro nodi CTETRA e quelli a sei nodi CPENTA sono molto rigidi e quindi non dovrebbero essere usati nella modellazione. Gli elementi a dieci nodi CTETRA e quelli a quindici nodi CPENTA sono più adattabili rispetto i precedenti. Gli elementi a otto nodi CHEXA sono i migliori e possono essere usati per modellare le più svariate geometrie.
La scelta del tipo di elemento va fatta quando si fa la mesh del pezzo utilizzando l'interfaccia grafica di Figura 25.

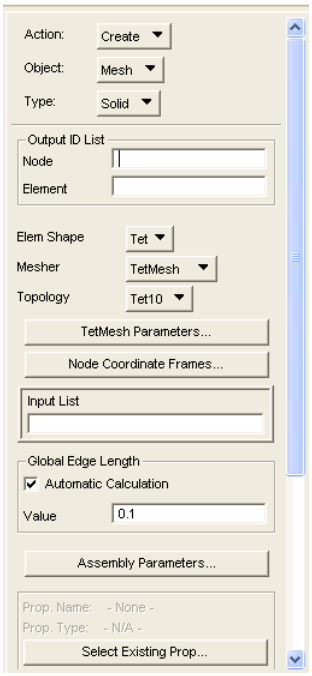


Figura 25: interfaccia grafica di come scegliere il tipo di mesh che si desidera fare.

L'elemento CTETRA ha quattro nodi ai vertici ed altri sei nodi sui lati (Figura 24).
I nodi G1 e G4 definiscono i vertici, mentre i nodi intermedi G5-G10 sono opzionali. La loro omissione rendono la mesh poco accurata, mentre gli elementi CTETRA a dieci nodi sono molto usati perché consentono una buona modellazione.

Di seguito un esempio di file BDF di Nastran

```
$ NASTRAN input file created by the MSC MSC.Nastran input file
$ translator ( MSC.Patran 12.0.041 ) on September 21, 2005 at 10:52:43.
$ Normal Modes Analysis, Database
SOL 103
CEND
SEALL = ALL
SUPER = ALL
ECHO = NONE
SUBCASE 1
$ Subcase name : Default
   SUBTITLE=Default
   METHOD = 1


stress=all
spc=2

BEGIN BULK
PARAM    POST    0
PARAM    AUTOSPC YES
PARAM   PRTMAXIM YES
EIGRL    1                       16       0
PSOLID  1       1
$ Pset: Property_1
CTETRA  1       1       5774    2133    6428    6367    24353   24376
        24380   35839   13174   17539
CTETRA  2       1       2196    2185    1526    2172    40246   18845
……
```

```
$ Material : Material_1
MAT1*    1               6.8+10                          .3
*        2900.


$ Nodes of the Entire Model
GRID*    1                                -0.02603926542250.06873768028045*A1
*A1      0.15908825799237
GRID*    2                                -0.02605234924510.07670429191051*A2
*A2      0.14779071433393
GRID*    3                                -0.02791697484020

$ Loads for Load Case : Default
SPCADD   2       1
$ Displacement Constraints of Load Set : vincoli
SPC1     1       123456  183     199     200     201     202     203
         204     225     226     227     228
SPC1     1       123456  264     THRU    277
$ Referenced Coordinate Frames
ENDDATA
```

Calcolo delle frequenze teoriche

Vibrazioni flessionali di una trave

Le **frequenze naturali flessionali** della trave possono essere calcolata analiticamente attraverso l'equazione di Eulero:

$$EI = \frac{\partial^4 v}{\partial x^4} + \rho S \frac{\partial^2 v}{\partial t^2} = 0 \qquad \textbf{(1)}$$

Dalla risoluzione della (1) si ottiene la seguente relazione per calcolare le frequenze naturali in base delle condizioni al contorno imposte:

$$f_n = \frac{\beta_n{}^2}{2\pi} \sqrt{\frac{E \cdot I}{\rho \cdot A}} \qquad \textbf{(2)}$$

Dove: E è il modulo di Young;

ρ è la densità;

A è la sezione;

I è il momento d'inerzia;

l è la lunghezza della trave.

Vengono riportati di seguito i valori dei coefficienti $\beta_n l$ per il caso di trave incastrata (**Tabella 1**) e per il caso di trave free-free (**Tabella 2**).

$\beta_1 l$	$\beta_2 l$	$\beta_3 l$	$\beta_4 l$	$\beta_5 l$	$\beta_6 l$	$\beta_7 l$	$\beta_8 l$	$\beta_9 l$	$\beta_{10} l$
1,875	4,694	7,854	10,995	14,137	17,278	20,420	23,562	26,703	29,845

Tabella 1: valori di βl per trave incastrata.

$\beta_1 l$	$\beta_2 l$	$\beta_3 l$	$\beta_4 l$	$\beta_5 l$	$\beta_6 l$	$\beta_7 l$	$\beta_8 l$
4,730	7,853	10,996	14,137	17,278	20,347	23,562	26,703

Tabella 2: valori di βl per trave free-free.

Vibrazioni torsionali di una trave

Le **frequenze naturali torsionali** per una trave incastrata possono essere calcolate analiticamente attraverso la seguente equazione differenziale:

$$Ip_m \frac{\partial^2 \theta}{\partial t^2} = GJ \frac{\partial^2 \theta}{\partial x^2} \qquad \textbf{(3)}$$

Dalla risoluzione della (3) si ottengono le relazioni per calcolare le frequenze naturali al variare delle condizioni al contorno. Per una trave incastrata ad una estremità la relazione è:

$$f_n = \frac{(2n+1)\cdot c}{4 \cdot l} \qquad \textbf{(4)}$$

dove $c = \sqrt{\frac{G \cdot J}{Ip_m}}$

J Torsional Constant;

$Ip_m = \rho \cdot Ip_A$ momento d'inerzia polare della massa;

$Ip_A = \frac{b \cdot h^3 + h \cdot b^3}{12}$ momento d'inerzia polare della sezione;

n=0,1,2,…..(la prima frequenza torsionale si ottiene con n=0)

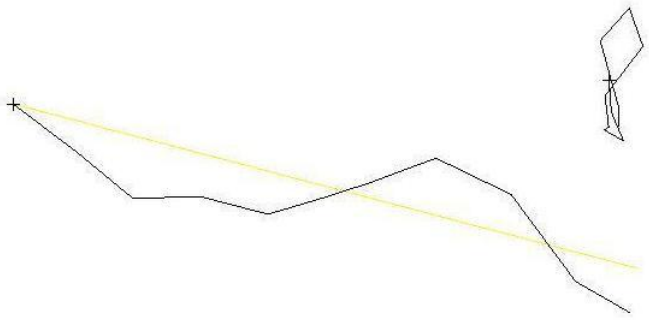


Figura 26: modo torsionale per una trave incastrata.

Torsional Constant

J è usato per indicare la Torsional Constant.

Sfortunatamente la stessa variabile usata per indicare la torsional constant è usata anche per indicare il momento polare d'inerzia della sezione (qui indicato con Ip_A). Queste due grandezze NON sono

la stessa cosa. Ad aggiungere confusione nel caso di sezione circolare i due sono numericamente identici.

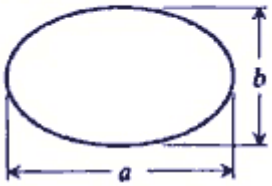
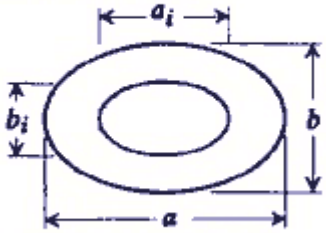
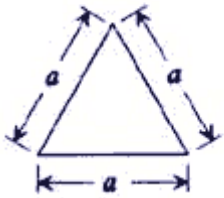
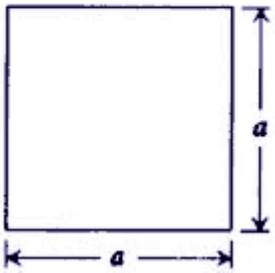
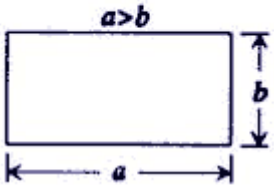
$$Ip_A = J = \frac{\pi \cdot r^4}{2} \qquad \textbf{(5)}$$

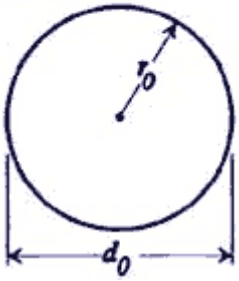
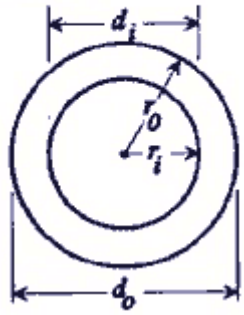
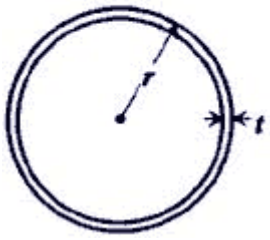
Il momento polare d'inerzia della sezione rispetto ad un asse OZ (asse polare) perpendicolare al piano della sezione è dato da:

$$Ip_A = \int r^2 dm = \int x^2 dm + \int y^2 dm = I_x + I_y \qquad \textbf{(6)}$$

La Torsional Constant varia invece a seconda della forma della sezione (Tabella 3).

TORSIONAL CONSTANT

1. Ellipse 	$J = \dfrac{\pi a^3 b^3}{16\left(a^2 + b^2\right)}$
2. Hollow ellipse  $k = a_i/a = b_i/b$	$J = \dfrac{\pi a^3 b^3}{16\left(a^2 + b^2\right)}\left(1 - k^4\right)$
3. Equilateral triangle 	$J = \dfrac{a^4\sqrt{3}}{80}$
4. Square 	$J = 0.1406a^4$
5. Rectangle 	$J = \dfrac{ab^3}{3}\left(1 - 0.630\dfrac{b}{a} + 0.052\dfrac{b^5}{a^5}\right)$

6.  	$J = \dfrac{4\overline{A^2}}{\int_0^L \left(1/t\,(s)\right) ds}$ where $\overline{A}$ =area enclosed by middle line of wall L=entire length of middle line of wall For constant t $J = \dfrac{4\overline{A^2}t}{L}$
7. Solid  	$J = \dfrac{1}{2}\pi r_0^4 = \dfrac{1}{32}\pi d_0^4$
8. Hollow  	$J = \dfrac{1}{2}\pi\left(r_0^4 - r_i^4\right) = \dfrac{1}{32}\pi\left(d_0^4 - d_i^4\right)$
9. Very Thin  	$J = 2\pi r^3 t$

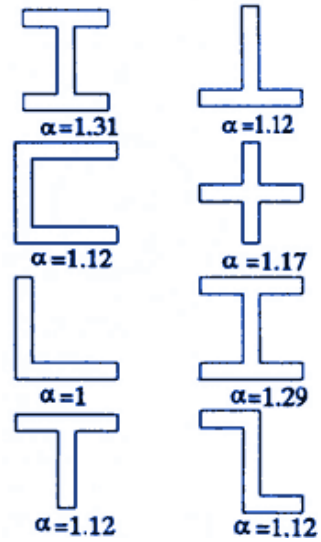
10. Any open section  	$J = \frac{a}{3}\sum_{i-1}^{M} b_i t_i^3$ *where M is number of* *straight or curved segments of thickness* t_i *and* *width or height* b_i *compo* sin *g the* sec *tion* *set* $a = 1$ *except as designated otherwise*

Tabella 3: Torsional Constant per le varie sezioni.

Vibrazioni longitudinali di una trave

Le **frequenze naturali longitudinali** per una trave possono essere calcolate analiticamente attraverso la seguente equazione differenziale:

$$\frac{\partial^2 u}{\partial t^2} = \frac{E}{\rho}\frac{\partial^2 u}{\partial x^2} \qquad \textbf{(7)}$$

Dalla risoluzione della (7) si ottengono le relazioni per calcolare le frequenze naturali al variare delle condizioni al contorno.

Per una trave incastrata la soluzione è:

$$f_n = \frac{(2n-1)}{4l}\sqrt{\frac{E}{\rho}} \qquad \textbf{(8)}$$

dove: ρ densità;

E modulo di Young;

n è in modo;

l è la lunghezza.

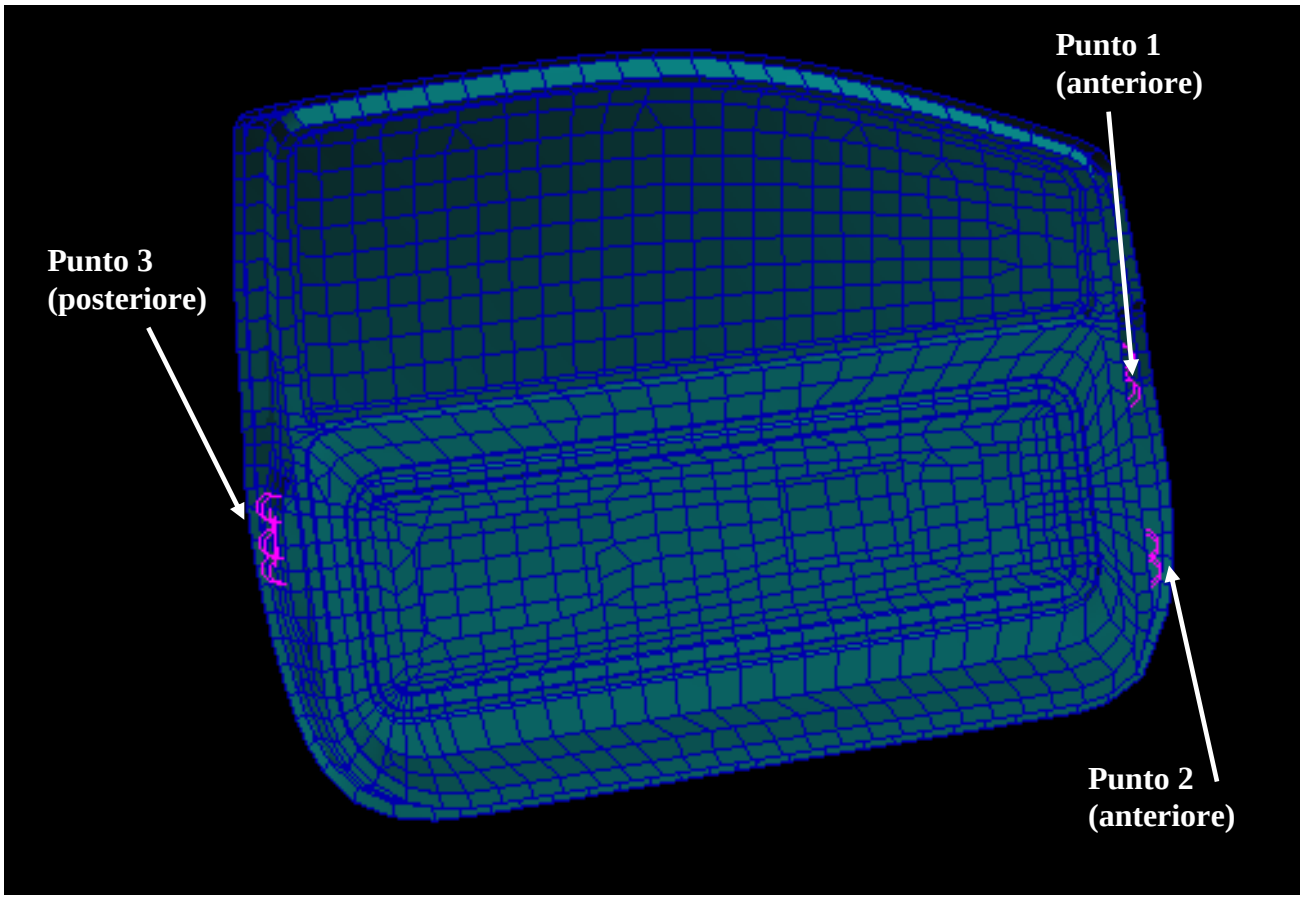
Per una trave in condizioni free-free l'equazione diventa:

$$f_n = \frac{n}{2l}\sqrt{\frac{E}{\rho}} \qquad \textbf{(9)}$$

Esercizio 4 (da portare in forma scritta all'esame)– PORTA DI AUTOMOBILE

Usando il modello di porta di auto (left_door.bdf), si richiede di:

1)Calcolare le prime 15 frequenze naturali e mostrare le relative forme modali del modello in condizioni libere
2)Vincolare il modello a telaio nei 3 punti evidenziati (anteriormente in 2 punti e posteriormente in un punto). Scegliere per ogni punto gli opportuni gradi di libertà da vincolare in base a considerazione ingegneristiche. Valutare le prime 10 frequenze naturali e mostrare le relative forme modali del modello così ottenuto.
3)Stampare i comandi principali del listato BDF.



FEM in MSC. NASTRAN

- **NASTRAN** is a finite element analysis (FEA) program that was originally developed for NASA in the late 1960s under United States government funding for the Aerospace industry.The MacNeal-Schwendler Corporation (MSC) was one of the principal and original developers of the public domain NASTRAN code. NASTRAN source code is integrated in a number of different software packages, which are distributed by a range of companies.
- NASTRAN software application was written to help design more efficient space vehicles such as the Space Shuttle.
- Nastran : **NA**sa **STR**uctural **AN**alysis

components

preprocessing:

mesh generation
material definitions
definition of loads and boundary conditions

solving:

solving the (linear) set of equations

postprocessing:

visualisation and analysis of results
(primary and secondary field variables)

displacement	stress/strain
temperature	heat flux
acoustic pressure	velocity/intensity
…	…

preprocessing

- co-ordinate systems
- nodes
- elements
- geometrical properties
- material properties
- units
- loads and constraints

illustration for MSC/NASTRAN

preprocessing

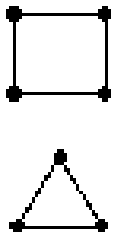
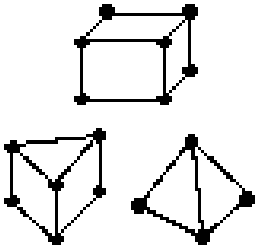
nodes

- nodes are called GRID points in NASTRAN
- grids are defined as points in space that have :
  - a unique number (integer)
  - a certain location X,Y,Z
    - coordinate systems aid in locating point
  - 6 Degrees Of Freedom (DOFs) to move in space
    - coordinate systems aid in interpreting displacement results

- GRID definition statement :
  - GRID ID CP X Y Z CD
  - where
    - ID : identification number
    - CP : reference to coordinate system that was used to position the grid
    - X,Y,Z : co-ordinates
    - CD : reference to coordinate system in which the input (loads, BC) and output (displacements) are defined

preprocessing

elements

Category	Spring Elements	Line Elements	Surface Elements	Solid Elements	Rigid Elements
Physical Behavior	Simple Spring	Rod, Bar, Beam	Membrane, Thin Plate	Thick Plate, Brick	Rigid Bar
MSC/NASTRAN Element Name	CELAS2*	CONROD* CROD CBAR	CQUAD4 CTRIA3	CHEXA CPENTA CTETRA	RBE2*
Associated Property Entry	None Required	PROD PBAR	PSHELL	PSOLID	None Required
					

preprocessing

geometry

3D Solid Elements	2D Surface Elements	1D Line Elements
<none>	Plate/Shell Thickness	Beam orientation (3th point) Beam cross section properties

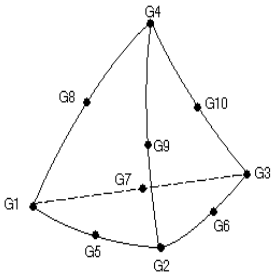
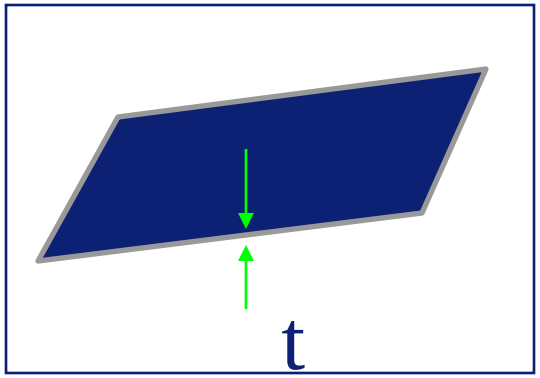
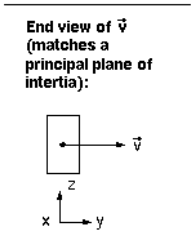
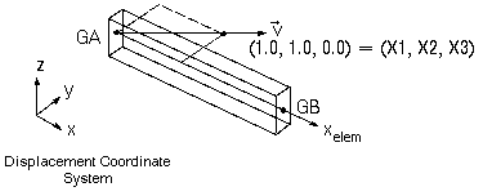


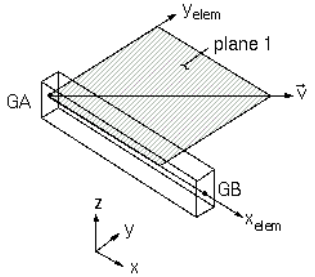
Figure 6-19. CTETRA Element Connection.







STEP 3 The plane formed by the element x-axis and orientation vector $\vec{v}$ is called plane 1. The element y-axis lies in plane 1 and is perpendicular to the element x-axis, as shown below:



preprocessing

material properties

- Basic Material Property Definitions :
  - *linear* : deformation are directly proportional to the applied load
  - *elastic* : an elastic structure returns to its original, undeformed shape when the load is removed
  - *homogeneous* : properties are independent of location within the material
  - *isotropic* : material properties do not change with the direction of the material
- MATERIAL definition statement :

  MAT1 ID E G NU RHO ... … GE
  - where
    - ID : identification number
    - E : Young's modulus
    - G : Shear modulus G = 0.5 * E / (1 + NU)
    - NU : Poisson's ratio
    - RHO : Mass density
    - GE : structural damping coefficient

preprocessing

units

- most FE solvers do not have an explicit notion of physical units.
- it is the user's responsibility to use a consistent set of units.
- popular unit sets : SI, English Units
- Common mistakes in FE models originate from wrong material values (due to wrong unit conversions) !!
- Use SI !!!

preprocessing

loads

- Static Loads :
  - concentrated loads applied to grid points (FORCE, MOMENT)
  - distributed loads on line elements (PLOAD1)
  - normal uniform pressure loads on surface (PLOAD, PLOAD2)
  - normal pressure load on face of 2D or 3D element (PLOAD4)
  - gravity or acceleration loads (GRAV)
- Enforced displacement (SPCD)

preprocessing

loads

- Dynamic Loads :
  - concentrated loads applied to grid points :

$$P(f) = A.\left[C(f) + i.D(f)\right].e^{i(\theta - 2\pi f\tau)}$$

  - RLOAD1 or RLOAD2 statement that refer to

    DAREA statements (spatial definition of load : A)

    2 TABLED1 statements (spectral definition C(f),D(f) real/imag for RLOAD1, amplitude/phase for RLOAD2)
  - Selection of dynamic loading with DLOAD case control statement (reference to RLOAD1 / 2)

preprocessing

constraints

- a constraint is the enforcement of a prescribed displacement on a single grid point or a set of points

- two basic types of constraints :
  - single point constraints (SPCs) :
    - enforces a displacement (for example zero displacement) to a single point
  - multiple point constraints (MPCs)
    - enforces a mathematical constraint relationship between one grid point and a set of grid points

solution sequence for mode calculation

solver

$$\left(\left[K\right]+ j\omega\left[C\right]-\omega^2\left[M\right]\right).\left\{X\right\}=\left\{F\right\}$$

- undamped
- no external forces

$$\left[K\right].\left\{\Phi_m\right\}=\omega_m^2\left[M\right].\left\{\Phi_m\right\}$$

ω_m : eigenfrequencies (# modes = total # dofs n)

Φ_m : eigenmodes (each eigenvector has size (nx1))

- mode calculation = standard eigenvalue problem

$$\left[M\right]^{-1}\left[K\right].\left\{\Phi_m\right\}=\lambda_m.\left\{\Phi_m\right\}$$

Lanczos algorithm :

iterative procedure to determine a subset of modes

solution sequence for dynamic response analysis

solver

$$\left(\left[K\right]+ j\omega\left[C\right]-\omega^2\left[M\right]\right).\left\{X\right\}=\left\{F\right\}$$

1. direct solution method:

- solving the FE matrix equation directly for the unknown nodal dofs
- dedicated large model solvers that fully benefit from matrix properties

2. modal solution method:

- projecting the original dofs onto a modal base
- that possibly leads to some substantial model size reduction

solution sequence for dynamic response analysis

solver

2. modal solution method:

$$\left(\left[K\right]+ j\omega\left[C\right]-\omega^2\left[M\right]\right).\left\{X\right\}=\left\{F\right\}$$

2.1. calculating the undamped modes

$$\left[K\right].\left\{\Phi_m\right\}=\omega_m^2\left[M\right].\left\{\Phi_m\right\}$$

ω_m : eigenfrequencies (# modes = total # dofs n)

Φ_m : eigenmodes (each eigenvector has size (nx1))

2.2. use the modal model for decouple the equations of motion

- accuracy depends on size of modal base m_a
- model size reduction: from *(nxn)* to *(m_axm_a)*

solution sequence for transient analysis

solver

$$\{F(t)\} = [K]\{d(t)\} + [C]\{\dot{d}(t)\} + [M]\{\ddot{d}(t)\}$$

1. direct

2. modal:

solver

- overview of some NASTRAN solution sequences:
  - SOL 101 : linear static analysis
  - SOL 103 : normal modes
  - SOL 107 / 110 : complex modes (direct/ modal)
  - SOL 108 / 111 : frequency response (direct/ modal)
  - SOL 109 / 112 : transient response (direct/ modal)
  - SOL 106 : non-linear statics followed by normal modes
  - SOL 200 : Design sensitivity and optimization

postprocessing

visualisation and analysis of results

(primary and secondary field variables)

displacement
temperature
acoustic pressure
…

stress/strain
heat flux
velocity/intensity
…

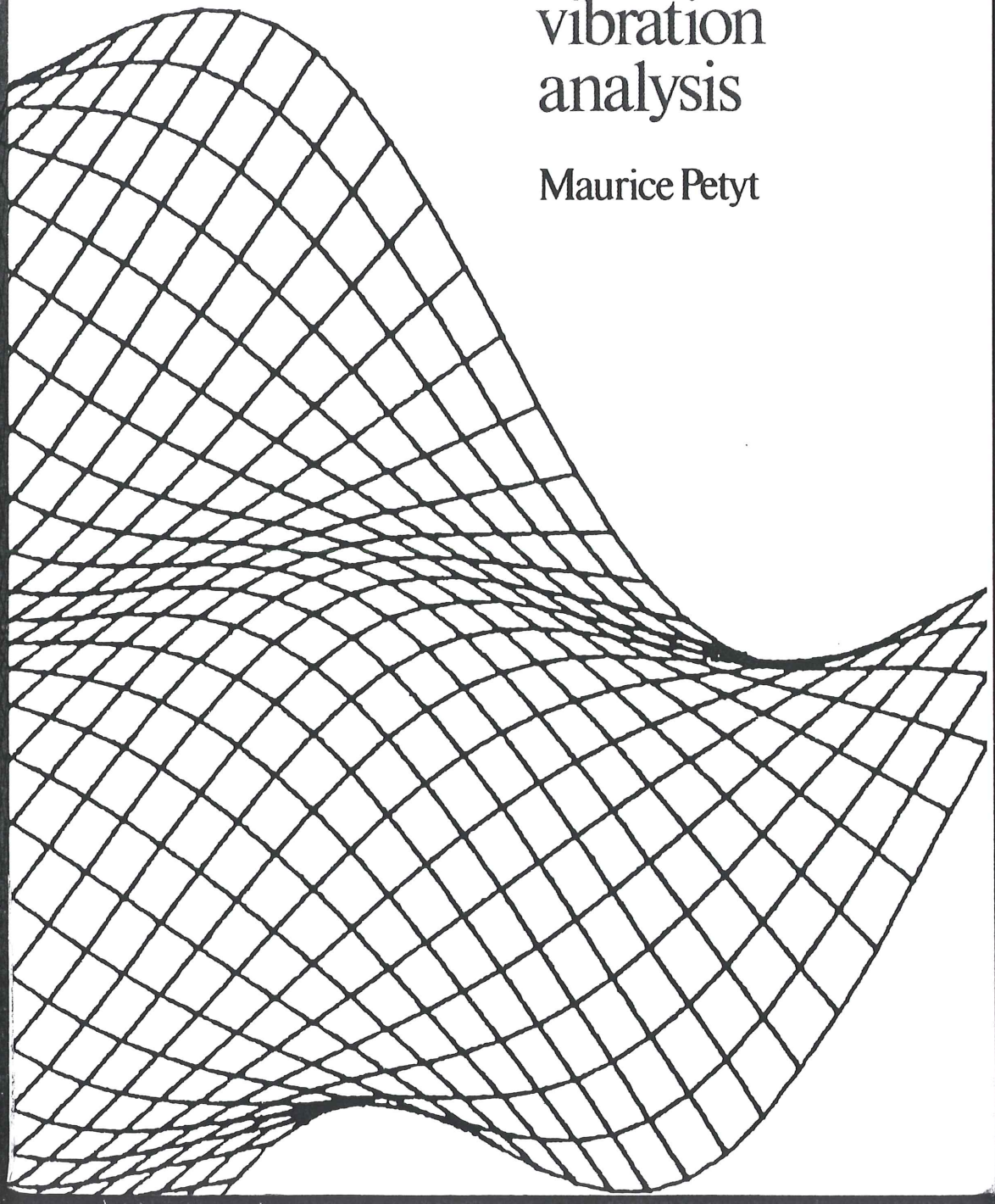
! secondary variable approximations are less accurate then primary variable approximations !

How to read a nastran file

```
$ NASTRAN input file created by the MSC MSC.Nastran input file
$ translator ( MSC.Patran 12.0.041 ) on September 21, 2005 at 10:52:43.
$ Normal Modes Analysis, Database
SOL 103
CEND
SEALL = ALL
SUPER = ALL
ECHO = NONE
SUBCASE 1
$ Subcase name : Default
   SUBTITLE=Default
   METHOD = 1


stress=all
spc=2

BEGIN BULK
PARAM    POST    0
PARAM    AUTOSPC YES
PARAM   PRTMAXIM YES
EIGRL    1                       16       0
PSOLID   1       1
$ Pset: Property_1
CTETRA   1       1       5774    2133    6428    6367    24353   24376
         24380   35839   13174   17539
CTETRA   2       1       2196    2185    1526    2172    40246   18845
--------------------------------------------------

$ Material : Material_1
MAT1*    1               6.8+10                          .3
*        2900.


$ Nodes of the Entire Model
GRID*    1                                -0.02603926542250.06873768028045*A1
*A1      0.15908825799237
GRID*    2                                -0.02605234924510.07670429191051*A2
*A2      0.14779071433393
GRID*    3                                -0.02791697484020

$ Loads for Load Case : Default
SPCADD   2       1
$ Displacement Constraints of Load Set : vincoli
SPC1     1       123456  183     199     200     201     202     203
         204     225     226     227     228
SPC1     1       123456  264     THRU    277
$ Referenced Coordinate Frames
ENDDATA
```

Introduction to finite element vibration analysis

Maurice Petyt



1

Formulation of the equations of motion

The first step in the analysis of any structural vibration problem is the formulation of the equations of motion. It is an important part of the exercise, since the success of the analysis is dependent upon the equations of motion being formulated correctly. This process will be less prone to errors if a routine procedure for formulating the equations can be established. In this chapter a number of methods will be presented and discussed.

1.1 Dynamic equilibrium

The equations of motion of any dynamic system can be written down using Newton's second law of motion, which states that 'the rate of change of momentum of a mass is equal to the force acting on it'.

Consider a mass, m, which is displaced a distance $u(t)$ when acted upon by a force $f(t)$, both being functions of time, t, as shown in Figure 1.1, then Newton's second law of motion gives

$$\frac{\mathrm{d}}{\mathrm{d}t}\left(m\frac{\mathrm{d}u}{\mathrm{d}t}\right)=f(t) \tag{1.1}$$

For constant m, which will be assumed throughout this book, equation (1.1) reduces to

$$m\frac{\mathrm{d}^2u}{\mathrm{d}t^2}=f \tag{1.2}$$

or

$$m\ddot{u}=f \tag{1.3}$$

where dots denote differentiation with respect to time.

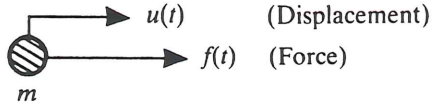


Figure 1.1 Motion of a single mass.

Equation (1.3) can be rewritten in the form

$$f - m\ddot{u} = 0 \tag{1.4}$$

If the term $-m\ddot{u}$ is now regarded as a force, then equation (1.4) represents an equation of equilibrium, that is, the sum of the forces acting on the mass is equal to zero. The introduction of this fictitious force, which is referred to as an inertia force, of magnitude $m\ddot{u}$, acting in the opposite direction to the acceleration, $\ddot{u}$, allows an equation of dynamic equilibrium to be formulated using the concepts of static equilibrium. This equation of dynamic equilibrium, when rearranged, gives the equation of motion of the system. This concept is known as d'Alembert's principle.

Example 1.1 Derive the equation of motion of the single mass, spring, damper system shown in Figure 1.2(a).

The forces acting on the mass consist of the externally applied force f, a restoring force ku due to the spring, a damping force $c\dot{u}$ due to the viscous damper and a fictitious inertia force $m\ddot{u}$. All act in the directions shown in Figure 1.2(b). For equilibrium

$$-m\ddot{u} - c\dot{u} - ku + f = 0 \tag{1.5}$$

Rearranging, gives the equation of motion

$$m\ddot{u} + c\dot{u} + ku = f \tag{1.6}$$

The above concepts can be extended to multi-degree of freedom systems. Consider a system of N masses. The equations of dynamic equilibrium are obtained by equating the sums of the forces and moments on each mass of the system to zero. This gives

$$\vec{f}_j - \frac{\mathrm{d}}{\mathrm{d}t}(m_j\dot{\vec{u}}_j) = 0 \qquad j = 1, 2, \ldots, N \tag{1.7}$$

and

$$\vec{L}_j - \frac{\mathrm{d}}{\mathrm{d}t}(\vec{J}_j) = 0 \qquad j = 1, 2, \ldots, N \tag{1.8}$$

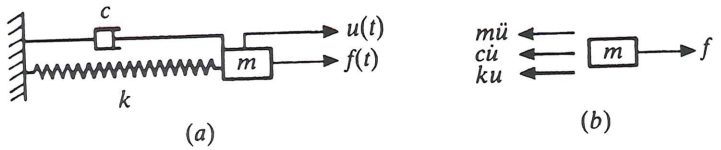


Figure 1.2 Single mass, spring, damper system.

In these equations $\vec{u}_j$ is the displacement of the mass m_j, $\vec{f}_j$ is the sum of the applied forces, $\vec{J}_j$ is the angular momentum, and $\vec{L}_j$ is the sum of the applied moments. If the vectors $\vec{u}_j$ do not represent independent motions, equations (1.7) and (1.8) must be modified by constraints of the form

$$g_j(\vec{u}_1, \vec{u}_2, \ldots, \vec{u}_N) = 0 \qquad j = 1, 2, \ldots, m \tag{1.9}$$

where m is the number of constraints. This aspect is discussed in Section 1.5.

Example 1.2 Derive the equations of motion of the system shown in Figure 1.3.

The mass m_1 has two forces acting on it due to the extension of the two springs joining it to the masses m_2 and m_3.

If the position vectors of m_1 and m_2 are $\vec{V}1$ and $\vec{V}2$ respectively, then the unit vector $\vec{n}_1$, along the line 2-1 is

$$\vec{n}_1 = \frac{1}{L_1}(\vec{V}1 - \vec{V}2) \tag{1.10}$$

where

$$L_1 = \mathrm{abs}\,(\vec{V}1 - \vec{V}2)$$

If the displacements of m_1 and m_2 are denoted by $\vec{U}_1$ and $\vec{U}_2$ then the extension, e_1, of the spring joining m_1 and m_2 is given by the scalar product

$$e_1 = (\vec{U}_1 - \vec{U}_2) \cdot \vec{n}_1 \tag{1.11}$$

If the stiffness of the spring is k_1, then the force, f_1, acting on the mass m_1 in the direction $\vec{n}_1$ is

$$f_1 = -k_1 e_1 = k_1(\vec{U}_2 - \vec{U}_1) \cdot \vec{n}_1 \tag{1.12}$$

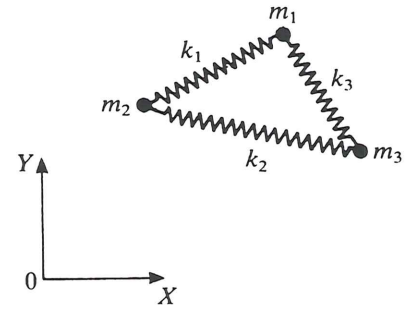


Figure 1.3 Multi-mass, spring system.

Similarly, the force, f_3, acting on the mass m_1 in the direction $\vec{n}_3$ is

$$f_3 = k_3(\vec{U}_3 - \vec{U}_1) \cdot \vec{n}_3 \tag{1.13}$$

where

$$\vec{n}_3 = \frac{1}{L_3}(\vec{V}1 - \vec{V}3) \tag{1.14}$$

and

$$L_3 = \text{abs}\,(\vec{V}1 - \vec{V}3).$$

The equation of dynamic equilibrium for m_1 is therefore

$$f_1\vec{n}_1 + f_3\vec{n}_3 - m_1\ddot{\vec{U}}_1 = 0 \tag{1.15}$$

When the components of each of the vectors are substituted in this equation, two scalar equations will be obtained. These can then be rearranged, in the manner shown in Example 1.1, to give the equations of motion of the mass m_1. The equations of motion of the masses m_2 and m_3 are obtained in a similar way.

1.2 Principle of virtual displacements

If the structure to be analysed is a complex one, then the vectoral addition of all the forces acting at each mass point is difficult. This difficulty may be overcome by first using d'Alembert's principle and then the principle of virtual displacements. By this means the equations of dynamic equilibrium and hence the equations of motion, are formulated indirectly.

The principle of virtual displacements states that 'if a system, which is in equilibrium under the action of a set of forces, is subjected to a virtual displacement, then the total work done by the forces will be zero'. In this context, a virtual displacement is a physically possible one, that is, any displacement which is compatible with the system constraints.

Example 1.3 Use the principle of virtual displacements to derive the equation of motion of the system shown in Figure 1.2.

Figure 1.2(b) shows the forces acting after the application of d'Alembert's principle. If the system is given a virtual displacement δu, then the principle of virtual displacements gives

$$-m\ddot{u}\delta u - c\dot{u}\delta u - ku\delta u + f\delta u = 0 \tag{1.16}$$

Rearranging gives

$$(-m\ddot{u} - c\dot{u} - ku + f)\delta u = 0 \tag{1.17}$$

Since δu is arbitrary and non-zero, then

$$m\ddot{u} + c\dot{u} + ku = f \tag{1.18}$$

The advantage of this approach is that the virtual work contributions are scalar quantities which can be added algebraically.

For a multi-degree of freedom system, the principle of virtual work gives

$$\sum_{j=1}^{N}\left(\vec{f}_j - \frac{\mathrm{d}}{\mathrm{d}t}(m_j\dot{\vec{u}}_j)\right)\cdot\delta\vec{u}_j + \sum_{j=1}^{N}\left(\vec{L}_j - \frac{\mathrm{d}}{\mathrm{d}t}(\vec{J}_j)\right)\cdot\delta\vec{\theta}_j = 0 \tag{1.19}$$

where the $\delta\vec{u}_j$ are virtual displacements and the $\delta\vec{\theta}_j$ virtual rotations. Since each of these is arbitrary, equations (1.7) and (1.8) must hold.

1.3 Hamilton's principle

Although the principle of virtual displacements overcomes the problem of vectorial addition of forces, virtual work itself is calculated from the scalar product of two vectors, one representing a force and one a virtual displacement. This disadvantage can be largely overcome by using Hamilton's principle to determine the equations of motion.

Consider a mass, m, which is acted upon by a force, f_T, causing a displacement, u, as shown in Figure 1.4. f_T represents the sum of all the applied forces, both conservative and non-conservative.

The work done by a conservative force in moving a mass from one point to another depends only on the position of the two points and is independent of the path taken between them. The work done by non-conservative forces does depend upon the path taken between the two points. Non-conservative forces are energy dissipating forces such as friction forces, or forces imparting energy to the system such as external forces.

The work done by a conservative force can be obtained from the change in potential energy. The potential energy $V(\vec{r})$ associated with position $\vec{r}$ is defined as the work done by a conservative force $\vec{f}$ in moving a mass from position $\vec{r}$ to a reference position $\vec{r}_0$. That is

$$V(\vec{r}) = \int_{\vec{r}}^{\vec{r}_0}\vec{f}\cdot\mathrm{d}\vec{r} \tag{1.20}$$

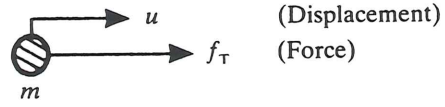


Figure 1.4 Motion of a single mass.

The work done by a conservative force $\vec{f}$ in moving a mass from position $\vec{r}_1$ to position $\vec{r}_2$, as shown in Figure 1.5, is

$$\begin{aligned} W &= \int_{\vec{r}_1}^{\vec{r}_2} \vec{f} \cdot \mathrm{d}\vec{r} \\ &= \int_{\vec{r}_1}^{\vec{r}_0} \vec{f} \cdot \mathrm{d}\vec{r} - \int_{\vec{r}_2}^{\vec{r}_0} \vec{f} \cdot \mathrm{d}\vec{r} \\ &= -\{V(\vec{r}_2) - V(\vec{r}_1)\} \end{aligned} \tag{1.21}$$

Since the force is a conservative one, the work done is independent of the path, and so in Figure 1.5 the path has been chosen to pass through the reference point 0.

Equation (1.21) states that the work done by a conservative force is minus the change in potential energy. In differential form this is

$$\delta W = -\delta V \tag{1.22}$$

The type of potential energy which will be considered in this book is the elastic potential energy, or strain energy U.

Consider a linear elastic spring of stiffness, k, which is stretched by an amount u. Then the force, f, in the spring in the direction of u is

$$f = -ku \tag{1.23}$$

and the potential energy

$$U = \int_u^0 f \, \mathrm{d}u = -\int_u^0 ku \, \mathrm{d}u = \tfrac{1}{2}ku^2 \tag{1.24}$$

Applying the principle of virtual displacements to the system in Figure 1.4 gives

$$f_T \delta u - m\ddot{u}\delta u = 0 \tag{1.25}$$

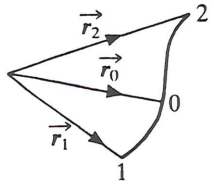


Figure 1.5 Path taken by a mass.

where δu is a virtual displacement.

$$\text{Now } f_T \delta u = \delta W = \text{work done by the forces} \tag{1.26}$$

and

$$m\ddot{u}\delta u = m\frac{\mathrm{d}}{\mathrm{d}t}(\dot{u}\delta u) - m\dot{u}\delta\dot{u} \tag{1.27}$$

where it has been assumed that

$$\frac{\mathrm{d}}{\mathrm{d}t}(\delta u) = \delta\left(\frac{\mathrm{d}u}{\mathrm{d}t}\right) = \delta\dot{u}$$

Equation (1.27) can be further modified as follows

$$\begin{aligned} m\ddot{u}\delta u &= m\frac{\mathrm{d}}{\mathrm{d}t}(\dot{u}\delta u) - \delta(\tfrac{1}{2}m\dot{u}^2) \\ &= m\frac{\mathrm{d}}{\mathrm{d}t}(\dot{u}\delta u) - \delta T \end{aligned} \tag{1.28}$$

where

$$T = \tfrac{1}{2}m\dot{u}^2 \tag{1.29}$$

represents the kinetic energy of the system.

Substituting equations (1.26) and (1.28) into equation (1.25) gives

$$\delta W - m\frac{\mathrm{d}}{\mathrm{d}t}(\dot{u}\delta u) + \delta T = 0$$

or, on rearranging

$$\delta T + \delta W = m\frac{\mathrm{d}}{\mathrm{d}t}(\dot{u}\delta u) \tag{1.30}$$

If the position of the mass is known at two instants of time t_1 and t_2, then its motion during this interval of time can be represented by a curve, as shown in Figure 1.6. A slightly different curve or path is obtained if, at any instant, a small variation in position δu is allowed with no associated change in time; that is $\delta t = 0$ (Figure 1.6). The stipulation is made, however, that at times t_1 and t_2 the two paths coincide, that is

$$\delta u = 0 \qquad \text{at } t = t_1 \text{ and } t = t_2 \tag{1.31}$$

The problem is to choose the true path from u_1 to u_2 from all the possible ones.

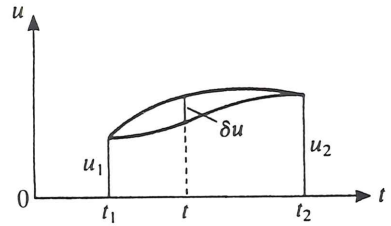


Figure 1.6 Variation in the motion of a mass.

Multiplying equation (1.30) by dt and integrating between t_1 and t_2 gives

$$\int_{t_1}^{t_2} (\delta T + \delta W)\, dt = \int_{t_1}^{t_2} m\frac{d}{dt}(\dot{u}\delta u)\, dt$$

$$= [m\dot{u}\delta u]_{t_1}^{t_2} = 0 \tag{1.32}$$

by virtue of equation (1.31). Equation (1.32), therefore, states that

$$\int_{t_1}^{t_2} (\delta T + \delta W)\, dt = 0 \tag{1.33}$$

Separating the forces into conservative and non-conservative forces, gives

$$\delta W = \delta W_c + \delta W_{nc} \tag{1.34}$$

Using equation (1.22), namely,

$$\delta W_c = -\delta V \tag{1.35}$$

equation (1.34) becomes

$$\delta W = -\delta V + \delta W_{nc} \tag{1.36}$$

Substituting equation (1.36) into equation (1.33) gives

$$\int_{t_1}^{t_2} (\delta T - \delta V + \delta W_{nc})\, dt = 0 \tag{1.37}$$

or

$$\int_{t_1}^{t_2} (\delta(T - V) + \delta W_{nc})\, dt = 0 \tag{1.38}$$

Note that equation (1.37) cannot be written in the form

$$\int_{t_1}^{t_2} \delta(T - V + W_{nc})\, dt = 0 \tag{1.39}$$

since a work function W_{nc} does not exist for non-conservative forces. However, the virtual work can always be calculated. Equation (1.38) is the mathematical statement of Hamilton's principle. For a conservative system

$\delta W_{nc} = 0$. In this case equation (1.38) shows that the integral of $(T - V)$ along the true time path is stationary. It can be shown, for the applications considered in this book, that the stationary value of the integral is a minimum.

The application of this principle leads directly to the equations of motion for any system. It can be applied to both discrete, multi-degree of freedom systems (as shown in the Appendix) and continuous systems (as illustrated in Section 2.11). The advantage of this formulation is that it uses scalar energy quantities. Vector quantities may only be required in calculating the work done by the non-conservative forces. As previously stated, the only potential energy of interest in this book is elastic strain energy U. The form of Hamilton's principle to be used is therefore

$$\int_{t_1}^{t_2} (\delta(T - U) + \delta W_{nc})\, dt = 0 \tag{1.40}$$

Example 1.4 Use Hamilton's principle to derive the equations of motion of the system shown in Figure 1.2.

For this system

$$T = \tfrac{1}{2}m\dot{u}^2$$

$$U = \tfrac{1}{2}ku^2 \tag{1.41}$$

$$\delta W_{nc} = f\delta u - c\dot{u}\delta u$$

Substituting into equation (1.40) gives

$$\int_{t_1}^{t_2} \delta(\tfrac{1}{2}m\dot{u}^2 - \tfrac{1}{2}ku^2)\, dt + \int_{t_1}^{t_2} (f\delta u - c\dot{u}\delta u)\, dt = 0 \tag{1.42}$$

that is

$$\int_{t_1}^{t_2} (m\dot{u}\delta\dot{u} - ku\delta u + f\delta u - c\dot{u}\delta u)\, dt = 0 \tag{1.43}$$

Now

$$\delta\dot{u} = \delta\left(\frac{du}{dt}\right) = \frac{d}{dt}(\delta u)$$

Hence integrating the first term by parts gives

$$\int_{t_1}^{t_2} m\dot{u}\delta\dot{u}\, dt = [m\dot{u}\delta u]_{t_1}^{t_2} - \int_{t_1}^{t_2} m\ddot{u}\delta u\, dt$$

$$= -\int_{t_1}^{t_2} m\ddot{u}\delta u\, dt \tag{1.44}$$

by virtue of equation (1.31).

Substituting equation (1.44) into equation (1.43) gives

$$\int_{t_1}^{t_2} (-m\ddot{u} - c\dot{u} - ku + f)\delta u \, dt = 0 \tag{1.45}$$

Since δu is arbitrary, equation (1.45) is satisfied only if

$$m\ddot{u} + c\dot{u} + ku = f \tag{1.46}$$

1.4 Lagrange's equations

When Hamilton's principle is applied to discrete systems it can be expressed in a more convenient form. To illustrate this, consider the system shown in Figure 1.2. The kinetic and strain energies are given by

$$T = \tfrac{1}{2}m\dot{u}^2 = T(\dot{u}), \qquad U = \tfrac{1}{2}ku^2 = U(u) \tag{1.47}$$

and the virtual work done by the non-conservative forces is

$$\delta W_{nc} = (f - c\dot{u})\delta u \tag{1.48}$$

Equation (1.40) therefore becomes

$$\int_{t_1}^{t_2} \left(\frac{\partial T}{\partial \dot{u}} \delta\dot{u} - \frac{\partial U}{\partial u} \delta u + (f - c\dot{u})\delta u \right) dt = 0 \tag{1.49}$$

Integrating the first term by parts gives

$$\begin{aligned} \int_{t_1}^{t_2} \frac{\partial T}{\partial \dot{u}} \delta\dot{u} \, dt &= \left[\frac{\partial T}{\partial \dot{u}} \delta u \right]_{t_1}^{t_2} - \int_{t_1}^{t_2} \frac{d}{dt}\left(\frac{\partial T}{\partial \dot{u}} \right) \delta u \, dt \\ &= -\int_{t_1}^{t_2} \frac{d}{dt}\left(\frac{\partial T}{\partial \dot{u}} \right) \delta u \, dt \end{aligned} \tag{1.50}$$

as a consequence of using equation (1.31).

Substituting equation (1.50) into equation (1.49) gives

$$\int_{t_1}^{t_2} \left\{ -\frac{d}{dt}\left(\frac{\partial T}{\partial \dot{u}} \right) - \frac{\partial U}{\partial u} + f - c\dot{u} \right\} \delta u \, dt = 0 \tag{1.51}$$

Since δu is arbitrary, then

$$\frac{d}{dt}\left(\frac{\partial T}{\partial \dot{u}} \right) + \frac{\partial U}{\partial u} + c\dot{u} = f \tag{1.52}$$

Introducing a dissipation function, D, which is defined by

$$D = \tfrac{1}{2}c\dot{u}^2 \tag{1.53}$$

the damping force is given by

$$c\dot{u} = \frac{\partial D}{\partial \dot{u}} \tag{1.54}$$

The dissipation function represents the instantaneous rate of energy dissipation which is given by

$\frac{1}{2}\times$ damping force $\times$ rate of extension of damper

Substituting the relationship (1.54) into equation (1.52) gives

$$\frac{d}{dt}\left(\frac{\partial T}{\partial \dot{u}} \right) + \frac{\partial D}{\partial \dot{u}} + \frac{\partial U}{\partial u} = f \tag{1.55}$$

Equation (1.55) is Lagrange's equation for a single degree of freedom system. Substituting equations (1.47) and (1.53) into equation (1.55) gives

$$m\ddot{u} + c\dot{u} + ku = f \tag{1.56}$$

which is the equation of motion of the system. It can be seen that the term $(d/dt)(\partial T/\partial \dot{u})$ gives the inertia force and $\partial U/\partial u$ the restoring force due to the spring.

In the case of a multi-degree of freedom system, the deformation of which is described by n independent displacements $q_1, q_2, \ldots, q_n$, then the kinetic energy is a function of the velocities $\dot{q}_j$ $(j = 1, 2, \ldots, n)$ only and the strain energy a function of the displacements q_j $(j = 1, 2, \ldots, n)$ only, that is

$$\begin{aligned} T &= T(\dot{q}_1, \dot{q}_2, \ldots, \dot{q}_n) \\ U &= U(q_1, q_2, \ldots, q_n) \end{aligned} \tag{1.57}$$

Similarly, the dissipation function is a function of the velocities $\dot{q}_j$, that is

$$D = D(\dot{q}_1, \dot{q}_2, \ldots, \dot{q}_n) \tag{1.58}$$

Also, the work done by the non-conservative forces can be written in the form (see Appendix)

$$\delta W_{nc} = \sum_{j=1}^{n} \left(Q_j - \frac{\partial D}{\partial \dot{q}_j} \right) \delta q_j \tag{1.59}$$

where the Q_j are generalised forces.

Lagrange's equations now take the form

$$\frac{d}{dt}\left(\frac{\partial T}{\partial \dot{q}_j} \right) + \frac{\partial D}{\partial \dot{q}_j} + \frac{\partial U}{\partial q_j} = Q_j, \qquad j = 1, 2, \ldots, n \tag{1.60}$$

These equations are derived in the Appendix.

and either

$$\theta_z = 0 \qquad \text{or} \qquad EI_z(x)\frac{\partial \theta_z}{\partial x} = 0$$

2.11 Show that the expressions for the kinetic and strain energy of a membrane element of variable thickness are

$$T = \tfrac{1}{2}\int_A \rho h(\dot{u}^2 + \dot{v}^2)\,\mathrm{d}A, \qquad U = \tfrac{1}{2}\int_A h\{\boldsymbol{\varepsilon}\}^{\mathrm{T}}[\mathbf{D}]\{\boldsymbol{\varepsilon}\}\,\mathrm{d}A$$

where $h = h(x, y)$, the other notation being defined in Section 2.5, provided the plate is sufficiently thin.

2.12 Show that the expressions for the kinetic and strain energy of a thin plate bending element of variable thickness are

$$T = \tfrac{1}{2}\int_A \rho h\dot{w}^2\,\mathrm{d}A, \qquad U = \tfrac{1}{2}\int_A \frac{h^3}{12}\{\boldsymbol{\chi}\}^{\mathrm{T}}[\mathbf{D}]\{\boldsymbol{\chi}\}\,\mathrm{d}A$$

where $h = h(x, y)$, the other notation being defined in Section 2.6.

3

Introduction to the finite element displacement method

The response of simple structures, such as uniform axial, torque and beam elements, may be obtained by solving the differential equations of motion together with the appropriate boundary conditions, as derived in Section 2.11. In many practical situations either the geometrical or material properties vary, or it may be that the shape of the boundaries cannot be described in terms of known functions. Also, practical structures consist of an assemblage of components of different types, namely, beams, plates, shells and solids. In these situations it is impossible to obtain analytical solutions to the equations of motion which satisfy the boundary conditions. This difficulty is overcome by seeking approximate solutions which satisfy Hamilton's principle (see Section 1.3).

There are a number of techniques available for determining approximate solutions to Hamilton's principle. One of the most widely used procedures is the Rayleigh–Ritz method, which is described in the next section. A generalisation of the Rayleigh–Ritz method, known as the finite element displacement method, is then introduced. The principlal features of this method are described by considering rods, shafts, beams and frameworks.

3.1 Rayleigh–Ritz method

The Rayleigh–Ritz method is first described with reference to the problem of determining the axial motion of the rod shown in Figure 3.1.

Hamilton's principle (Section 1.3) requires that

$$\int_{t_1}^{t_2} (\delta(T - U) + \delta W)\,\mathrm{d}t = 0 \tag{3.1}$$

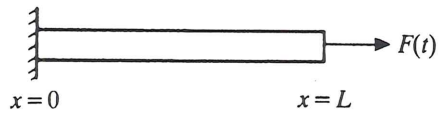


Figure 3.1 Rod subject to an axial force.

From Section 2.1 the energy functions are

$$T = \tfrac{1}{2}\int_0^L \rho A \dot{u}^2 \, dx$$

$$U = \tfrac{1}{2}\int_0^L EA\left(\frac{\partial u}{\partial x}\right)^2 dx \tag{3.2}$$

$$\delta W = F\delta u(L)$$

Since Hamilton's principle is derived using the principle of virtual displacements, then the solution $u(x, t)$ which is required is the one which satisfies both (3.1) and the geometric boundary condition

$$u(0) = 0 \tag{3.3}$$

Satisfaction of Hamilton's principle will ensure that both the equation of motion and the natural boundary condition at $x = L$ will be satisfied (see Section 2.11).

The Rayleigh-Ritz method approximates the solution with a finite expansion of the form

$$u^n(x, t) = \sum_{j=1}^{n} \phi_j(x) q_j^n(t) \tag{3.4}$$

where the $q_j^n(t)$ are unknown functions of time, t, and the $\phi_j(x)$ are prescribed functions of x, which are linearly independent. A set of functions are said to be linearly independent if

$$\sum_{j=1}^{n} \alpha_j \phi_j(x) = 0 \qquad \text{for all } x \tag{3.5}$$

implies that

$$\alpha_j = 0 \qquad \text{for } j = 1, 2, \ldots, n \tag{3.6}$$

Each of the functions $\phi_j(x)$ must satisfy the geometric boundary condition (3.3) in order to ensure that the solution, as given by equation (3.4), satisfies this condition. Therefore

$$\phi_j(0) = 0 \qquad j = 1, 2, \ldots, n \tag{3.7}$$

Since the strain energy expression (3.2) involves the first derivative of u with respect to x, then each of the functions $\phi_j(x)$ should have a finite derivative. This implies that these functions must be continuous.

A continuous deformable body, such as the rod considered here, consists of an infinity of material points, and therefore, it has infinitely many degrees of freedom. By assuming that the motion is given by the expression (3.4), the continuous system has been reduced to a system with a finite number of degrees of freedom. This has been achieved by applying the constraints

$$q_{n+1}^n = q_{n+2}^n = \cdots = 0 \tag{3.8}$$

The expression (3.4) is substituted into equation (3.1) and the $q_j^n(t)$ found. Since the system has been reduced to one with a finite number of degrees of freedom, then the application of Hamilton's principle leads to Lagrange's equations (Sections 1.4 and A2). These give, in matrix form (Section 1.4)

$$[\mathbf{M}]\{\ddot{\mathbf{q}}^n\} + [\mathbf{K}]\{\mathbf{q}^n\} = \{\mathbf{Q}^n\} \tag{3.9}$$

where

$$\{\mathbf{q}^n\}^{\mathrm{T}} = \lfloor q_1^n \quad q_2^n \quad \cdots \quad q_n^n \rfloor \tag{3.10}$$

The inertia and stiffness matrices are determined by substituting (3.4) into the kinetic and strain energy expressions (3.2) respectively. The elements of these matrices are given by

$$M_{jk} = \int_0^L \rho A \phi_j(x) \phi_k(x) \, dx$$

$$K_{jk} = \int_0^L EA \phi_j'(x) \phi_k'(x) \, dx \tag{3.11}$$

where primes denote differentiation with respect to x. The generalised forces Q_j^n are obtained by calculating the virtual work done by the applied load $F(t)$. From (3.2) and (3.4)

$$\delta W = F(t)\delta u(L) = F(t) \sum_{j=1}^{n} \phi_j(L) \delta q_j^n(t)$$

$$= \sum_{j=1}^{n} Q_j^n \delta q_j^n \tag{3.12}$$

This gives

$$Q_j^n = \phi_j(L) F(t) \tag{3.13}$$

Equation (3.9) is solved for the $\{\mathbf{q}^n\}$, which are then substituted into (3.4) to give an approximate solution for $u(x, t)$. Methods of solving equation (3.9) are described in Chapters 9 and 10.

If the integrals in (3.1) involve derivatives up to order p, then the functions $\phi_j(x)$ of equation (3.4) must satisfy the following criteria in order to ensure convergence of the solution.

(1) Be linearly independent.
(2) Be continuous and have continuous derivatives up to order $(p-1)$. In this book only the cases $p=1$ and 2 will be considered.
(3) Satisfy the geometric boundary conditions. These involve derivatives up to order $(p-1)$ (see Section 2.11).
(4) Form a complete series.

A series of functions is said to be complete if the 'mean square error' vanishes in the limit, that is

$$\lim_{n\to\infty} \int_0^L \left(u - \sum_{j=1}^{n} \phi_j q_j^n\right)^2 \mathrm{d}x = 0 \tag{3.14}$$

Polynomials (i.e., $1, x, x^2, \ldots$), trigonometric functions, Legendre, Tchebycheff and Jacobi or hypergeometric polynomials are all series of functions which are complete. An approximate solution which satisfies (3.14) is said to 'converge in the mean'.

In order to assess the convergence of the method, solutions are obtained using the sequence of functions $u^1, u^2, u^3, \ldots, u^n$. This sequence is called a minimising sequence. Using a minimising sequence ensures monotonic convergence of the solution. Using functions $\phi_j(x)$, which form a complete series, ensures monotonic convergence to the true solution.

The proof of convergence of the Rayleigh-Ritz method is based upon the proof of convergence of the expansion of an arbitrary function by means of an infinite series of linearly independent functions. If polynomials are used, then use can be made of Weierstrass's Approximation Theorem which states that: 'Any function which is continuous in the interval $a \leqslant x \leqslant b$ may be approximated uniformly by polynomials in this interval.' This theorem [3.1] asserts the possibility of uniform convergence rather than just convergence in the mean. Since the functions are required to have continuous derivatives up to order $(p-1)$, then all derivatives up to this order will converge uniformly.

These statements can be extended to functions of more than one variable. Further details are given in references [3.2-3.5]. It should be noted that in using the Rayleigh-Ritz method the equations of motion and natural boundary conditions will only be satisfied approximately.

Another problem of interest in vibration analysis is that of determining the natural frequencies and modes of free vibration of a structure. In this case $\delta W = 0$ in (3.1). The value of the integral of $(T-U)$, I^n, obtained by substituting (3.4) into it, will be greater than the true minimum because of the application of the constraints (3.8). Using the sequence of functions $u^1, u^2, u^3, \ldots, u^n$, it follows that

$$I^1 \geqslant I^2 \geqslant I^3 \geqslant \cdots \geqslant I^n \tag{3.15}$$

since the inclusion of more terms in (3.4) is equivalent to relaxing successive constraints. In this equation (3.9) reduces to

$$[\mathbf{M}]\{\ddot{\mathbf{q}}^n\} + [\mathbf{K}]\{\mathbf{q}^n\} = 0 \tag{3.16}$$

Since the motion is harmonic then

$$\{\mathbf{q}^n(t)\} = \{\mathbf{A}^n\} \sin \omega t \tag{3.17}$$

where the amplitudes $\{\mathbf{A}^n\}$ are independent of time and ω is the frequency of vibration. Substituting (3.17) into (3.16) gives

$$[\mathbf{K} - \omega^2\mathbf{M}]\{\mathbf{A}^n\} = 0 \tag{3.18}$$

Equation (3.18) represents a set of n linear homogeneous equations in the unknowns $A_1^n, A_2^n, \ldots, A_n^n$. The condition that these equations should have a non-zero solution is that the determinant of coefficients should vanish, that is

$$\det[\mathbf{K} - \omega^2\mathbf{M}] = |\mathbf{K} - \omega^2\mathbf{M}| = 0 \tag{3.19}$$

Equation (3.19) can be expanded to give a polynomial of degree n in ω^2. This polynomial equation will have n roots $\omega_1^2, \omega_2^2, \ldots, \omega_n^2$. Such roots are called 'eigenvalues'. Since $[\mathbf{M}]$ is positive definite, and $[\mathbf{K}]$ is either positive definite or positive semi-definite (see Section 1.4), the eigenvalues are all real and either positive or zero [3.6]. However, they are not necessarily all different from one another. The quantities $\omega_1, \omega_2, \ldots, \omega_n$, which are also real and either positive or zero, are approximate values of the first n natural frequencies of the system. Moreover, these approximate values will be greater than the true frequencies of the system [3.7].

Corresponding to each eigenvalue ω^2, there exists a unique solution (to within an arbitrary constant) to equation (3.18) for $\{\mathbf{A}^n\}$. These solutions are known as 'eigenvectors'. When combined with the prescribed functions $\phi_j(x)$ they define the shapes of the modes of vibration in an approximate sense. The approximate shape of a mode of vibration is given by (see equation (3.4)):

$$u^n(x) = \sum_{j=1}^{n} \phi_j(x) A_j^n \tag{3.20}$$

The solution of equation (3.18) is known as an 'eigenproblem'. Numerical methods of determining the solutions of eigenproblems, as defined by this equation, are presented in Chapter 8. These solutions, as indicated above, give approximate solutions for the natural frequencies and modes of free vibration. Convergence to the true frequencies and mode shapes is obtained as the number of terms in the approximating expression (3.4) is increased. This statement is illustrated by means of the examples below.

Example 3.1 Use the Rayleigh-Ritz method to estimate the lower frequencies and mode shapes of the clamped-free rod shown in Figure 3.1. Compare the results with the exact solution.

For free vibration, the equation of motion of the rod is (see equation (2.115))

$$EA\frac{\partial^2 u}{\partial x^2}-\rho A\frac{\partial^2 u}{\partial t^2}=0 \tag{3.21}$$

Assuming harmonic motion

$$u(x,t)=\psi(x)\sin\omega t \tag{3.22}$$

Substituting (3.22) into equation (3.21) gives

$$\frac{\mathrm{d}^2\psi}{\mathrm{d}x^2}+\omega^2\left(\frac{\rho}{E}\right)\psi=0 \tag{3.23}$$

The boundary conditions are (see Section 2.11)

$$u(0,t)=0, \qquad \frac{\partial u(L,t)}{\partial x}=0 \tag{3.24}$$

Substituting (3.22) into the boundary conditions (3.24) gives

$$\psi(0)=0, \qquad \frac{\mathrm{d}\psi(L)}{\mathrm{d}x}=0 \tag{3.25}$$

The solutions of equation (3.23) subject to the boundary conditions (3.25) are

$$\omega_r=\frac{(2r-1)\pi}{2}\left(\frac{E}{\rho L^2}\right)^{1/2}, \qquad \psi_r(x)=\sin(2r-1)\frac{\pi x}{2L} \qquad r=1,2,\ldots \tag{3.26}$$

To obtain an approximate solution using the Rayleigh-Ritz method, assume the prescribed functions in (3.4) to be

$$\phi_j(x)=x^j \tag{3.27}$$

Note that each of these satisfy the geometric boundary condition $\phi_j(0)=0$.

The elements of the stiffness and inertia matrices in equation (3.18) are, from equations (3.11)

$$K_{jk}=\int_0^L EAj\cdot k\cdot x^{j+k-2}\,\mathrm{d}x=\frac{jk}{(j+k-1)}EAL^{j+k-1}$$
$$M_{jk}=\int_0^L \rho Ax^{j+k}\,\mathrm{d}x=\frac{1}{(j+k+1)}\rho AL^{j+k+1} \tag{3.28}$$

One term solution

Using only one term in the series (3.4), equation (3.18) reduces to

$$\left(EAL-\omega^2\frac{\rho AL^3}{3}\right)A_1^1=0$$

the solution of which gives $\omega_1=1.732(E/\rho L^2)^{1/2}$.

Two term solution

Increasing the number of terms to two in the series (3.4), gives the following equation:

$$\left[EA\begin{bmatrix}L & L^2\\ L^2 & 4L^3/3\end{bmatrix}-\omega^2\rho A\begin{bmatrix}L^3/3 & L^4/4\\ L^4/4 & L^5/5\end{bmatrix}\right]\begin{bmatrix}A_1^2\\ A_2^2\end{bmatrix}=0$$

Letting $\omega^2\rho L^2/E=\lambda$, the above equation simplifies to

$$\begin{bmatrix}(1-\lambda/3) & (1-\lambda/4)\\ (1-\lambda/4) & (4/3-\lambda/5)\end{bmatrix}\begin{bmatrix}A_1^2\\ A_2^2L\end{bmatrix}=0$$

This equation has a non-zero solution provided

$$\begin{vmatrix}(1-\lambda/3) & (1-\lambda/4)\\ (1-\lambda/4) & (4/3-\lambda/5)\end{vmatrix}=0$$

Expanding gives

$$\frac{\lambda^2}{240}-\frac{13\lambda}{90}+\frac{1}{3}=0$$

The two roots of this equation are

$$\lambda=2.486 \qquad \text{and} \qquad 32.18$$

and the natural frequencies of the system are

$$\omega_1 = \lambda_1^{1/2}\left(\frac{E}{\rho L^2}\right)^{1/2} = 1.577\left(\frac{E}{\rho L^2}\right)^{1/2}$$

and

$$\omega_2 = \lambda_2^{1/2}\left(\frac{E}{\rho L^2}\right)^{1/2} = 5.673\left(\frac{E}{\rho L^2}\right)^{1/2}$$

From the homogeneous equations

$$A_2^2 = -\frac{(1-\lambda/3)}{(1-\lambda/4)L}A_1^2$$

When

$$\lambda = 2.486, \qquad A_2^2 = -0.4527\frac{A_1^2}{L}$$

$$\lambda = 32.18, \qquad A_2^2 = -1.3806\frac{A_1^2}{L}$$

The modes of vibration are therefore given by

$$u = A_1^2 L\left\{\frac{x}{L} - 0.4527\left(\frac{x}{L}\right)^2\right\}$$

and

$$u = A_1^2 L\left\{\frac{x}{L} - 1.3806\left(\frac{x}{L}\right)^2\right\}$$

The approximate values of $\omega(\rho L^2/E)^{1/2}$ are compared with the exact values in Table 3.1. As postulated, the approximate frequencies are greater than the exact ones and approach the exact ones as the number of terms is increased.

The approximate mode shapes for the two term solution are compared with the exact mode shapes in Figure 3.2. The differences between the approximate and exact shapes for the first mode are too small to show up on the scale used.

Example 3.2 Use the Rayleigh-Ritz method to estimate the lower frequencies of the cantilever beam shown in Figure 3.3. Compare the results with the exact solution.

Table 3.1. *Comparison of approximate frequencies with exact solution for a rod*

	R-R solutions		
Mode	1 term	2 term	Exact solution
1	1.732	1.577	1.571
2	—	5.673	4.712

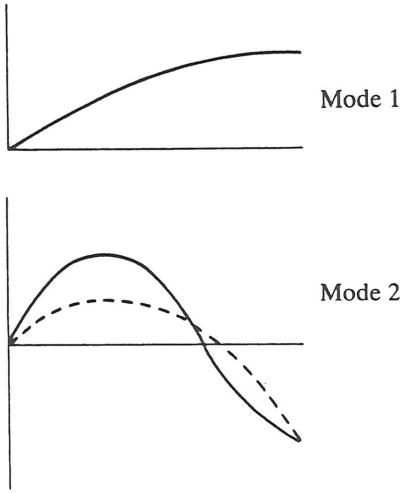


Figure 3.2 Axial modes of vibration of a rod. —— Exact; - - - approximate (R-R).

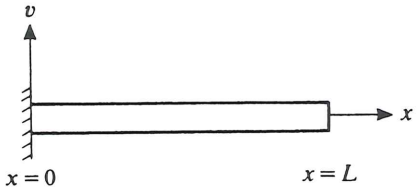


Figure 3.3 Cantilever beam.

From Section 2.11 the equation of motion of the beam is

$$EI_z \frac{\partial^4 v}{\partial x^4} + \rho A \frac{\partial^2 v}{\partial t^2} = 0 \tag{3.29}$$

and the boundary conditions are

$$v(0, t) = 0, \qquad \frac{\partial v}{\partial x}(0, t) = 0$$
$$\frac{\partial^2 v}{\partial x^2}(L, t) = 0, \qquad \frac{\partial^3 v}{\partial x^3}(L, t) = 0 \tag{3.30}$$

The solutions of equation (3.29) subject to the boundary conditions (3.30) are given by [3.8]:

$$v_r(x, t) = \psi_r(x) \sin \omega_r t$$

where

$$\omega_r = (\beta_r L)^2 \left(\frac{EI_z}{\rho A L^4}\right)^{1/2} \tag{3.31}$$

$$\psi_r(x) = \{\cosh \beta_r x - \cos \beta_r x - \eta_r(\sinh \beta_r x - \sin \beta_r x)\} \tag{3.32}$$

and

$$\eta_r = \frac{\cos \beta_r L + \cosh \beta_r L}{\sin \beta_r L + \sinh \beta_r L}$$

From Section 2.3 the energy functions are

$$T = \tfrac{1}{2} \int_0^L \rho A \dot{v}^2 \, dx$$
$$U = \tfrac{1}{2} \int_0^L EI_z \left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx \tag{3.33}$$

To obtain an approximate solution using the Rayleigh-Ritz method, assume an expansion of the form

$$v^n(x, t) = \sum_{j=1}^{n} \phi_j(x) A_j^n \sin \omega t \tag{3.34}$$

where

$$\phi_j(x) = x^{j+1}.$$

Each of the functions $\phi_j(x)$ satisfy the geometric boundary conditions at $x = 0$, that is

$$\phi_j(0) = 0, \qquad \frac{d\phi_j}{dx}(0) = 0$$

Table 3.2. *Comparison of approximate frequencies with exact solution for a beam*

	Approximate solutions		
Mode	1 term	2 term	Exact solution
1	4.472	3.533	3.516
2	—	34.807	22.035

Substituting (3.34) into the energy expressions (3.33) gives the elements of the inertia and stiffness matrices in equation (3.18), namely

$$M_{jk} = \int_0^L \rho A x^{j+k+2} \, dx = \frac{1}{(j+k+3)} \rho A L^{j+k+3}$$
$$K_{jk} = \int_0^L EI_z (j+1) j (k+1) k x^{j+k-2} \, dx$$
$$= \frac{(j+1) j (k+1) k}{(j+k-1)} EI_z L^{j+k-1} \tag{3.35}$$

The approximate values of $\omega(\rho A L^4 / EI_z)^{1/2}$ are compared with the exact solutions in Table 3.2 for various values of n.

3.2 Finite element displacement method

When analysing either structures of complex shape or built-up structures, difficulties arise in constructing a set of prescribed functions which satisfy the geometric boundary conditions. These difficulties can be overcome by using the Finite Element Displacement Method. This method provides an automatic procedure for constructing the approximating functions in the Rayleigh-Ritz method.

The prescribed functions are constructed in the following manner:

(1) Select a set of reference or 'node' points on the structure.
(2) Associate with each node point a given number of degrees of freedom (displacement, slope, etc.).
(3) Construct a set of functions such that each one gives a unit value for one degree of freedom and zero values for all the others.

This procedure is illustrated for the axial motion of a rod in Figure 3.4 and the bending vibration of a beam in Figure 3.5.

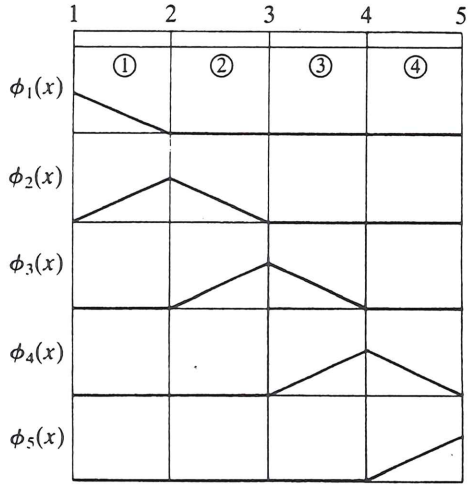


Figure 3.4 Prescribed functions for a rod.

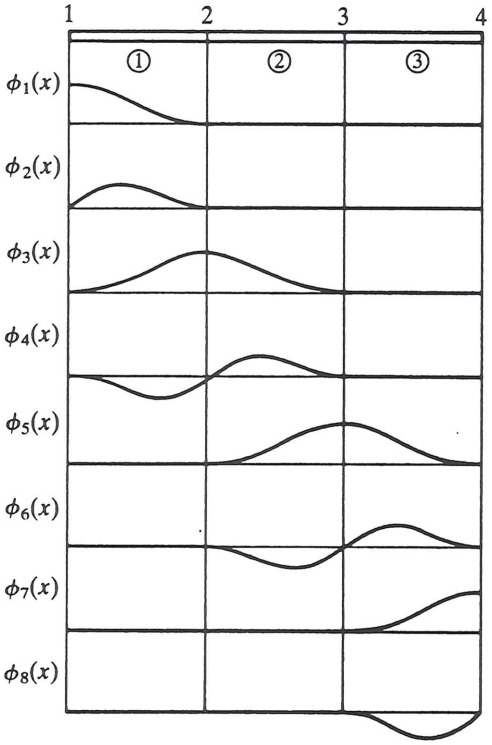


Figure 3.5 Prescribed functions for a beam.

In Figure 3.4 five node points have been selected at equal intervals. The region between each pair of adjacent nodes is referred to as an 'element'. It is shown in the previous section that only the prescribed functions themselves need be continuous for a rod. (This implies that the first derivative, which appears in the strain energy expression, can be discontinuous.) Therefore, the axial displacement, u, is the only degree of freedom required at each node point. In the figure, five prescribed functions are illustrated. They have been constructed by giving each node point in turn a unit axial displacement, whilst maintaining zero displacement at all other nodes. If these functions were to be used to analyse a clamped-free rod, then the first function, $\phi_1(x)$, would be omitted, since it does not satisfy the geometric boundary condition at $x=0$. For a clamped-clamped rod both $\phi_1(x)$ and $\phi_5(x)$ would be omitted.

In Figure 3.5, four node points have been selected at equal intervals. Thus the beam has been divided into three elements. The highest derivative appearing in the energy expressions for a beam is the second (see equations (3.33)). Therefore, the Rayleigh–Ritz procedure requires the prescribed functions and their first derivative to be continuous. Hence, it will be necessary to take v and $\partial v/\partial x$ as degrees of freedom at each node. In the figure the odd numbered prescribed functions have been constructed by giving each node point in turn a unit lateral displacement, whilst maintaining zero displacement at all other nodes. At the same time the rotations are kept zero at all nodes. The even numbered prescribed functions are constructed by giving each node in turn a unit rotation, whilst the rotations at all other nodes are kept zero. In addition, the displacements at all nodes are zero. Again, the geometric boundary conditions are satisfied by omitting the appropriate functions. For example, the functions $\phi_1(x)$ and $\phi_2(x)$ are omitted when analysing a cantilever beam.

Referring back to Figure 3.4, it can be seen that the variation of axial displacement over each element is zero except for two cases, the number being equal to the number of nodes (2) multiplied by the number of degrees of freedom at each node (1) for a single element. These two displacement variations are identical for each element. In the same way, each element of the beam in Figure 3.5 deforms in only four of the prescribed functions, being equal to the number of nodes (2) multiplied by the number of degrees of freedom at each node (2). Again the displacement variations for each element are identical. Because of this feature, it is simpler to evaluate the energy expressions for each element and then add the contributions from the elements together. This technique is illustrated in the following sections where explicit expressions for the prescribed functions over a single element are derived. These functions are referred to as 'element displacement functions'. In some texts the term 'shape function' is used, but it will not be used here.

In order to satisfy the convergence criteria of the Rayleigh-Ritz method, the element displacement functions should satisfy the following conditions:

(1) Be linearly independent.
(2) Be continuous and have continuous derivatives up to order $(p-1)$ both within the element and across element boundaries. An element which satisfies this condition is referred to as a 'conforming' element.
(3) If polynomial functions are used, then they must be complete polynomials of at least degree p. If any terms of degree greater than p are used, they need not be complete. (A complete polynomial of degree n in m variables has $(n+m)!/n!m!$ independent terms.) However, the rate of convergence is governed by the order of completeness of the polynomial. The element displacement functions need not be polynomials, but this possibility is not considered in this book.
(4) Satisfy the geometric boundary conditions.

In the Rayleigh-Ritz method, convergence is obtained as the number of prescribed functions is increased. To increase the number of prescribed functions in the finite element method, the number of node points, and therefore the number of elements, is increased. A complete discussion of the convergence of the finite element method is given in reference [3.9].

3.3 Axial vibration of rods

There are a number of ways of determining the displacement functions of a single element. The most common of these are as follows:

(1) By inspection.
(2) Assume a polynomial function having the appropriate number of terms. Then evaluate it and, if necessary, its derivatives at the nodes to obtain the coefficients in terms of the nodal degrees of freedom.
(3) Solve the equations of static equilibrium to determine the deformation of the element due to prescribed boundary displacements.

In practice the most appropriate method is used for each type of element. All three procedures are now illustrated using an axial element.

It is shown in Figure 3.4 that the deformation of an axial element is given by the combination of two linear functions. Using the non-dimensional coordinate $\xi = x/a$, defined in Figure 3.6, it is easily seen that the displacement variation for such an element is given by

$$u = \tfrac{1}{2}(1-\xi)u_1 + \tfrac{1}{2}(1+\xi)u_2 \tag{3.36}$$

where u_1, u_2 are the axial displacements of nodes 1 and 2.

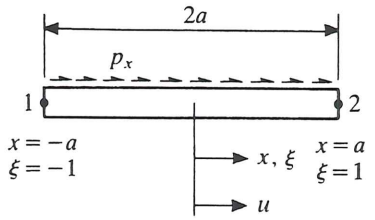


Figure 3.6 Geometry of a single axial element.

Alternatively, since the element has 2 nodes and 1 degree of freedom at each node, the displacement variation can be represented by a polynomial having 2 constants, namely

$$u = \alpha_1 + \alpha_2 \xi \tag{3.37}$$

Note that the highest derivative which occurs in the energy expressions is the first (see equations (2.11) to (2.13)), and so a polynomial of at least degree one must be used to satisfy the convergence criteria.

Evaluating (3.37) at $\xi = \mp 1$ gives

$$u_1 = \alpha_1 - \alpha_2, \qquad u_2 = \alpha_1 + \alpha_2 \tag{3.38}$$

Solving for α_1 and α_2 gives

$$\alpha_1 = \tfrac{1}{2}(u_1+u_2), \qquad \alpha_2 = \tfrac{1}{2}(u_2-u_1) \tag{3.39}$$

Substituting (3.39) into (3.37) gives

$$\begin{aligned} u &= \tfrac{1}{2}(u_1+u_2) + \tfrac{1}{2}(u_2-u_1)\xi \\ &= \tfrac{1}{2}(1-\xi)u_1 + \tfrac{1}{2}(1+\xi)u_2 \end{aligned} \tag{3.40}$$

An expression which is identical to (3.36) has therefore been obtained.

The equation of static equilibrium for the element can be deduced from equation (2.115) to be

$$\frac{d^2u}{dx^2} = 0 \tag{3.41}$$

(It is assumed that there is no distributed loading, only end forces necessary to sustain prescribed displacements.) Changing to the non-dimensional coordinate ξ gives

$$\frac{d^2u}{d\xi^2} = 0 \tag{3.42}$$

The general solution of this equation is

$$u = \alpha_1 + \alpha_2 \xi. \tag{3.43}$$

Substituting for $\partial\theta_x/\partial x$ in equation (3.104) from (3.107) gives

$$\begin{bmatrix}\tau_{xy}\\ \tau_{xz}\end{bmatrix}=\begin{bmatrix}\partial\psi/\partial y-z\\ \partial\psi/\partial z+y\end{bmatrix}\frac{M_x}{J} \tag{3.108}$$

The twisting moments at the nodes are obtained by substituting the element energy expressions (3.98) to (3.100) into Lagrange's equations. This gives

$$\begin{bmatrix}M_x(-1)\\ M_x(+1)\end{bmatrix}=\frac{GJ}{2a}\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\{\boldsymbol{\theta}\}_e + \frac{\rho I_x a}{3}\begin{bmatrix}2 & 1\\ 1 & 2\end{bmatrix}\{\ddot{\boldsymbol{\theta}}\}_e - m_x^e a\begin{bmatrix}1\\ 1\end{bmatrix} \tag{3.109}$$

The twisting moment at any section can be obtained by considering equilibrium of part of the element between -1 and ξ.

$$M_x(\xi)=-M_x(-1)+\rho I_x a\int_{-1}^{\xi}\ddot{\theta}_x(\xi)\,\mathrm{d}\xi - a\int_{-1}^{\xi}m_x(\xi)\,\mathrm{d}\xi \tag{3.110}$$

Substituting for θ_x from (3.95) and assuming m_x is constant gives

$$M_x(\xi)=-M_x(-1)+\rho I_x a\left[\tfrac{1}{2}(\xi-\tfrac{1}{2}\xi^2+\tfrac{3}{2}) \quad \tfrac{1}{2}(\xi+\tfrac{1}{2}\xi^2+\tfrac{1}{2})\right]\{\ddot{\boldsymbol{\theta}}\}_e - m_x^e a(\xi+1) \tag{3.111}$$

The shear stresses are given by equations (3.108) and (3.111) combined.

3.5 Bending vibration of beams

It is shown in Section 3.2 that it is necessary to take v and $\partial v/\partial x$ as degrees of freedom at each node of a beam element. Therefore the element shown in Figure 3.14, which has two nodes, has a total of four degrees of freedom. The displacement function can thus be represented by a polynomial having four constants, namely

$$v=\alpha_1+\alpha_2\xi+\alpha_3\xi^2+\alpha_4\xi^3 \tag{3.112}$$

This expression can be written in the following matrix form

$$v=\begin{bmatrix}1 & \xi & \xi^2 & \xi^3\end{bmatrix}\begin{bmatrix}\alpha_1\\ \alpha_2\\ \alpha_3\\ \alpha_4\end{bmatrix} \tag{3.113}$$

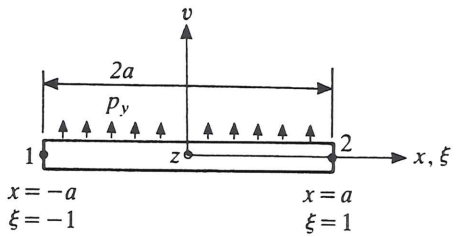


Figure 3.14 Geometry of a single beam element.

or

$$v=\lfloor\mathbf{P}(\xi)\rfloor\{\boldsymbol{\alpha}\} \tag{3.114}$$

Differentiating (3.112) gives

$$a\theta_z=a\frac{\partial v}{\partial x}=\frac{\partial v}{\partial\xi}=\alpha_2+2\alpha_3\xi+3\alpha_4\xi^2 \tag{3.115}$$

Evaluating (3.112) and (3.115) at $\xi=\mp1$ gives

$$\begin{bmatrix}v_1\\ a\theta_{z1}\\ v_2\\ a\theta_{z2}\end{bmatrix}=\begin{bmatrix}1 & -1 & 1 & -1\\ 0 & 1 & -2 & 3\\ 1 & 1 & 1 & 1\\ 0 & 1 & 2 & 3\end{bmatrix}\begin{bmatrix}\alpha_1\\ \alpha_2\\ \alpha_3\\ \alpha_4\end{bmatrix} \tag{3.116}$$

or

$$\{\bar{\mathbf{v}}\}_e=[\mathbf{A}]_e\{\boldsymbol{\alpha}\}. \tag{3.117}$$

Solving for $\{\boldsymbol{\alpha}\}$ gives

$$\{\boldsymbol{\alpha}\}=[\mathbf{A}]_e^{-1}\{\bar{\mathbf{v}}\}_e \tag{3.118}$$

where

$$[\mathbf{A}]_e^{-1}=\frac{1}{4}\begin{bmatrix}2 & 1 & 2 & -1\\ -3 & -1 & 3 & -1\\ 0 & -1 & 0 & 1\\ 1 & 1 & -1 & 1\end{bmatrix} \tag{3.119}$$

Equation (3.118) can be written in the alternative form

$$\{\boldsymbol{\alpha}\}=[\mathbf{C}]_e\{\mathbf{v}\}_e \tag{3.120}$$

where

$$\{\mathbf{v}\}_e^{\mathrm{T}}=\lfloor v_1 \quad \theta_{z1} \quad v_2 \quad \theta_{z2}\rfloor \tag{3.121}$$

and

$$[\mathbf{C}]_e = \frac{1}{4}\begin{bmatrix} 2 & a & 2 & -a \\ -3 & -a & 3 & -a \\ 0 & -a & 0 & a \\ 1 & a & -1 & a \end{bmatrix} \tag{3.122}$$

Substituting (3.120) into (3.114) gives

$$v = \lfloor \mathbf{P}(\xi) \rfloor [\mathbf{C}]_e \{\mathbf{v}\}_e \tag{3.123}$$

This can be expressed in the form

$$v = \lfloor \mathbf{N}(\xi) \rfloor \{\mathbf{v}\}_e \tag{3.124}$$

where

$$\lfloor \mathbf{N}(\xi) \rfloor = \lfloor N_1(\xi) \quad aN_2(\xi) \quad N_3(\xi) \quad aN_4(\xi) \rfloor \tag{3.125}$$

The displacement functions in (3.125) are given by

$$\begin{aligned} N_1(\xi) &= \tfrac{1}{4}(2 - 3\xi + \xi^3) \\ N_2(\xi) &= \tfrac{1}{4}(1 - \xi - \xi^2 + \xi^3) \\ N_3(\xi) &= \tfrac{1}{4}(2 + 3\xi - \xi^3) \\ N_4(\xi) &= \tfrac{1}{4}(-1 - \xi + \xi^2 + \xi^3) \end{aligned} \tag{3.126}$$

The energy expressions for the single element shown in Figure 3.14 are from Section 2.3

$$T_e = \tfrac{1}{2}\int_{-a}^{+a} \rho A \dot{v}^2 \, dx \tag{3.127}$$

$$U_e = \tfrac{1}{2}\int_{-a}^{+a} EI_z \left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx \tag{3.128}$$

$$\delta W_e = \int_{-a}^{+a} p_y \delta v \, dx \tag{3.129}$$

Substituting the displacement expression (3.124) into the kinetic energy (3.127) gives

$$\begin{aligned} T_e &= \tfrac{1}{2}\int_{-a}^{+a} \rho A \dot{v}^2 \, dx = \tfrac{1}{2}\int_{-1}^{+1} \rho A \dot{v}^2 a \, d\xi \\ &= \tfrac{1}{2}\{\dot{\mathbf{v}}\}_e^{\mathrm{T}} \rho A a \int_{-1}^{+1} \lfloor \mathbf{N}(\xi) \rfloor^{\mathrm{T}} \lfloor \mathbf{N}(\xi) \rfloor \, d\xi \{\dot{\mathbf{v}}\}_e \end{aligned} \tag{3.130}$$

Therefore the element inertia matrix is given by

$$[\mathbf{m}]_e = \rho A a \int_{-1}^{+1} \lfloor \mathbf{N}(\xi) \rfloor^{\mathrm{T}} \lfloor \mathbf{N}(\xi) \rfloor \, d\xi \tag{3.131}$$

Substituting for the functions $N_j(\xi)$ from (3.126) and integrating gives

$$[\mathbf{m}]_e = \frac{\rho A a}{105}\begin{bmatrix} 78 & 22a & 27 & -13a \\ 22a & 8a^2 & 13a & -6a^2 \\ 27 & 13a & 78 & -22a \\ -13a & -6a^2 & -22a & 8a^2 \end{bmatrix} \tag{3.132}$$

In deriving this result, it is simpler to use the expression (3.123) for the displacement v. This approach requires the integral $\int_{-1}^{+1} \lfloor \mathbf{P}(\xi) \rfloor^{\mathrm{T}} \lfloor \mathbf{P}(\xi) \rfloor \, d\xi$ to be evaluated, which is much simpler than the expression (3.131).

Substituting the displacement expression (3.124) into the strain energy (3.128) gives

$$\begin{aligned} U_e &= \tfrac{1}{2}\int_{-a}^{+a} EI_z \left(\frac{\partial^2 v}{\partial x^2}\right)^2 dx = \tfrac{1}{2}\int_{-1}^{+1} EI_z \frac{1}{a^4}\left(\frac{\partial^2 v}{\partial \xi^2}\right)^2 a \, d\xi \\ &= \tfrac{1}{2}\{\mathbf{v}\}_e^{\mathrm{T}} \frac{EI_z}{a^3}\int_{-1}^{+1} \lfloor \mathbf{N}''(\xi) \rfloor^{\mathrm{T}} \lfloor \mathbf{N}''(\xi) \rfloor \, d\xi \{\mathbf{v}\}_e \end{aligned} \tag{3.133}$$

The element stiffness matrix is therefore

$$[\mathbf{k}]_e = \frac{EI_z}{a^3}\int_{-1}^{+1} \lfloor \mathbf{N}''(\xi) \rfloor^{\mathrm{T}} \lfloor \mathbf{N}''(\xi) \rfloor \, d\xi \tag{3.134}$$

Substituting for the functions $N_j(\xi)$ from (3.126) and integrating gives

$$[\mathbf{k}]_e = \frac{EI_z}{2a^3}\begin{bmatrix} 3 & 3a & -3 & 3a \\ 3a & 4a^2 & -3a & 2a^2 \\ -3 & -3a & 3 & -3a \\ 3a & 2a^2 & -3a & 4a^2 \end{bmatrix} \tag{3.135}$$

The virtual work done by the distributed forces becomes, after substituting (3.124) into (3.129)

$$\begin{aligned} \delta W_e &= \int_{-a}^{+a} p_y \delta v \, dx = \int_{-1}^{+1} p_y \delta v \, a \, d\xi \\ &= \{\delta \mathbf{v}\}_e^{\mathrm{T}} a \int_{-1}^{+1} p_y \lfloor \mathbf{N}(\xi) \rfloor^{\mathrm{T}} \, d\xi \end{aligned} \tag{3.136}$$

The element load matrix is therefore

$$\{\mathbf{f}\}_e = a\int_{-1}^{+1} p_y [\mathbf{N}(\xi)]^{\mathrm{T}}\, \mathrm{d}\xi \tag{3.137}$$

Substituting for the functions $N_j(\xi)$ from (3.126) and assuming p_y to have the constant value p_y^e over the element gives

$$\{\mathbf{f}\}_e = p_y^e \frac{a}{3}\begin{bmatrix} 3 \\ a \\ 3 \\ -a \end{bmatrix} \tag{3.138}$$

The assembly process for a beam element is similar to that of an axial element. For element e with nodes e and $(e+1)$, the four rows and columns of the inertia matrix (3.132) are added into rows and columns $(2e-1)$ to $(2e+2)$ of the inertia matrix for the complete beam. The stiffness matrix is treated in the same way. The four rows of the element load matrix are added into rows $(2e-1)$ to $(2e+2)$ of the assembled load matrix.

Example 3.7 Use the finite element displacement method to estimate the lower frequencies of the cantilever beam shown in Figure 3.3. Compare the results with the exact solution.

One element solution

The kinetic and strain energies of a beam of length, L, which is represented by a single element, are given by the expressions (3.130) to (3.135) with $a = L/2$. Imposing the conditions that $v_1 = \theta_{z1} = 0$ and substituting into Lagrange's equations gives the equation of free vibration

$$\left[\frac{EI_z}{L^3}\begin{bmatrix} 12 & -6L \\ -6L & 4L^2 \end{bmatrix} - \omega^2 \frac{\rho AL}{210}\begin{bmatrix} 78 & -11L \\ -11L & 2L^2 \end{bmatrix}\right]\begin{bmatrix} v_2 \\ \theta_{z2} \end{bmatrix} = 0$$

Letting $(\omega^2 \rho AL^4/210EI_z) = \lambda$, this equation simplifies to

$$\begin{bmatrix} (12-78\lambda) & (-6+11\lambda) \\ (-6+11\lambda) & (4-2\lambda) \end{bmatrix}\begin{bmatrix} v_2 \\ L\theta_{z2} \end{bmatrix} = 0$$

This equation has a non-zero solution provided

$$\begin{vmatrix} (12-78\lambda) & (-6+11\lambda) \\ (-6+11\lambda) & (4-2\lambda) \end{vmatrix} = 0$$

Expanding gives

$$35\lambda^2 - 204\lambda + 12 = 0$$

The two roots of this equation are

$$\lambda = 0.0594295 \quad \text{and} \quad 5.76914$$

and the natural frequencies of the system are

$$\omega_1 = (210\lambda_1)^{1/2}\left(\frac{EI_z}{\rho AL^4}\right)^{1/2} = 3.533\left(\frac{EI_z}{\rho AL^4}\right)^{1/2}$$

and

$$\omega_2 = (210\lambda_2)^{1/2}\left(\frac{EI_z}{\rho AL^4}\right)^{1/2} = 34.807\left(\frac{EI_z}{\rho AL^4}\right)^{1/2}$$

Table 3.2 shows that the values of the coefficient for these two frequencies should be 3.516 and 22.035 respectively. The errors produced by a one element solution are therefore 0.48 and 58% respectively.

Repeating the analysis using two, three and four elements gives the errors shown in Figure 3.15 when compared with the exact solution. Notice that the convergence in this case is better than that obtained for the rod (Figure 3.9). This is in keeping with the observation made in reference [3.13] that the convergence of the Rayleigh-Ritz method is improved if the order of

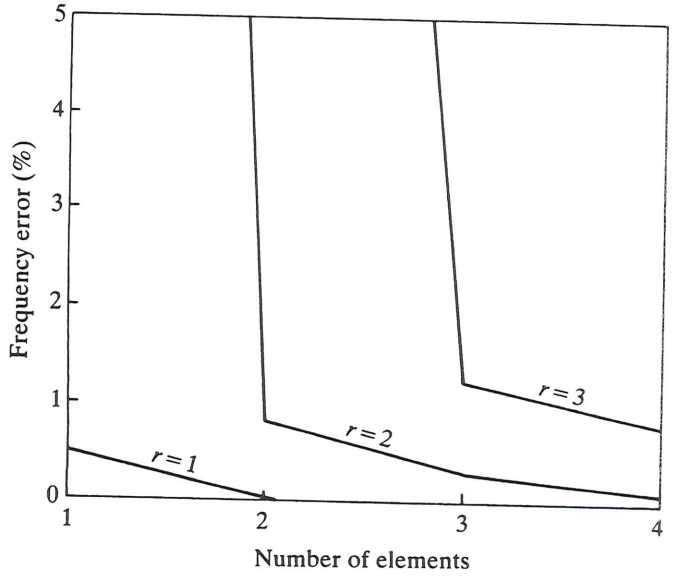


Figure 3.15 Flexural vibration of a cantilever beam.

the derivatives in the energy expressions is higher. Results for a variety of boundary conditions are presented in reference [3.14].

The shear force and bending moment at the two nodes are obtained by substituting the element energy expressions (3.130) to (3.138) into Lagrange's equations. This gives

$$\begin{bmatrix} Q(-1) \\ M_z(-1) \\ Q(+1) \\ M_z(+1) \end{bmatrix} = [\mathbf{k}]_e\{\mathbf{v}\}_e + [\mathbf{m}]_e\{\ddot{\mathbf{v}}\}_e - \{\mathbf{f}\}_e \tag{3.139}$$

where $[\mathbf{k}]_e$, $[\mathbf{m}]_e$, $\{\mathbf{f}\}_e$ and $\{\mathbf{v}\}_e$ are defined by equations (3.135), (3.132), (3.138) and (3.121) respectively.

The shear force and bending moment at any section can be obtained by considering equilibrium of the part of the element between -1 and ξ. This gives

$$Q(\xi) = Q(-1) + \rho A a \int_{-1}^{\xi} \ddot{v}(\xi_1)\,\mathrm{d}\xi_1 - a\int_{-1}^{\xi} p_y(\xi_1)\,\mathrm{d}\xi_1 \tag{3.140}$$

$$M(\xi) = M(-1) - Q(-1)a(1+\xi) - \rho A a^2 \int_{-1}^{\xi} \ddot{v}(\xi_1)(\xi - \xi_1)\,\mathrm{d}\xi_1 + a^2\int_{-1}^{\xi} p_y(\xi_1)(\xi - \xi_1)\,\mathrm{d}\xi_1 \tag{3.141}$$

The integrals are evaluated after substituting for v from (3.124). Some applications of this method can be found in reference [3.15].

The distribution of the direct stress component, σ_x, over a cross-section can be calculated using a combination of equations (2.126) and (2.127), namely

$$\sigma_x = -yM_z/I_z \tag{3.142}$$

The method of determining the distribution of shear stress depends upon the shape of the cross-section [3.16].

3.6 Vibration of plane frameworks

Consider a plane framework, such as the one shown in Figure 3.16, which is vibrating in its own plane. It can be seen that the framework consists of members which are inclined to one another at various angles. When applying

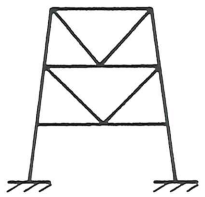


Figure 3.16 Example of a plane framework.

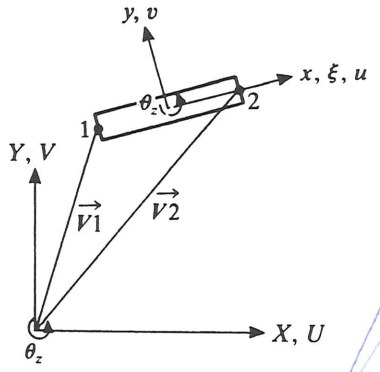


Figure 3.17 Geometry of a plane framework element.

the finite element method to such a structure, the following procedure is used:

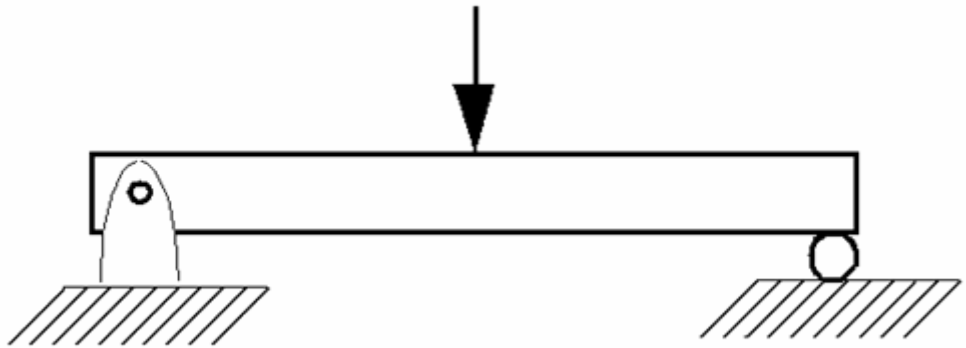
(1) Divide each member into the appropriate number of elements.
(2) Derive the energy expressions for each element in terms of nodal degrees of freedom relative to a 'local' set of axes.
(3) Transform the energy expressions for each element into expressions involving nodal degrees of freedom relative to a common set of 'global' axes.
(4) Add the energies of the elements together.

Figure 3.17 shows a typical element together with its local axes x and y which are inclined to the global axes X and Y. The local axis of x lies along the centroidal axis which joins nodes 1 and 2. The local y-axis is perpendicular to the x-axis and passes through the mid-point of the line joining 1 and 2.

Each member of a plane framework is capable of both axial and bending deformations. Therefore the energy functions for an element are a combination of the energy functions derived in Sections 2.1 and 2.3. These are, in

WORKSHOP 2

SIMPLY SUPPORTED BEAM

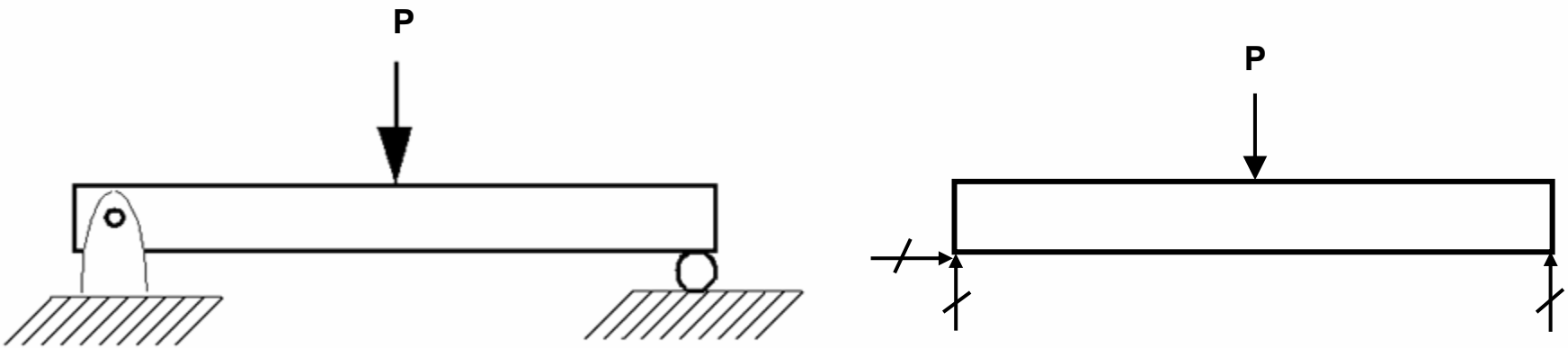


Workshop Objectives

- A finite element model must be properly constrained to prevent rigid body motion. This workshop demonstrates what happens when a model is not adequately constrained.

Problem Description

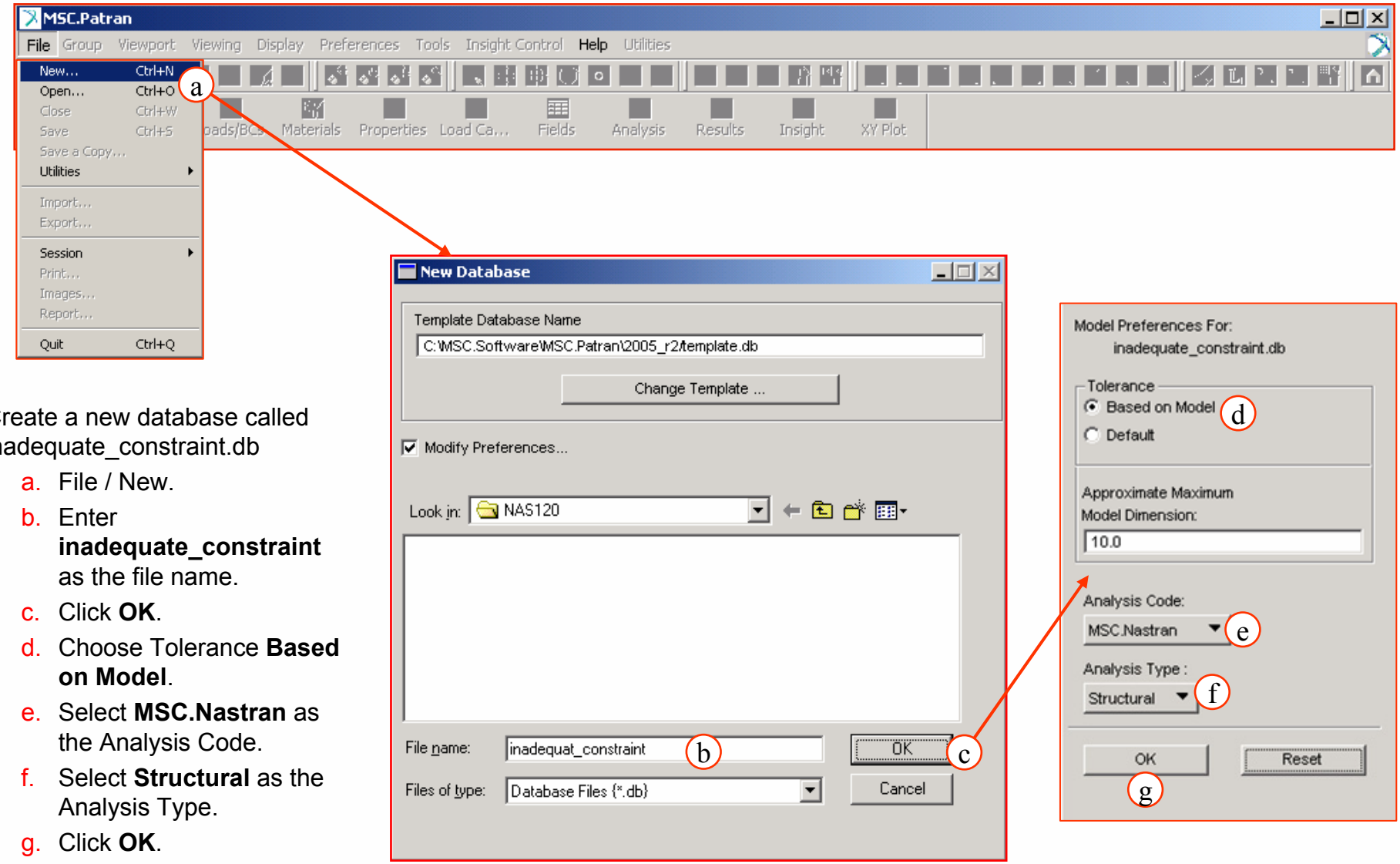
- Analyze a simply-supported beam with a concentrated load
- Beam dimension 1” x 1” x 12”
- E = 30 x 10^6 psi
- ν =0.3
- Load = 200 lb



- **Suggested Exercise Steps**

1. Create a new database and name it inadequate_constraint.db.
2. Create a solid to represent the beam.
3. Mesh the solid to create 3D elements.
4. Create in-plane boundary conditions.
5. Apply loads.
6. Create material properties.
7. Create physical properties.
8. Run analysis with MSC.Nastran.
9. View fatal errors in the .f06 file.
10. Add new boundary condition to properly constrain model.
11. Re-run the analysis. View the .f06 file.
12. Access the results file.
13. Plot results.

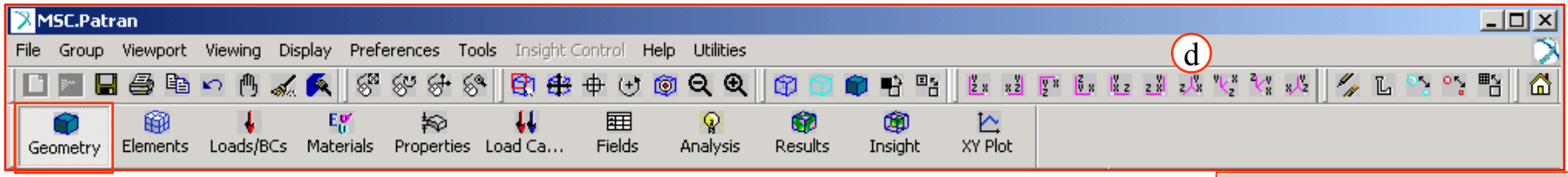
Step 1. Create New Database



Create a new database called inadequate_constraint.db

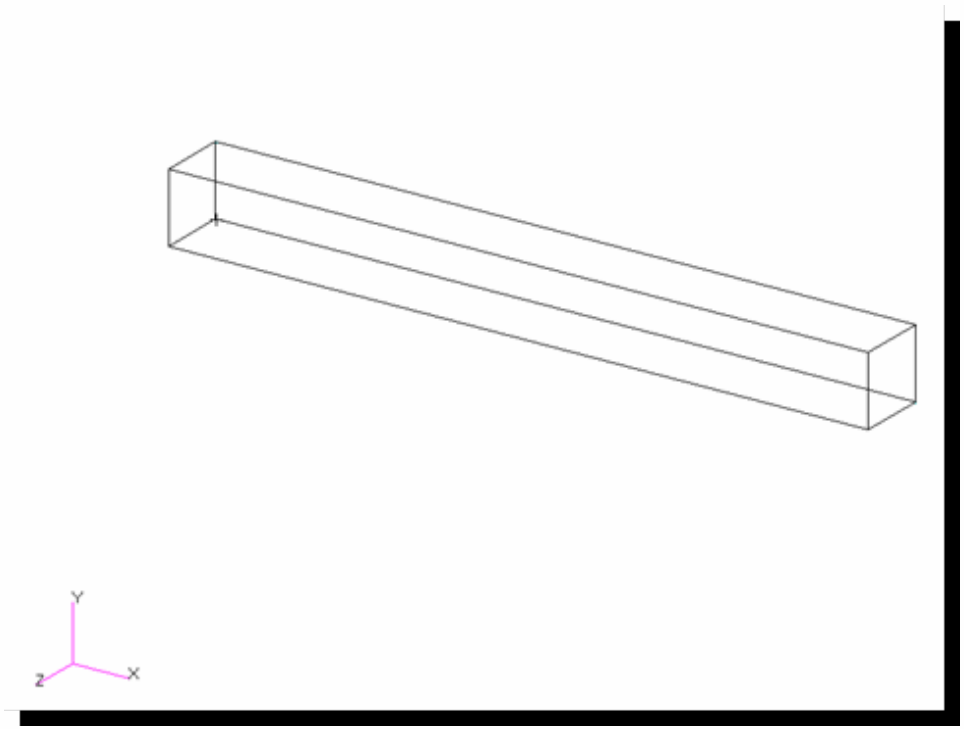
a. File / New.
b. Enter **inadequate_constraint** as the file name.
c. Click **OK**.
d. Choose Tolerance **Based on Model**.
e. Select **MSC.Nastran** as the Analysis Code.
f. Select **Structural** as the Analysis Type.
g. Click **OK**.

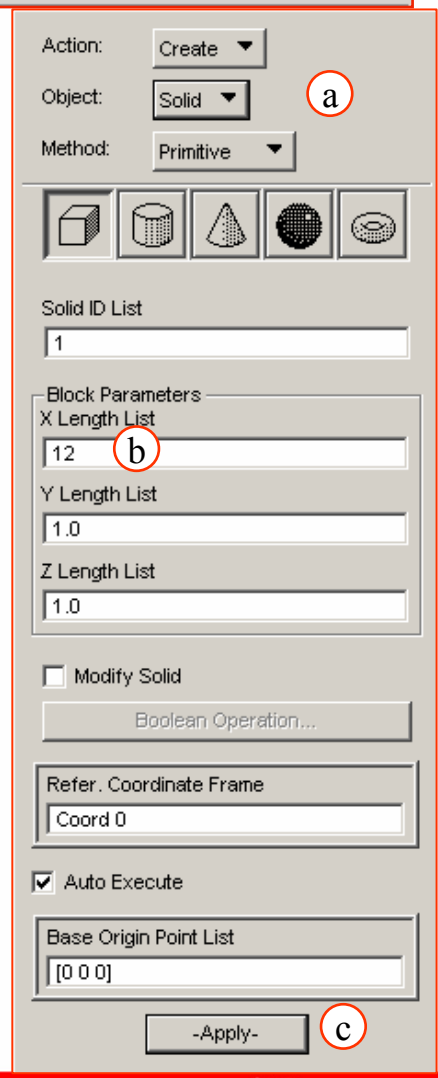
Step 2. Create Geometry



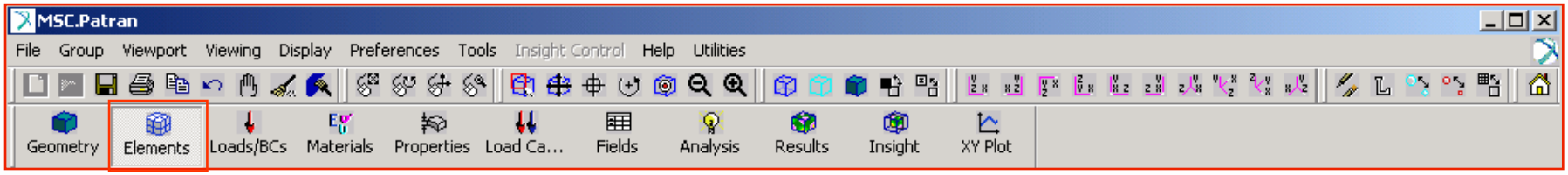
Create a solid

a. Geometry: Create / Solid / Primitive
b. Enter **12** for the X Length
c. Click **Apply**.
d. Change to iso 1 view



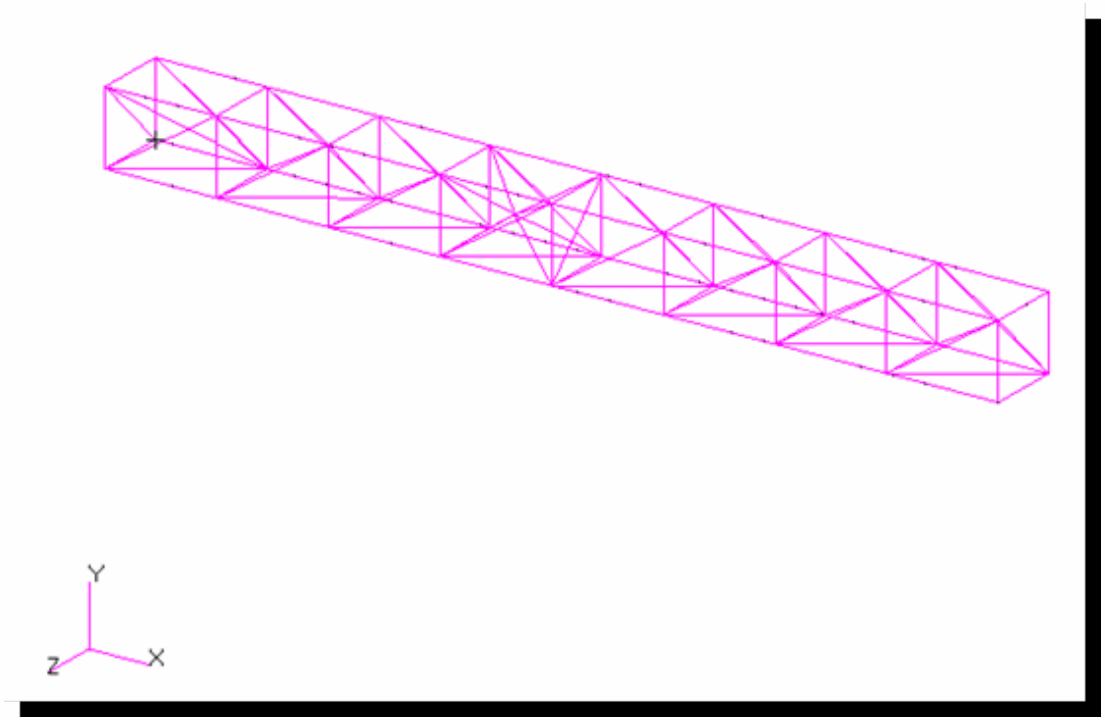


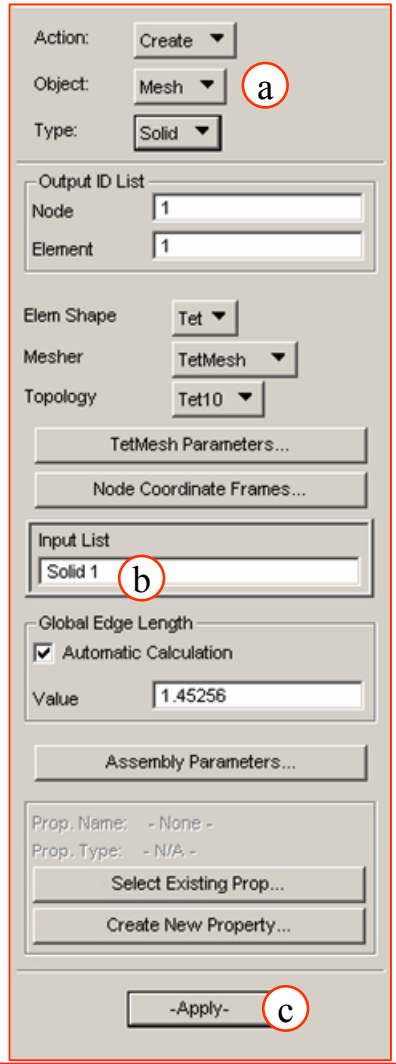
Step 3. Mesh the Solid



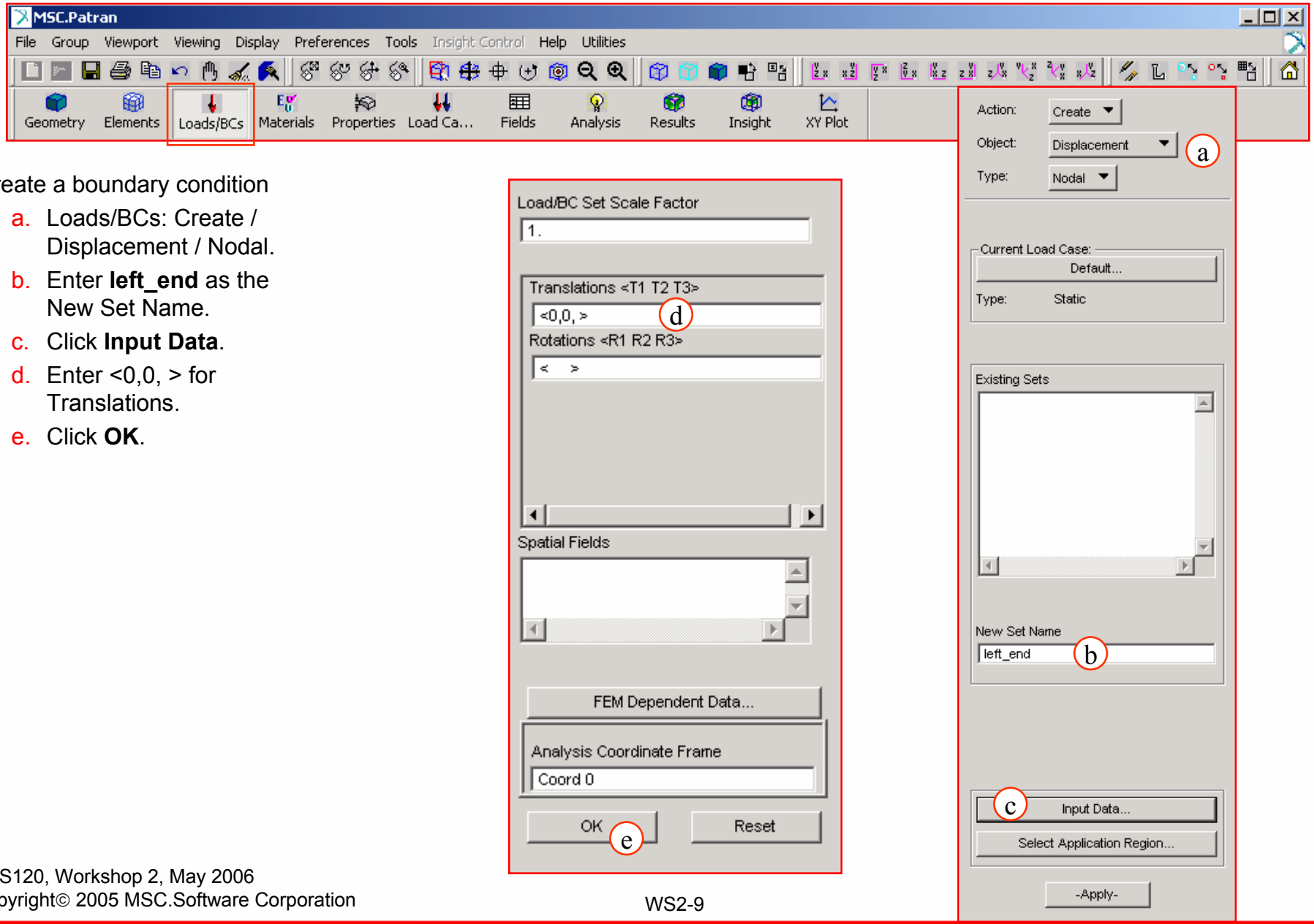
Create a solid mesh

a. Elements: Create / Mesh / Solid
b. Screen pick the solid
c. Click **Apply**.





Step 4. Create Boundary Conditions



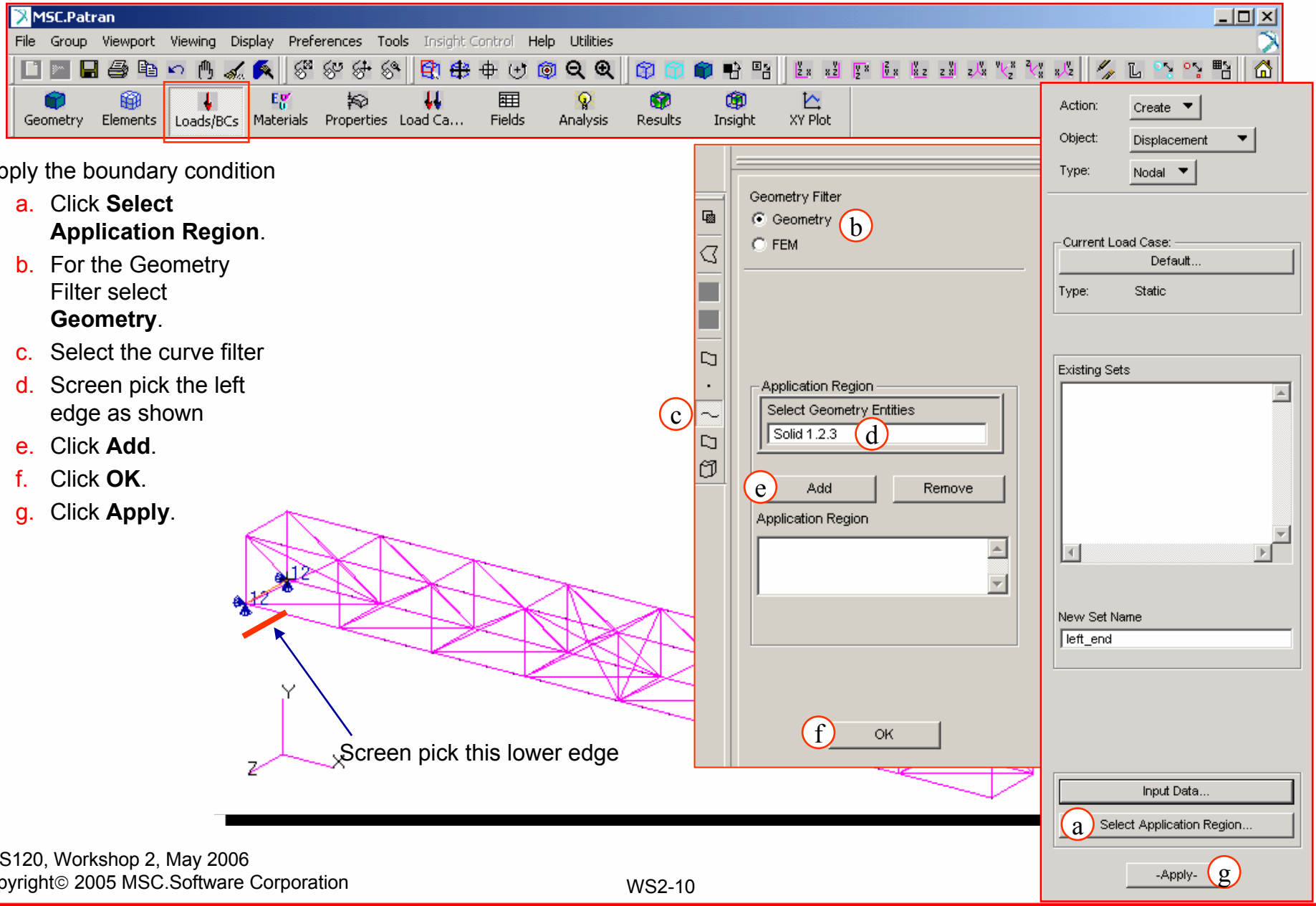
Create a boundary condition

a. Loads/BCs: Create / Displacement / Nodal.
b. Enter **left_end** as the New Set Name.
c. Click **Input Data**.
d. Enter <0,0, > for Translations.
e. Click **OK**.

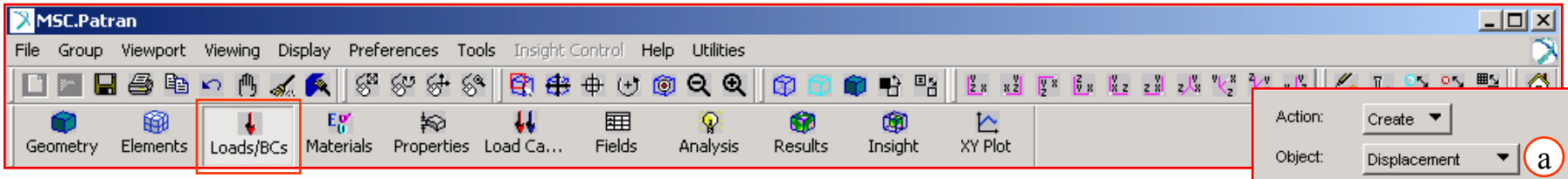
Step 4. Create Boundary Conditions

Apply the boundary condition

a. Click **Select Application Region**.
b. For the Geometry Filter select **Geometry**.
c. Select the curve filter
d. Screen pick the left edge as shown
e. Click **Add**.
f. Click **OK**.
g. Click **Apply**.

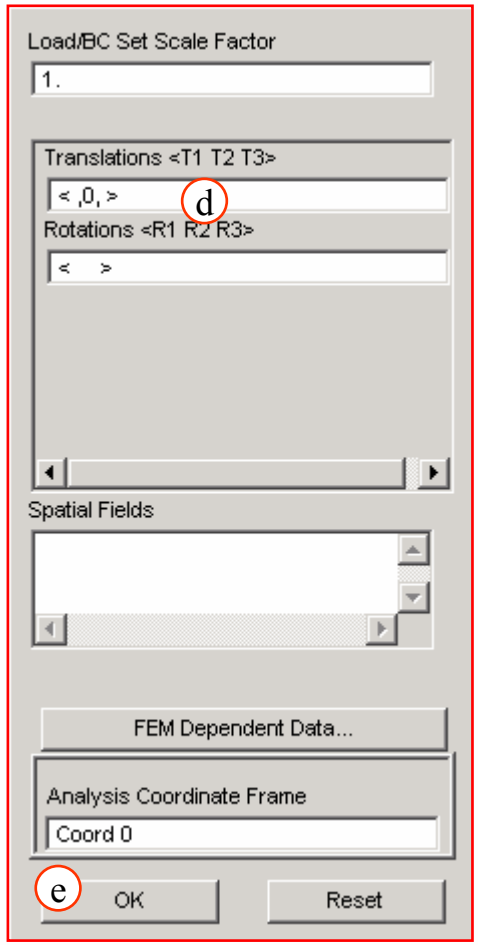


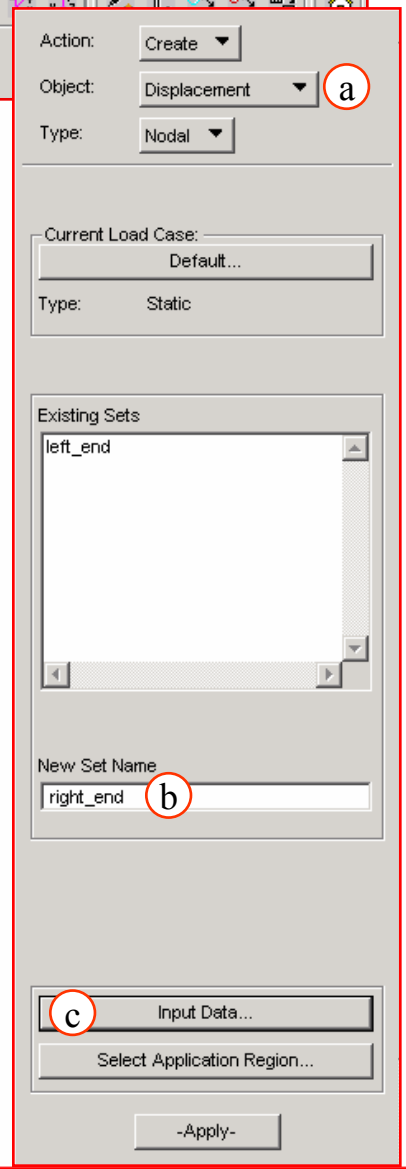
Step 4. Create Boundary Conditions



Create another boundary condition

a. Loads/BCs: Create / Displacement / Nodal.
b. Enter **right_end** as the New Set Name.
c. Click **Input Data**.
d. Enter **< ,0, >** for Translations.
e. Click **OK**.

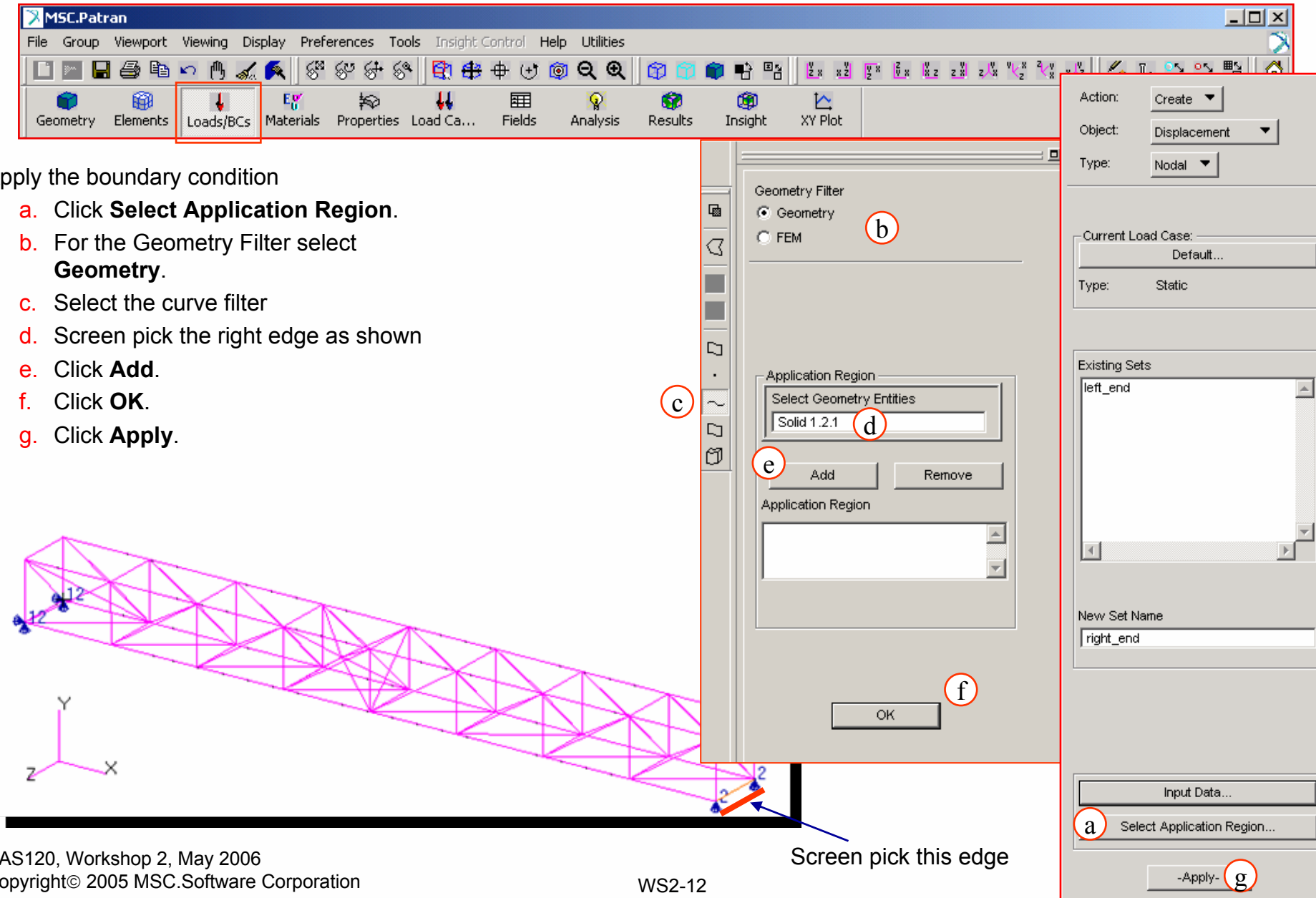




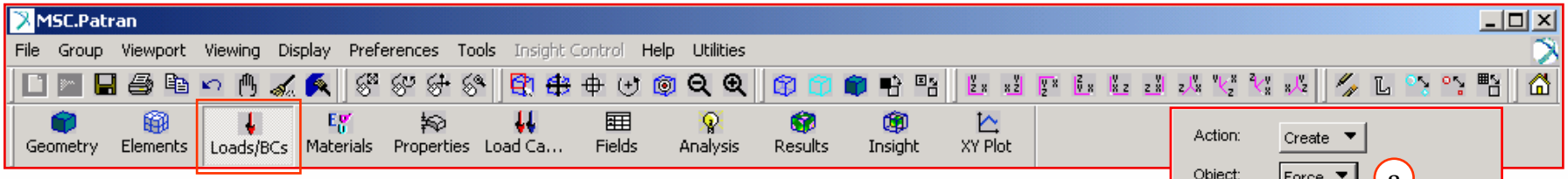
Step 4. Create Boundary Conditions

Apply the boundary condition

a. Click **Select Application Region**.
b. For the Geometry Filter select **Geometry**.
c. Select the curve filter
d. Screen pick the right edge as shown
e. Click **Add**.
f. Click **OK**.
g. Click **Apply**.

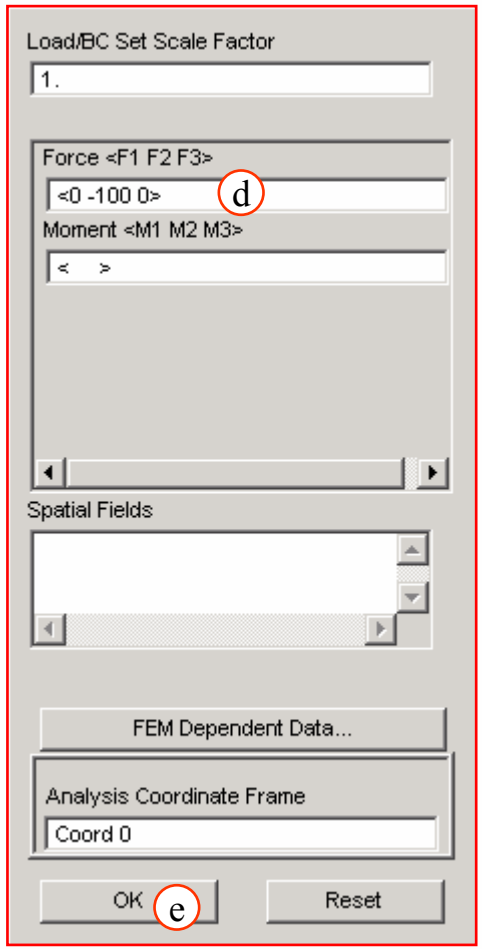


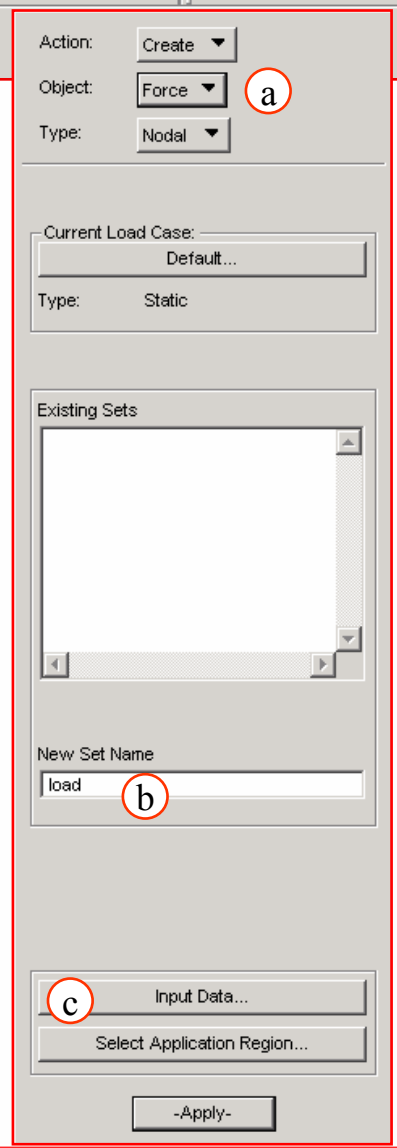
Step 5. Apply Load



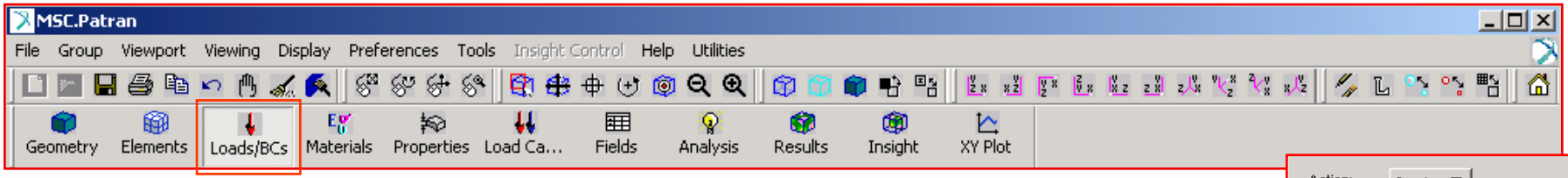
Create a load

a. Loads/BCs: Create / Force / Nodal.
b. Enter **load** as the New Set Name.
c. Click **Input Data**.
d. Enter **<0 -100 0>** for Force.
e. Click **OK**.



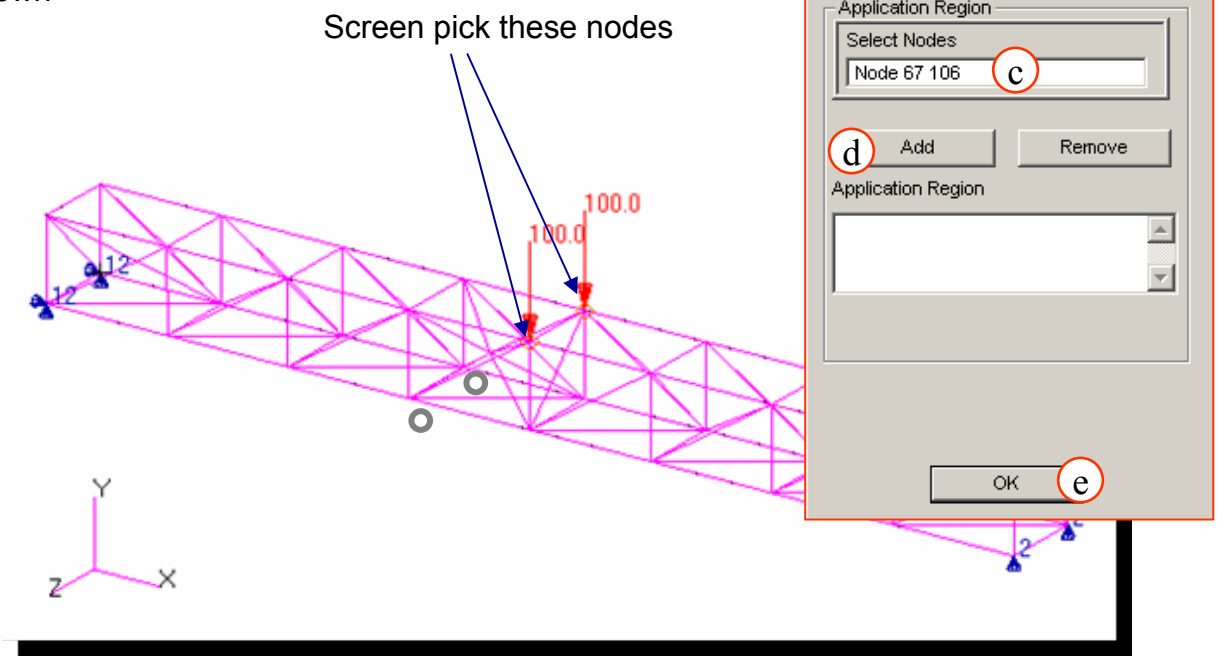


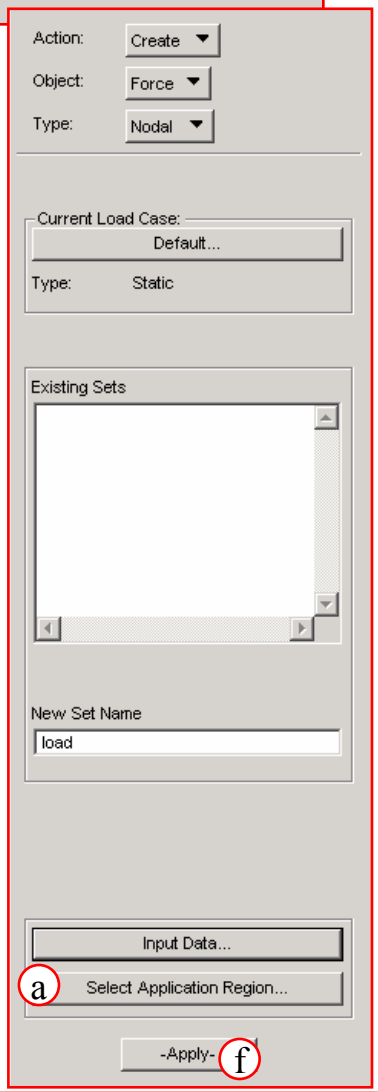
Step 5. Apply Load



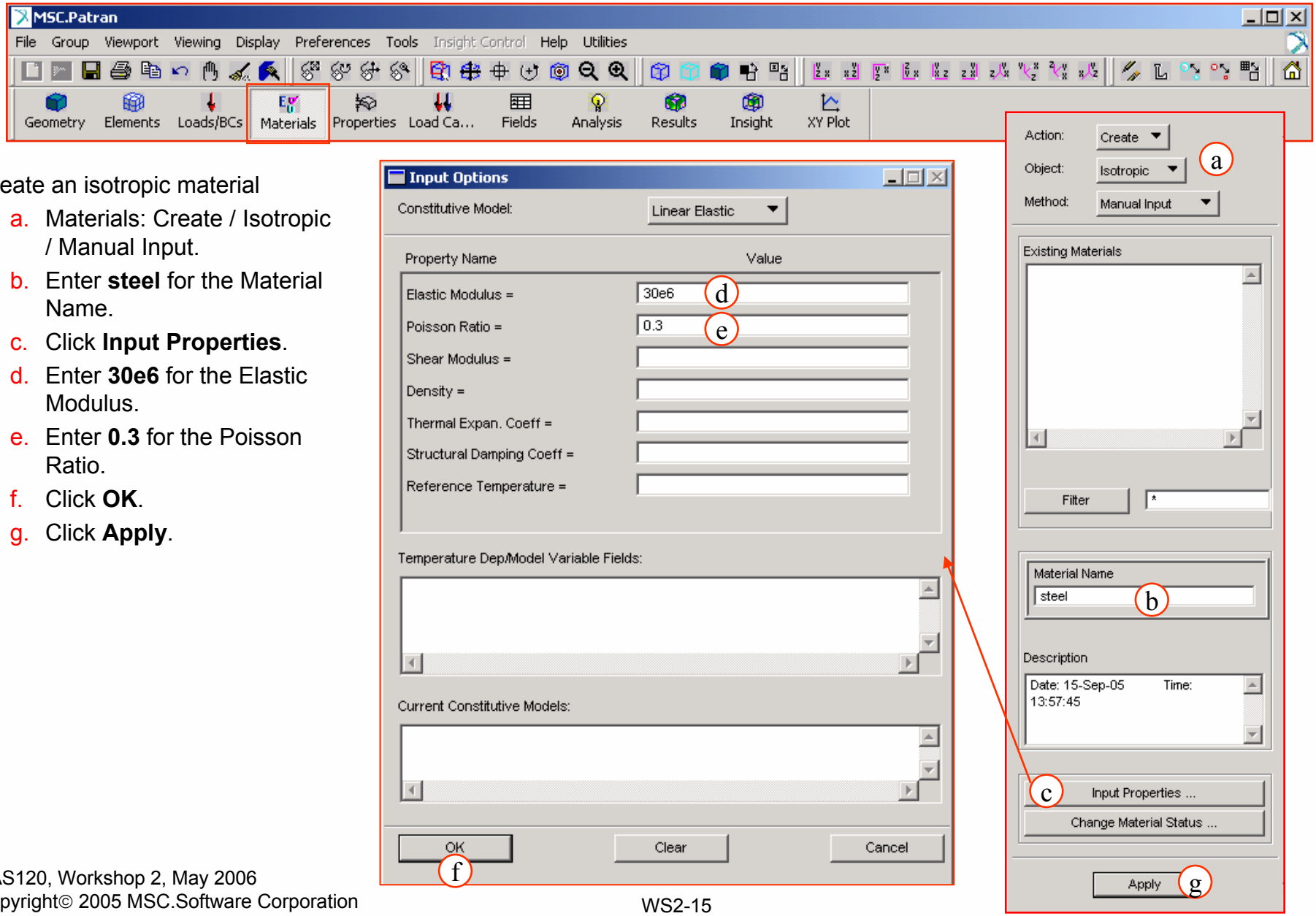
Apply the load

a. Click **Select Application Region**.
b. For the Geometry Filter select **FEM**.
c. Shift/pick the two nodes as shown
d. Click **Add**.
e. Click **OK**.
f. Click **Apply**.





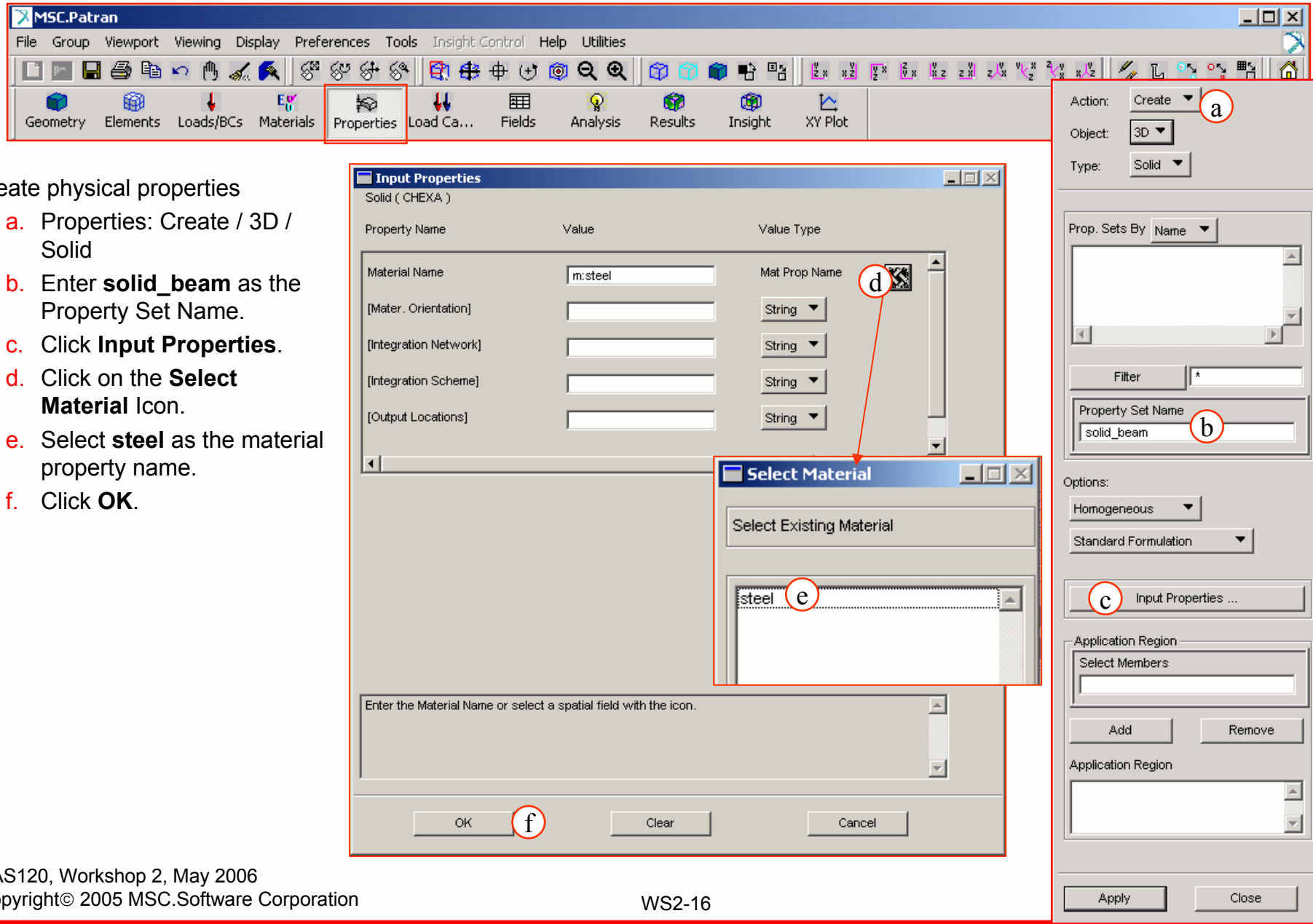
Step 6. Create Material Properties



Create an isotropic material

a. Materials: Create / Isotropic / Manual Input.
b. Enter **steel** for the Material Name.
c. Click **Input Properties**.
d. Enter **30e6** for the Elastic Modulus.
e. Enter **0.3** for the Poisson Ratio.
f. Click **OK**.
g. Click **Apply**.

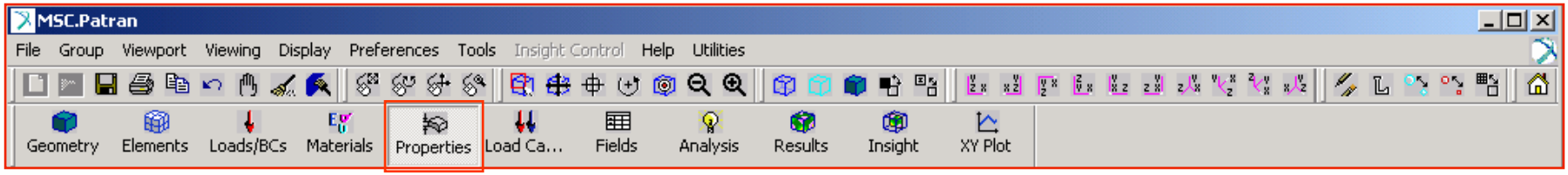
Step 7. Create Physical Properties



Create physical properties

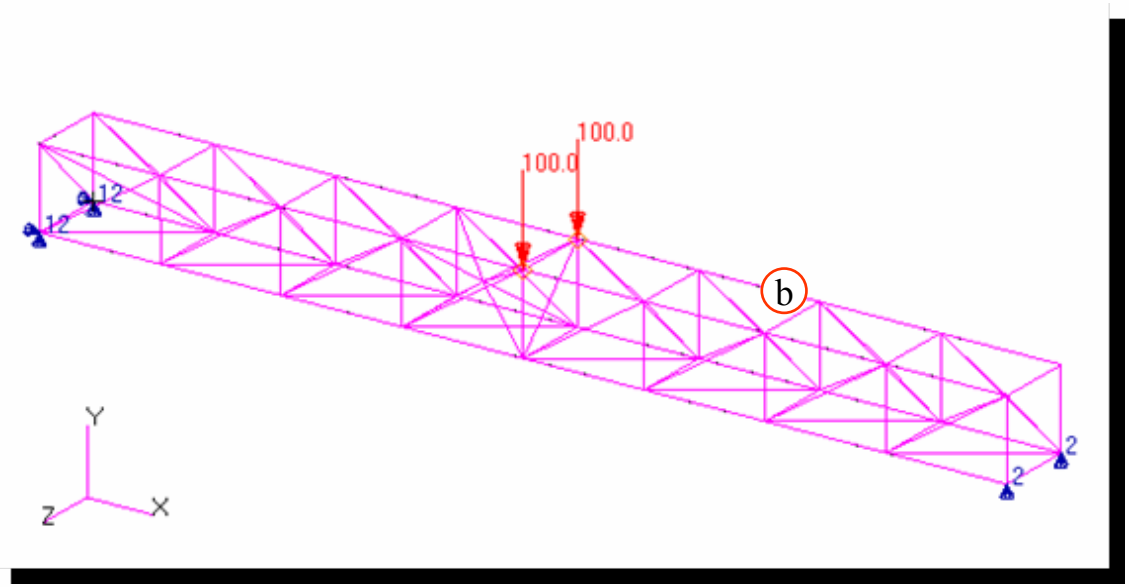
a. Properties: Create / 3D / Solid
b. Enter **solid_beam** as the Property Set Name.
c. Click **Input Properties**.
d. Click on the **Select Material** Icon.
e. Select **steel** as the material property name.
f. Click **OK**.

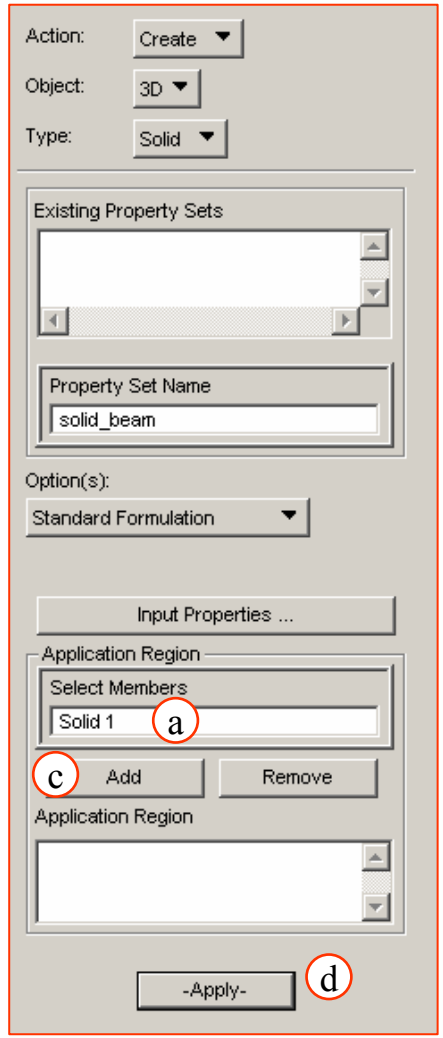
Step 7. Create Physical Properties



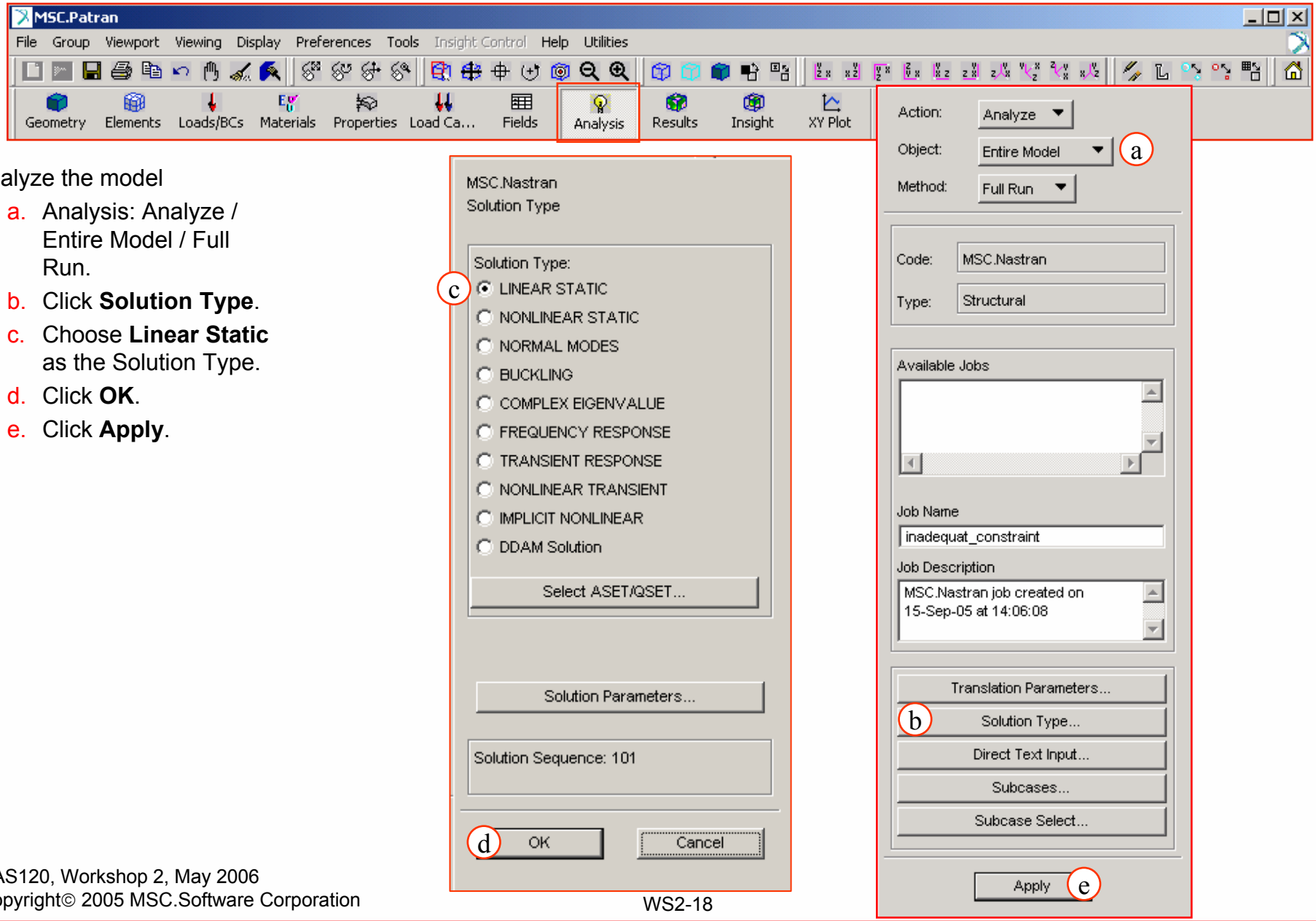
Apply the physical properties

a. Click in the Select Members box.
b. Screen pick the solid
c. Click **Add**.
d. Click **Apply**.





Step 8. Run Linear Static Analysis



Analyze the model

a. Analysis: Analyze / Entire Model / Full Run.
b. Click **Solution Type**.
c. Choose **Linear Static** as the Solution Type.
d. Click **OK**.
e. Click **Apply**.

Step 9. View F06 File

Examine the .f06 file

a. Open the file titled **inadequate_constraint.f06** with any text editor.

b. Examine the warning and fatal messages.

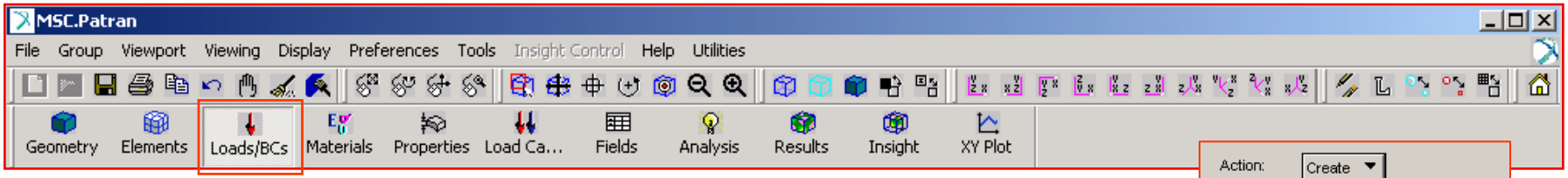
```
 *** USER INFORMATION MESSAGE 4158 (DFMSA)
 ---- STATISTICS FOR SPARSE DECOMPOSITION OF DATA BLOCK KLL       FOLLOW
        NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL =      1
        MAXIMUM RATIO OF MATRIX DIAGONAL TO FACTOR DIAGONAL =  4.8E+14 AT ROW NUMBER        180
 *** USER WARNING MESSAGE 4698 (DCMPD)
     STATISTICS FOR DECOMPOSITION OF MATRIX KLL      .
     THE FOLLOWING DEGREES OF FREEDOM HAVE FACTOR DIAGONAL RATIOS GREATER THAN
      1.00000E+07 OR HAVE NEGATIVE TERMS ON THE FACTOR DIAGONAL.
     USER INFORMATION:
     THIS MESSAGE MAY BE IGNORED IF NO GRID POINT IDS OR HIGH RATIO MESSAGES APPEAR IN THE TABLE ON THE NEXT PAGE.
1    MSC.NASTRAN JOB CREATED ON 17-OCT-03 AT 10:21:30                    OCTOBER  17, 2003  MSC.NASTRAN  9/23/03

0                                                                                                      SUBCASE 1
0
          GRID POINT ID       DEGREE OF FREEDOM   MATRIX/FACTOR DIAGONAL RATIO        MATRIX DIAGONAL

                61                    T3                   -4.80409E+14                  2.48875E+07
 ^^^
 ^^^ USER    FATAL    MESSAGE 9050 (SEKRRS)
 ^^^ RUN TERMINATED DUE TO EXCESSIVE PIVOT RATIOS IN MATRIX KLL.
 ^^^ USER ACTION:  CONSTRAIN MECHANISMS WITH SPCI OR SUPORTI ENTRIES OR SPECIFY PARAM,BAILOUT, 1 TO
     CONTINUE THE RUN WITH MECHANISMS.
```

Why has the job failed?

a. The warning message in the .f06 file lists T3 as the problem degree of freedom.

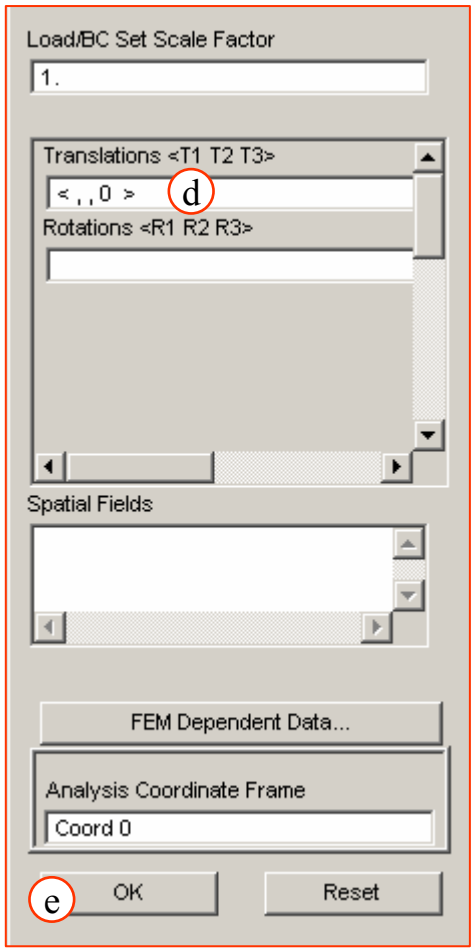
b. With constraints in the x-y plane only, the beam has a rigid body motion in the z direction. Need to add a constraint in the z direction.

Step 10. Add New Boundary Condition

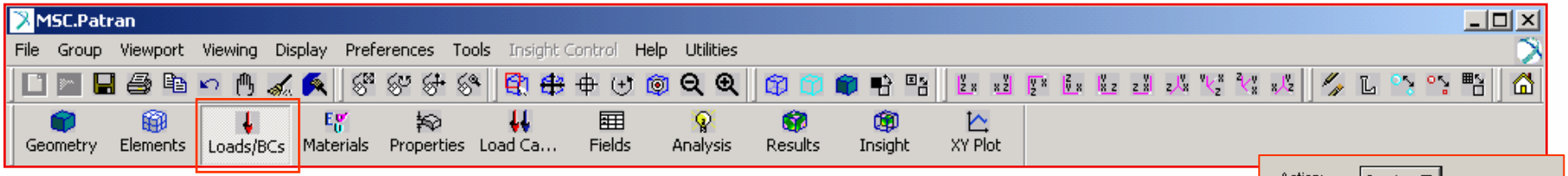


Add a boundary condition

a. Loads/BCs: Create / Displacement / Nodal.
b. Enter **z_constraint** as the New Set Name.
c. Click **Input Data**.
d. Enter **< , ,0 >** for Translations.
e. Click **OK**.

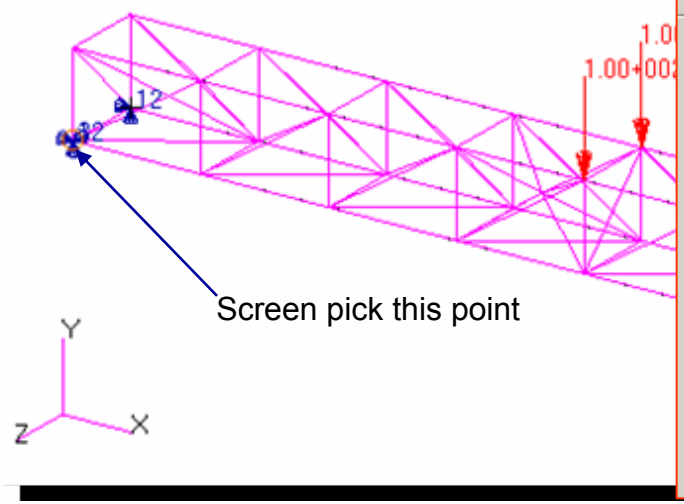


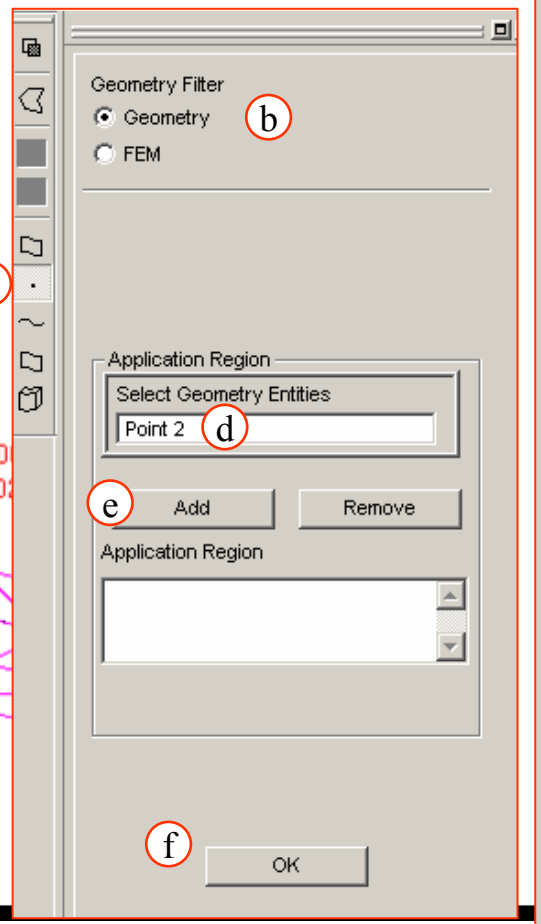
Step 10. Add New Boundary Condition

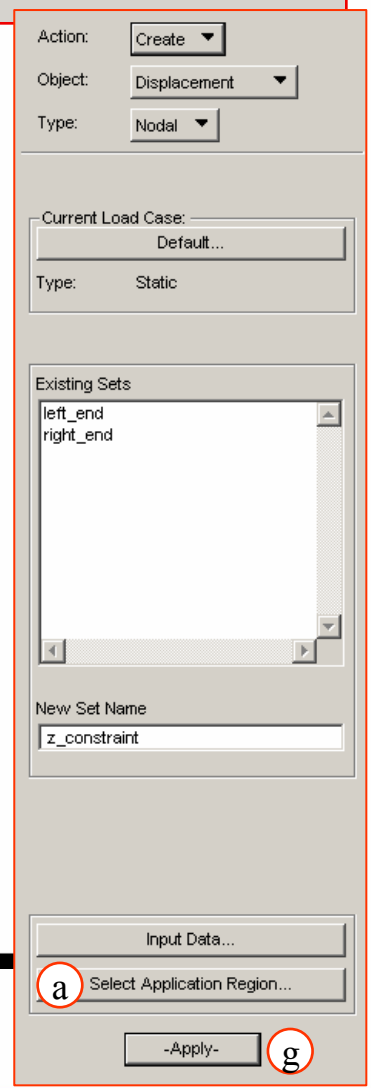


Apply the boundary condition

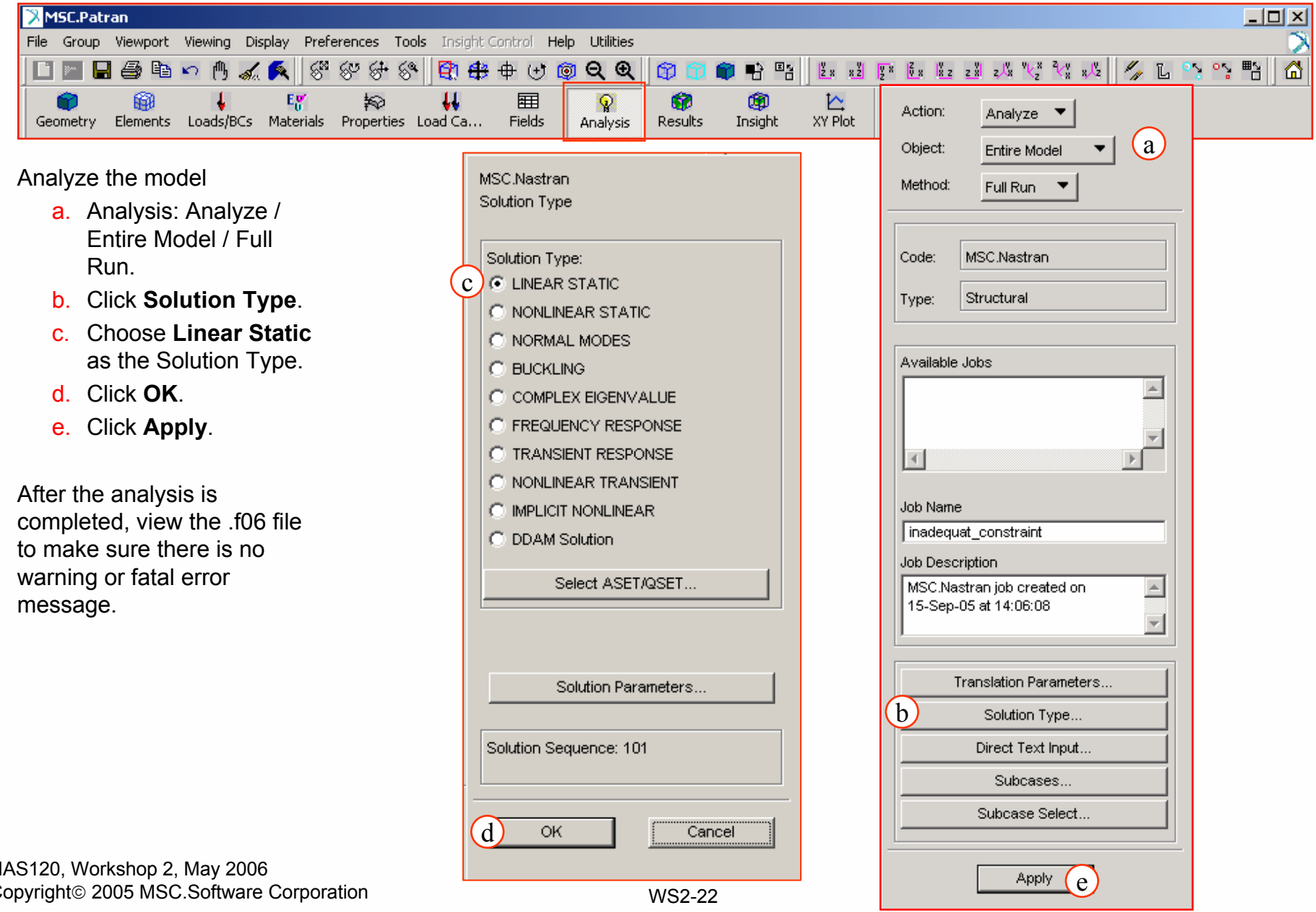
a. Click **Select Application Region**.
b. For the Geometry Filter select **Geometry**.
c. Select the point filter
d. Screen pick the left corner as shown
e. Click **Add**.
f. Click **OK**.
g. Click **Apply**.







Step 11. Re-run Linear Static Analysis

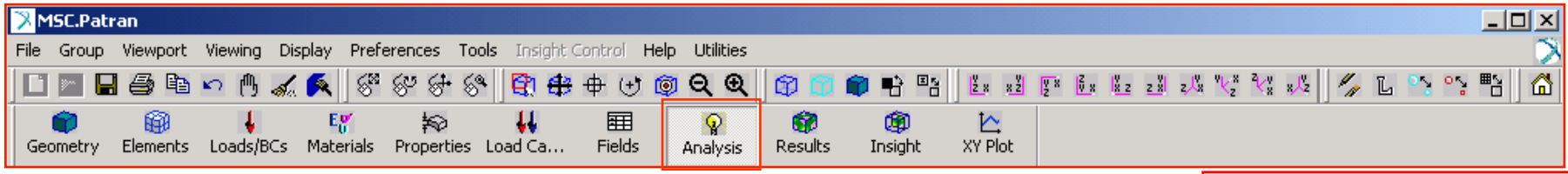


Analyze the model

a. Analysis: Analyze / Entire Model / Full Run.
b. Click **Solution Type**.
c. Choose **Linear Static** as the Solution Type.
d. Click **OK**.
e. Click **Apply**.

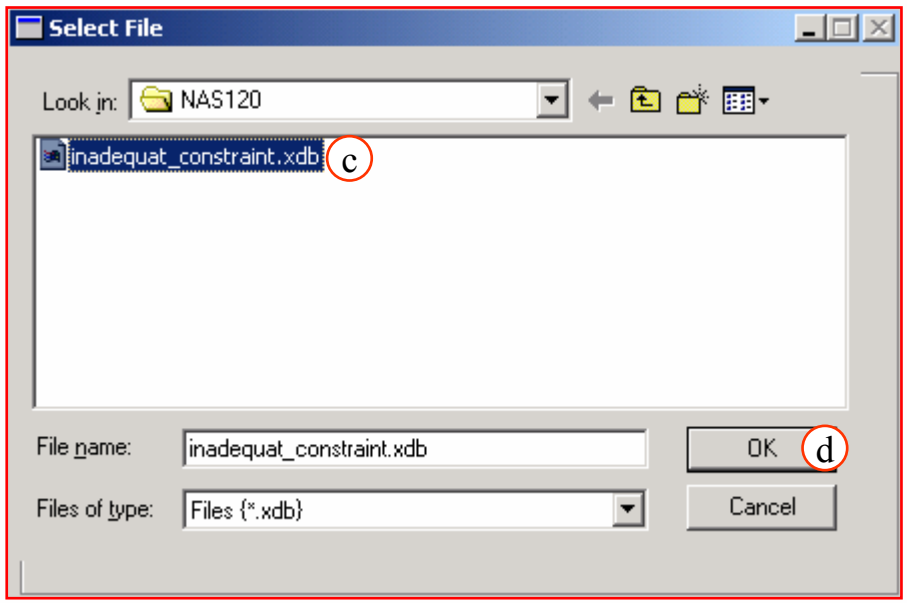
After the analysis is completed, view the .f06 file to make sure there is no warning or fatal error message.

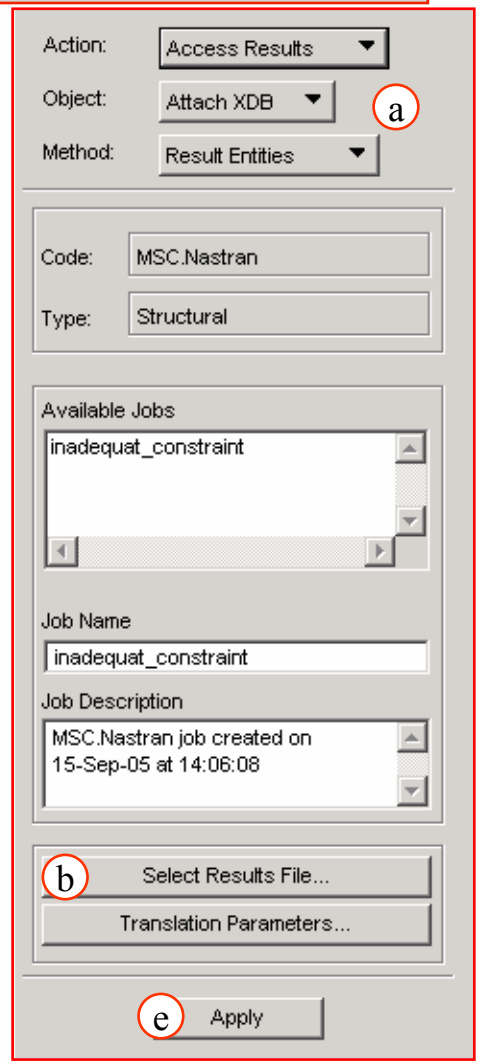
Step 12. Access the Results File



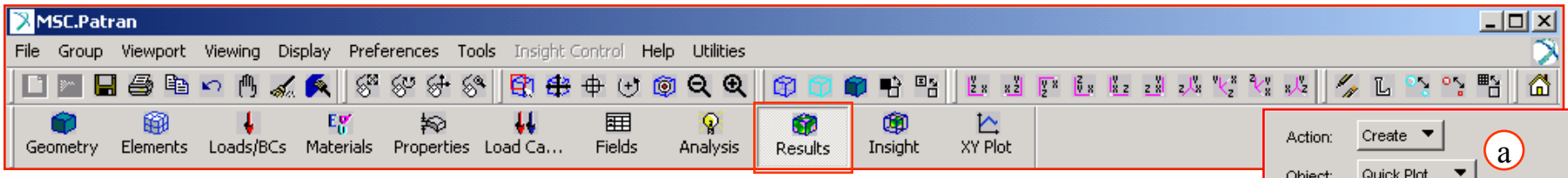
Access the results file

a. Analysis: Access Results / Attach XDB / Result Entities.
b. Click **Select Results File**.
c. Select the file **inadequate_constraint.xdb**
d. Click **OK**.
e. Click **Apply**.





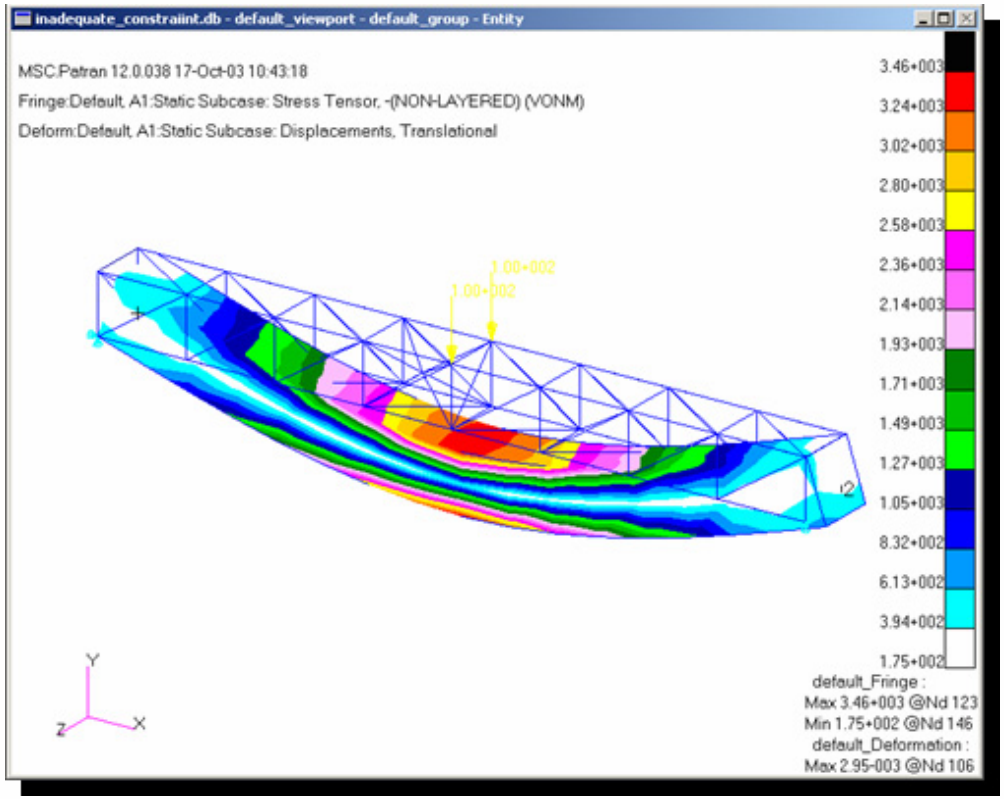
Step 13. Plot the Results

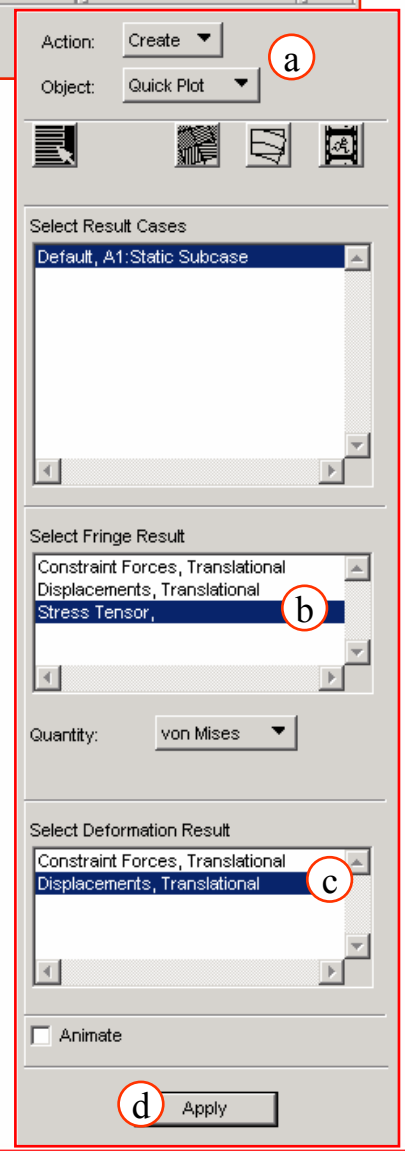


Plot the results

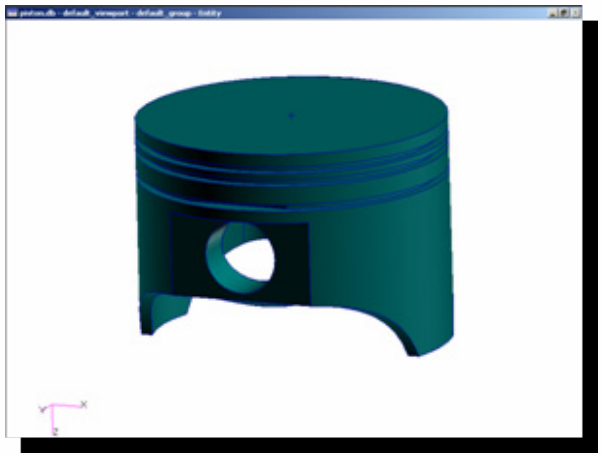
a. Results: Create / Quick Plot
b. Select **Stress Tensor** for fringe result
c. Select **Displacement, Translational** for deformation result
d. Click **Apply**.

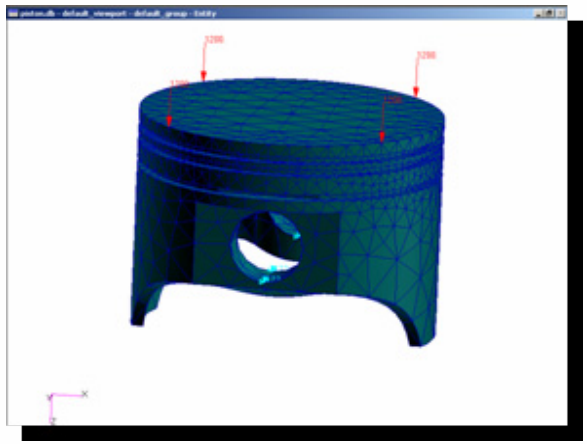
-- End of workshop --

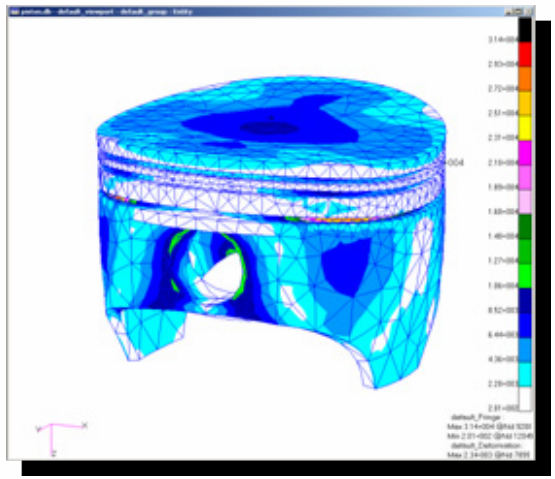




WORKSHOP 1
PISTON HEAD ANALYSIS



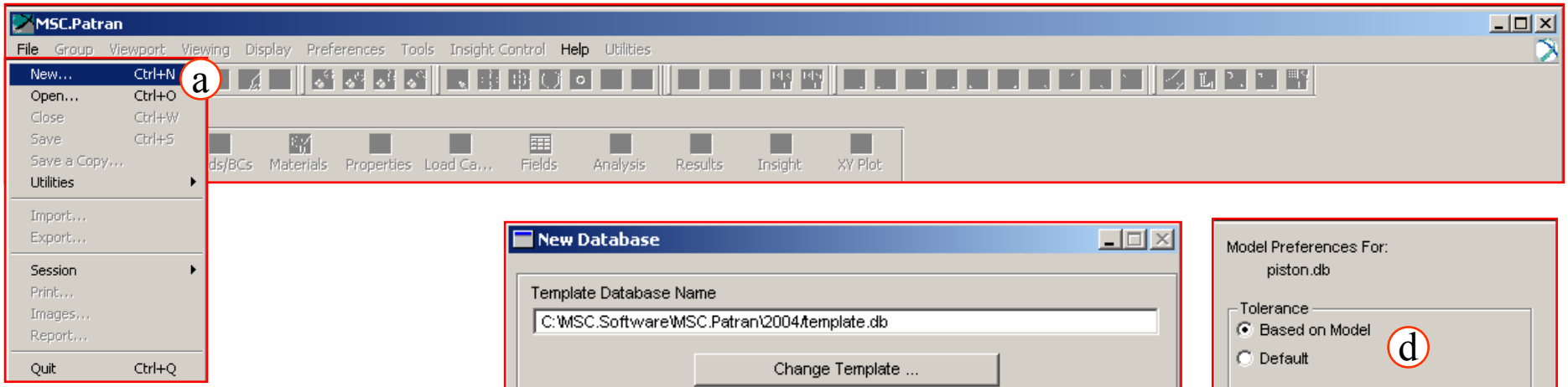


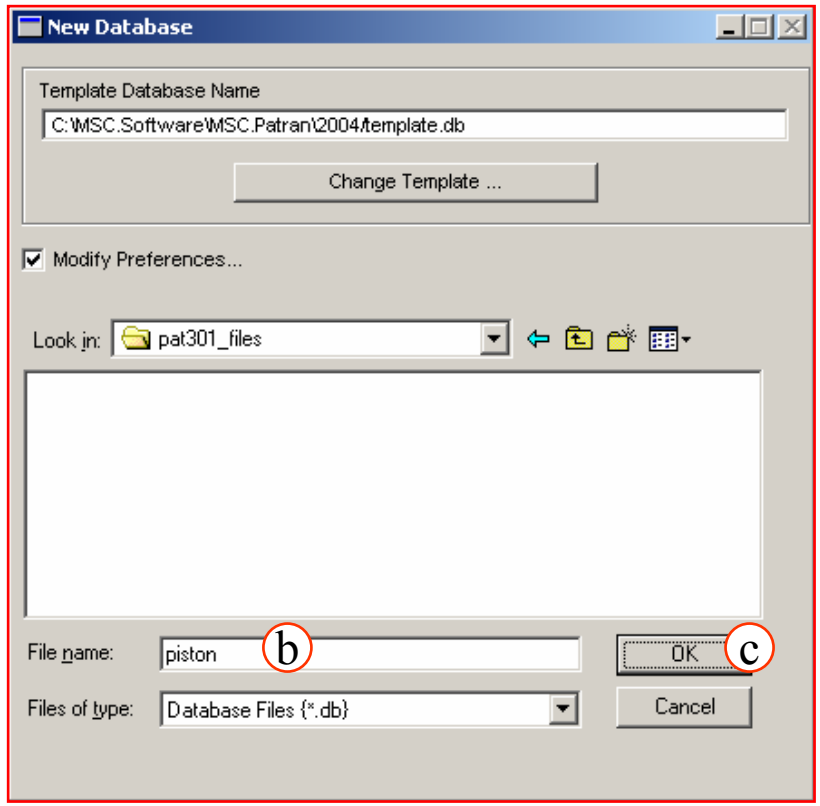


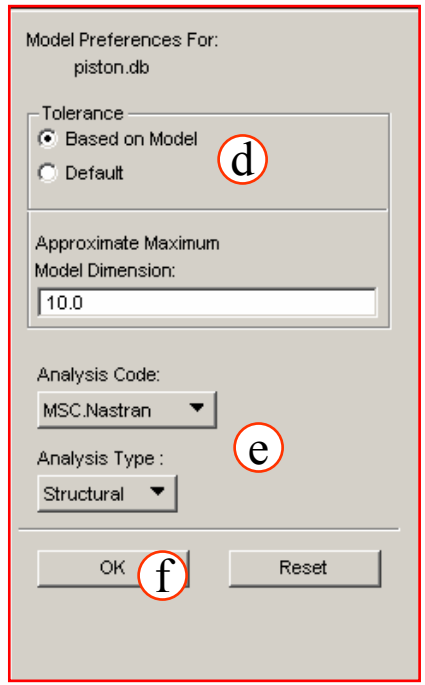
- **Workshop Objectives**
  - Become familiar all the steps necessary to create a very simple model, analyze it, and postprocess the results from its analysis using MSC.Patran and MSC.Nastran.
- **Problem Description**
  - Determine if the stress in this typical piston model is low enough for the material to be in the linear elastic region.
  - Piston material: Steel with $E = 30 \times 10^6$ psi and $\nu = 0.3$
  - Pressure on piston = 1200 psi
- **Software Version**
  - MSC.Patran 2005r2
  - MSC.Nastran 2005r2b

- **Key Concepts and Steps:**
  - **Database:** create a new database with Analysis Code = MSC.Nastran and Analysis Type = Structural
  - **Geometry:** import Parasolid geometry of the piston
  - **Elements:** mesh the geometry with solid Tet10 elements
  - **Loads/BCs:** constrain the pinhole surfaces, and apply pressure to the top of the piston
  - **Materials:** specify an isotropic material for Steel
  - **Properties:** create a 3D solid property
  - **Analysis:** Solution Type = Nastran Linear Static, Solution Sequence = 101, Method = Full Run
  - **Analysis:** access analysis results by attaching the XDB file to database
  - **Results:** plot von Mises stress and displacement results

Step 1. Create a New Database



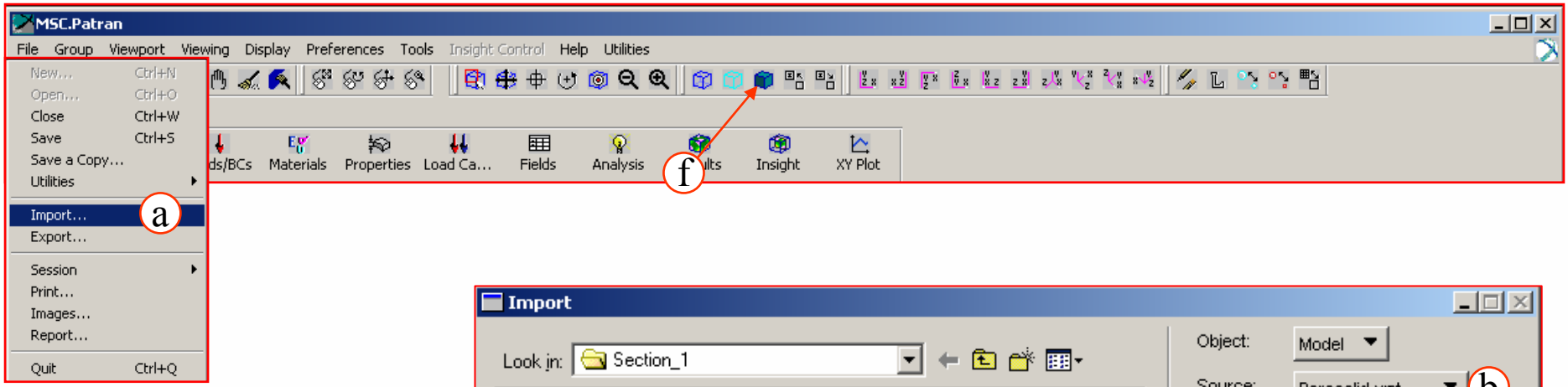


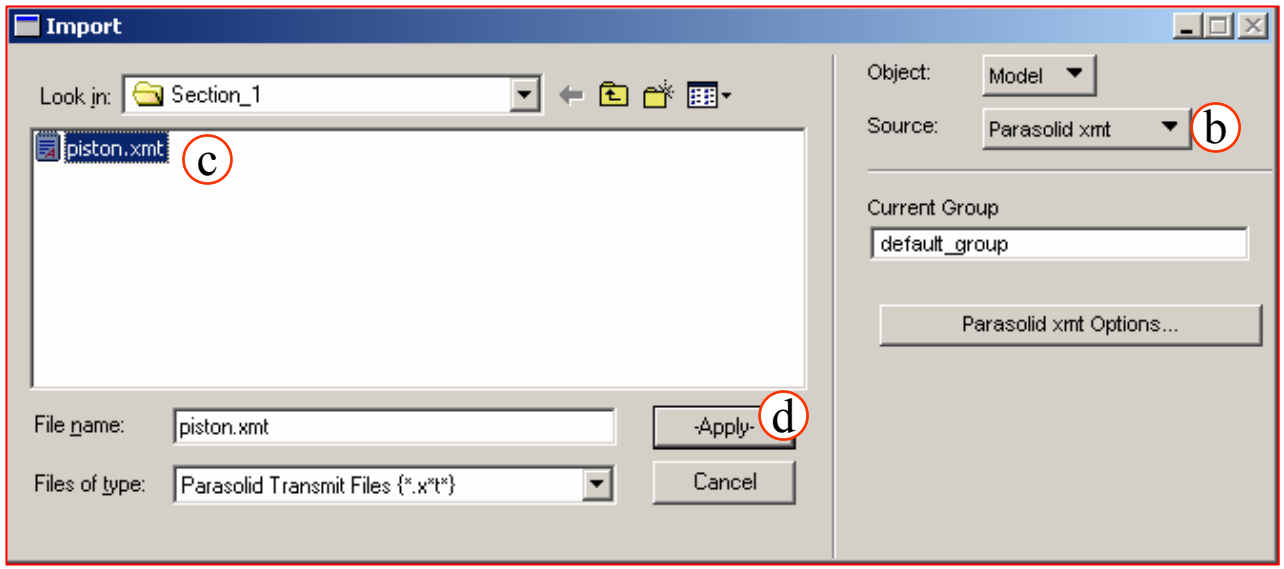


Create a new database called **piston.db** and set the model preferences.

a. File / New.
b. Enter **piston.db** for the *File Name.*
c. Click on **OK**.
d. Set the *Tolerance* under *Model Preferences* to **Based on Model**.
e. Make sure that the *Analysis Code* and *Analysis Type* are set to **MSC.Nastran** and **Structural**, respectively.
f. Click on **OK**.

Step 2. Import the Model Geometry

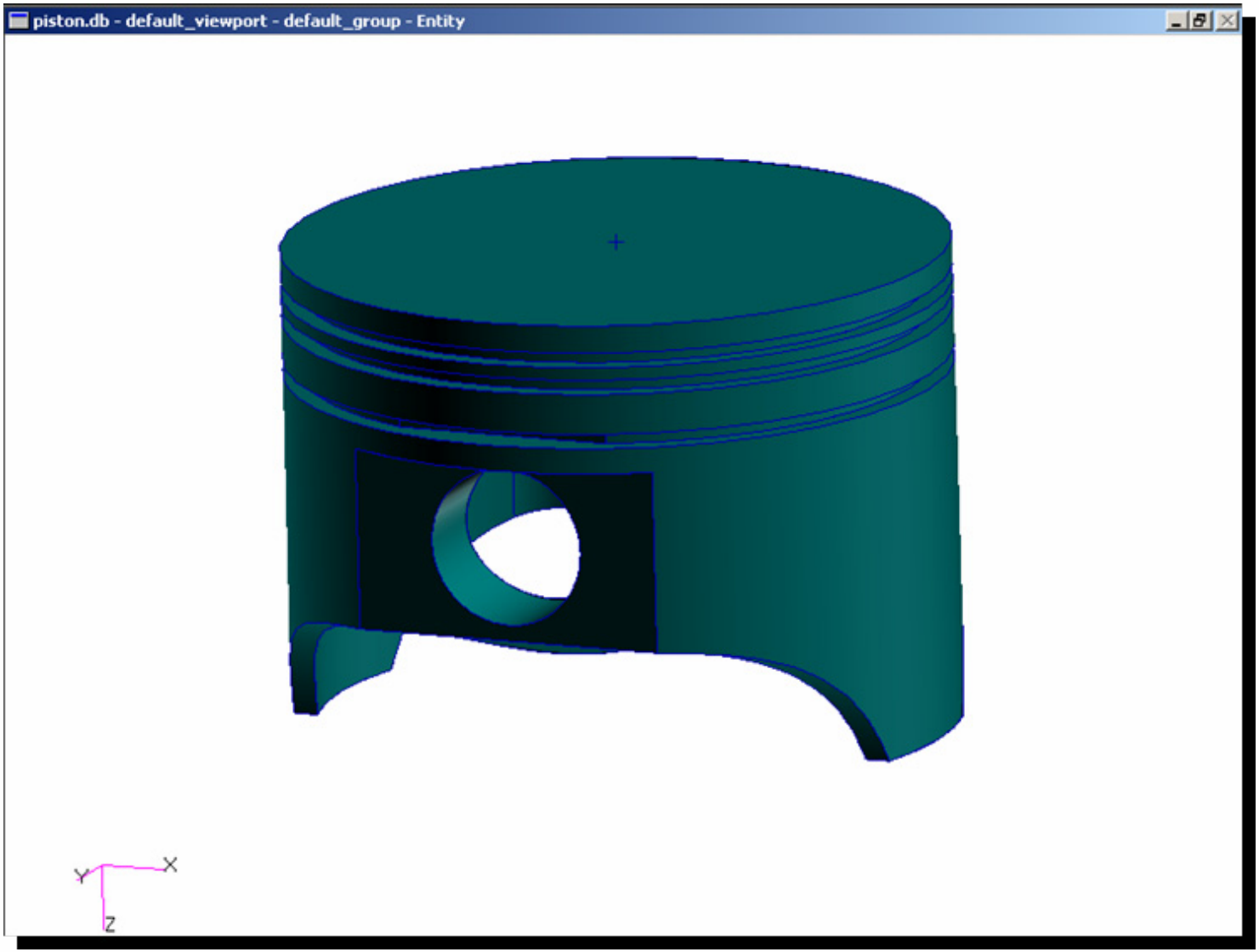




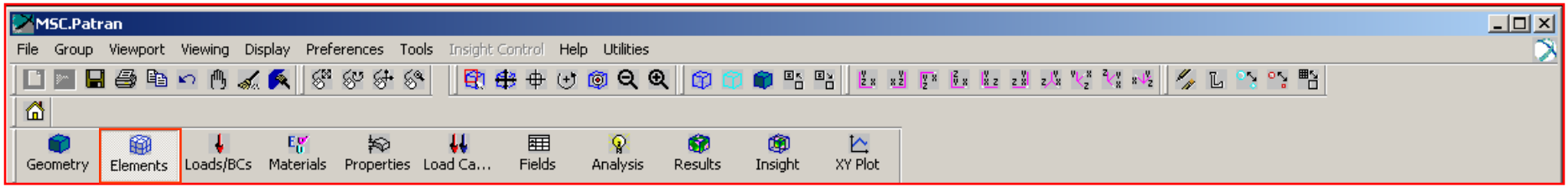
Import the model geometry.

a. File / Import
b. Make sure *Source* is set to **Parasolid.xmt**.
c. Select **piston.xmt**.
d. Click on **Apply**.
e. Click on **OK** when **Import Summary** form appears.
f. Click on **Smooth Shaded** icon and **Top View** icons.
g. Use the middle mouse button and rotate the model to attain the same view shown on next page.

Step 2. Import the Model Geometry (Cont.)

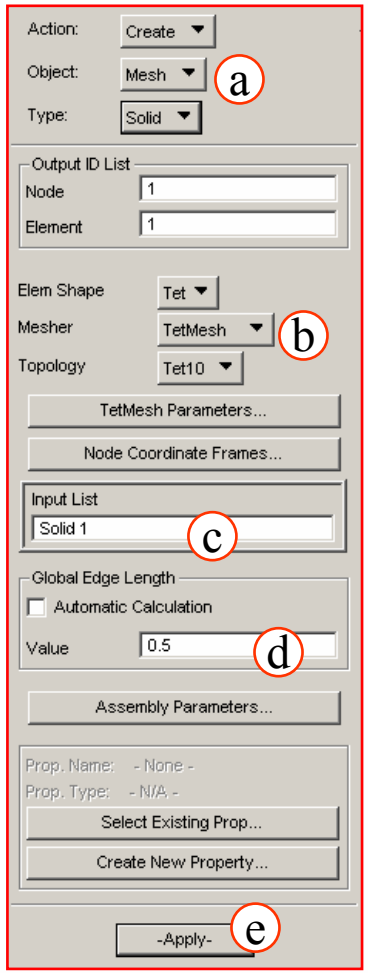


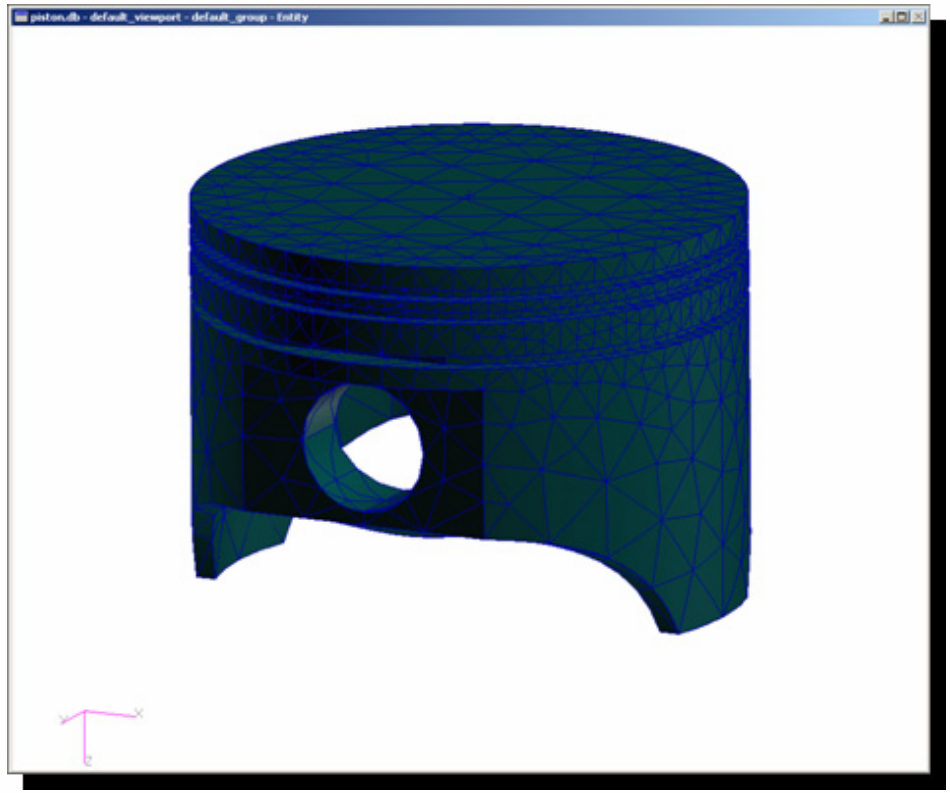
Step 3. Create a Finite Element Mesh



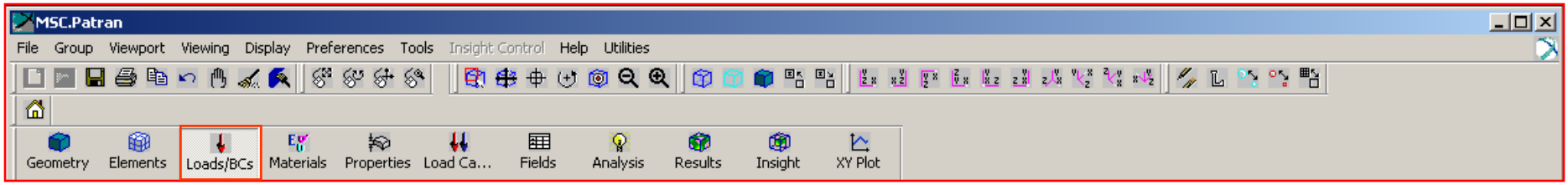
Create a finite element mesh for the model using the TetMesh-er.

a. Elements : Create / Mesh / Solid.
b. Set *Elem. Shape*, *Mesher*, and *Topology* to **Tet**, **TetMesh**, and **Tet10**, respectively.
c. Select **Solid 1** under *Input List*.
d. Uncheck **Automatic Calculation** and enter **0.5** for the *Global Edge Length*.
e. Click on **Apply**.



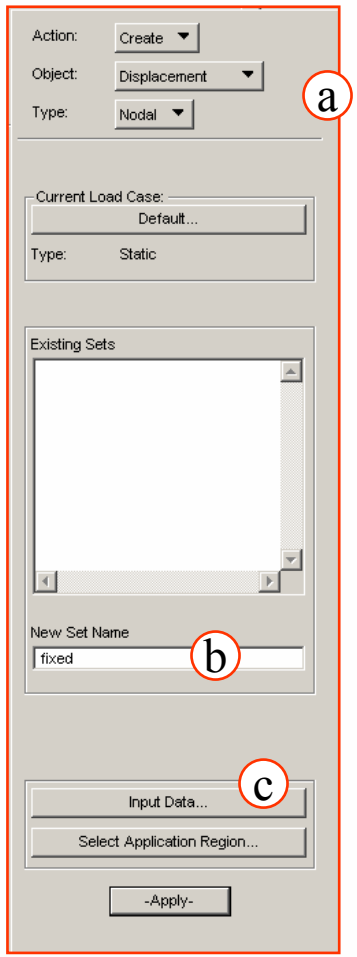


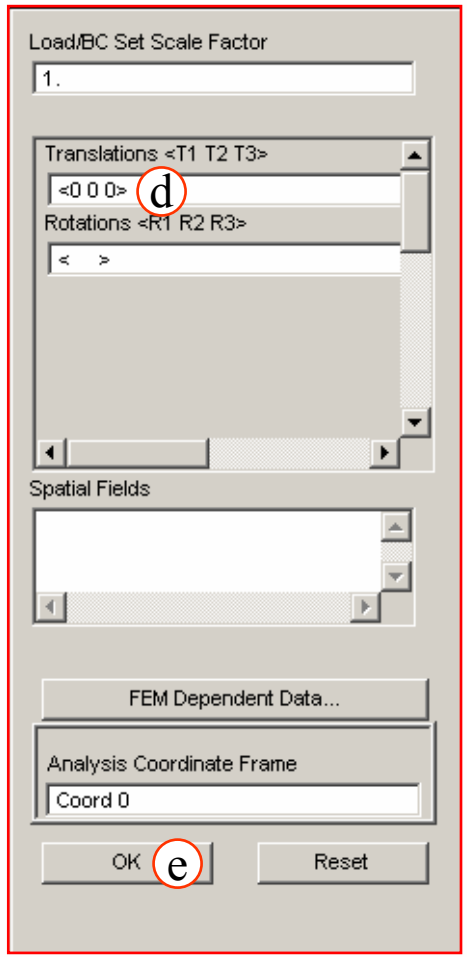
Step 5. Create Constraints



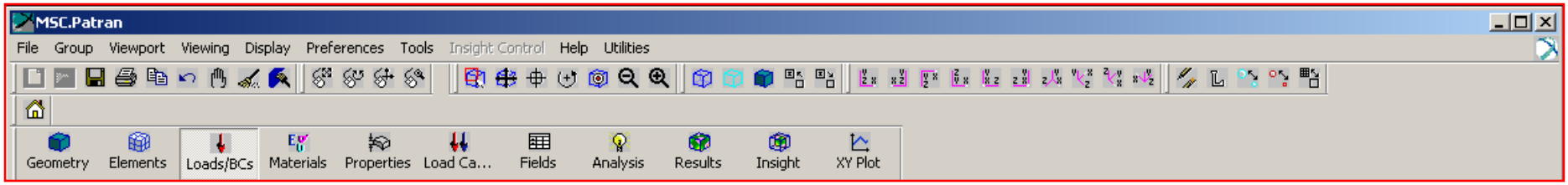
Constrain the two pinhole surfaces.

a. Loads/BCs : Create / Displacement / Nodal.
b. Enter **fixed** for the *New Set Name*.
c. Click on **Input Data…**
d. Enter **<0 0 0>** under *Translations* only.
e. Click on **OK**.



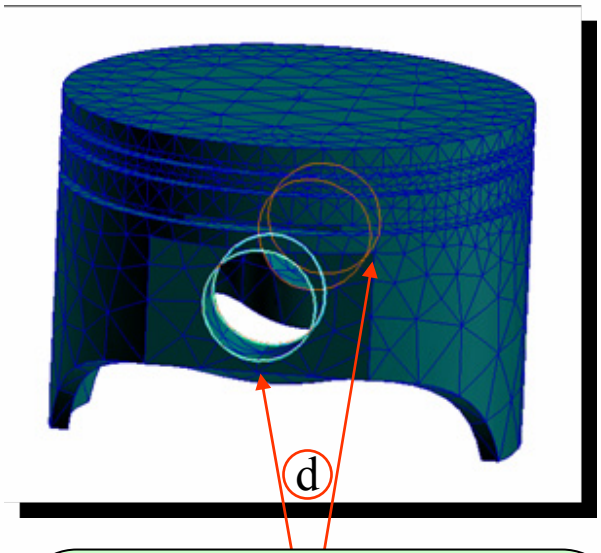


Step 5. Create Constraints (Cont.)

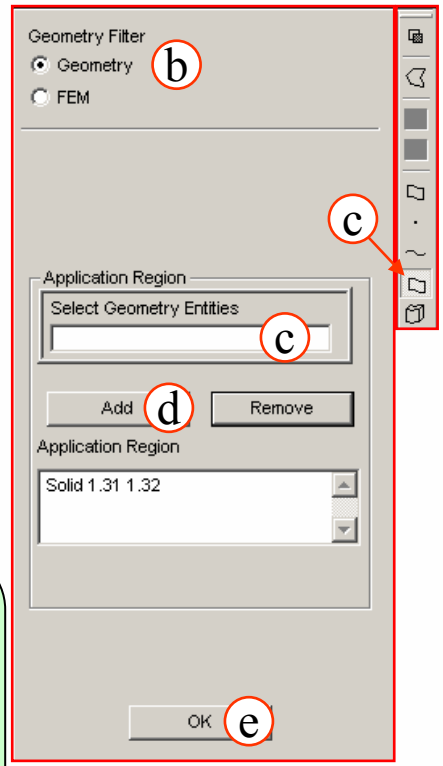


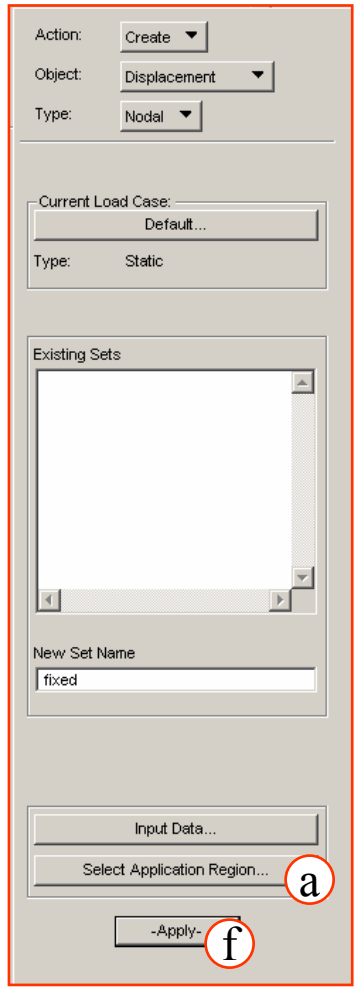
Select the application region for the nodal constraints.

a. Click on **Select Application Region…**
b. Set the *Geometry Filter* to **Geometry**.
c. Click under *Select Geometry Entities*, then on the **Surface or Face** icon.
d. Select both pinhole faces and click **Add**.
e. Click on **OK**.
f. Click on **Apply**.

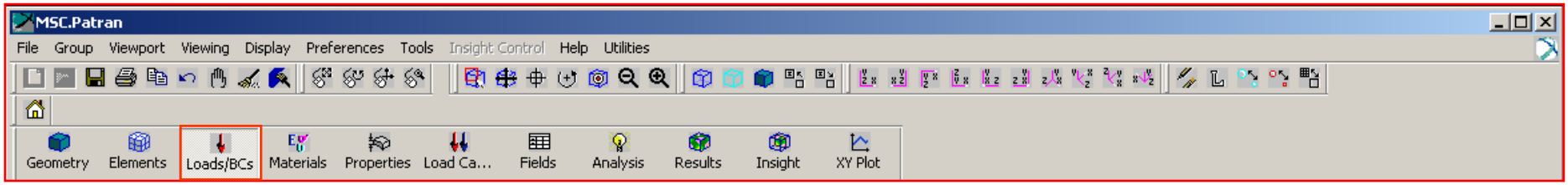


Illustrated above are the two solid faces that are used to define the constraint application region for the model. After choosing the first solid face, for the application region, and clicking **Add**, use the middle-mouse button to rotate the model and select the other solid face.



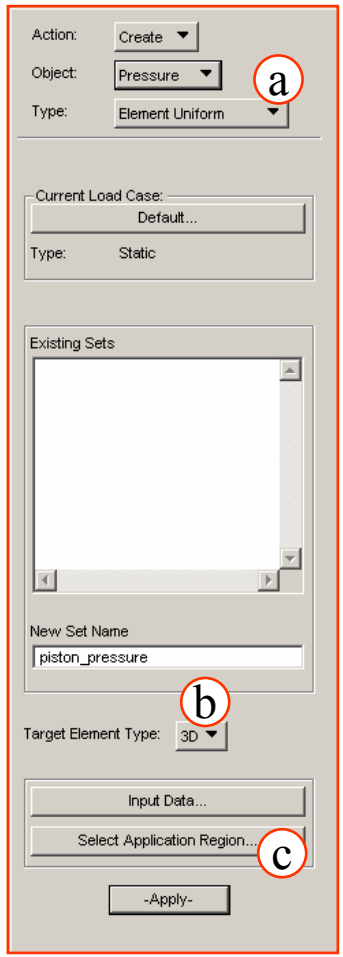


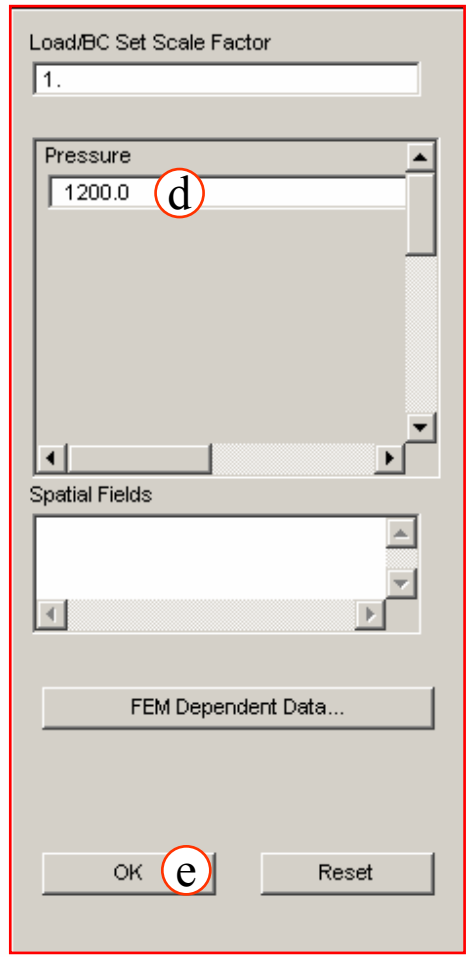
Step 6. Create a Pressure Load



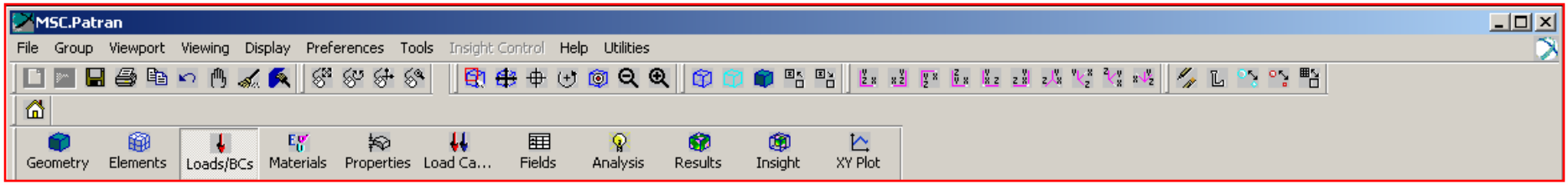
Create a pressure that will be applied to the top surface of the piston.

a. Loads/BCs : Create / Pressure/ Element Uniform.
b. Enter **piston_pressure** for the *New Set Name*.
c. Click on **Input Data…**
d. Enter **1200.0** for the *Pressure*.
e. Click on **OK**.



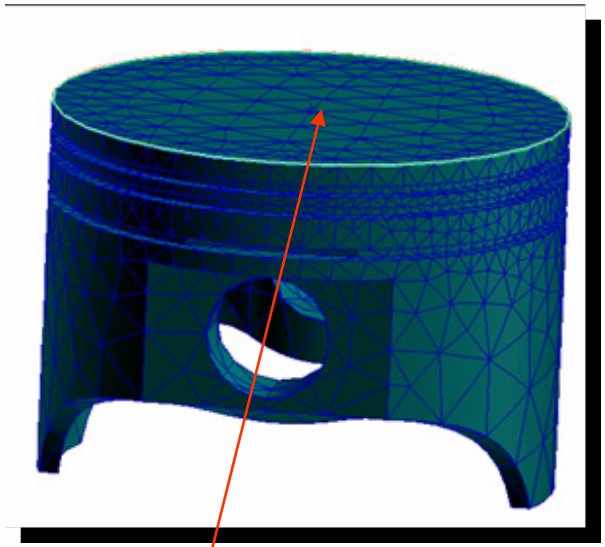


Step 6. Create a Pressure Load (Cont.)

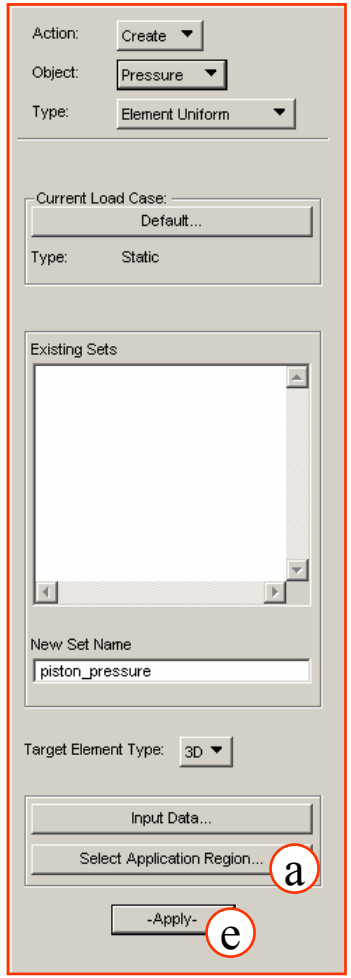


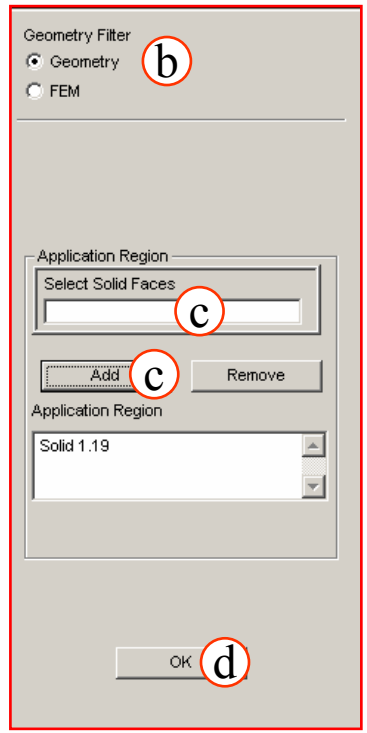
Select the application region for the pressure.

a. Click on **Select Application Region…**
b. Set the *Geometry Filter* to **Geometry**.
c. Click under *Select Solid Faces, then* select the top face of the piston, and click **Add**.
d. Click on **OK**.
e. Click on **Apply**.

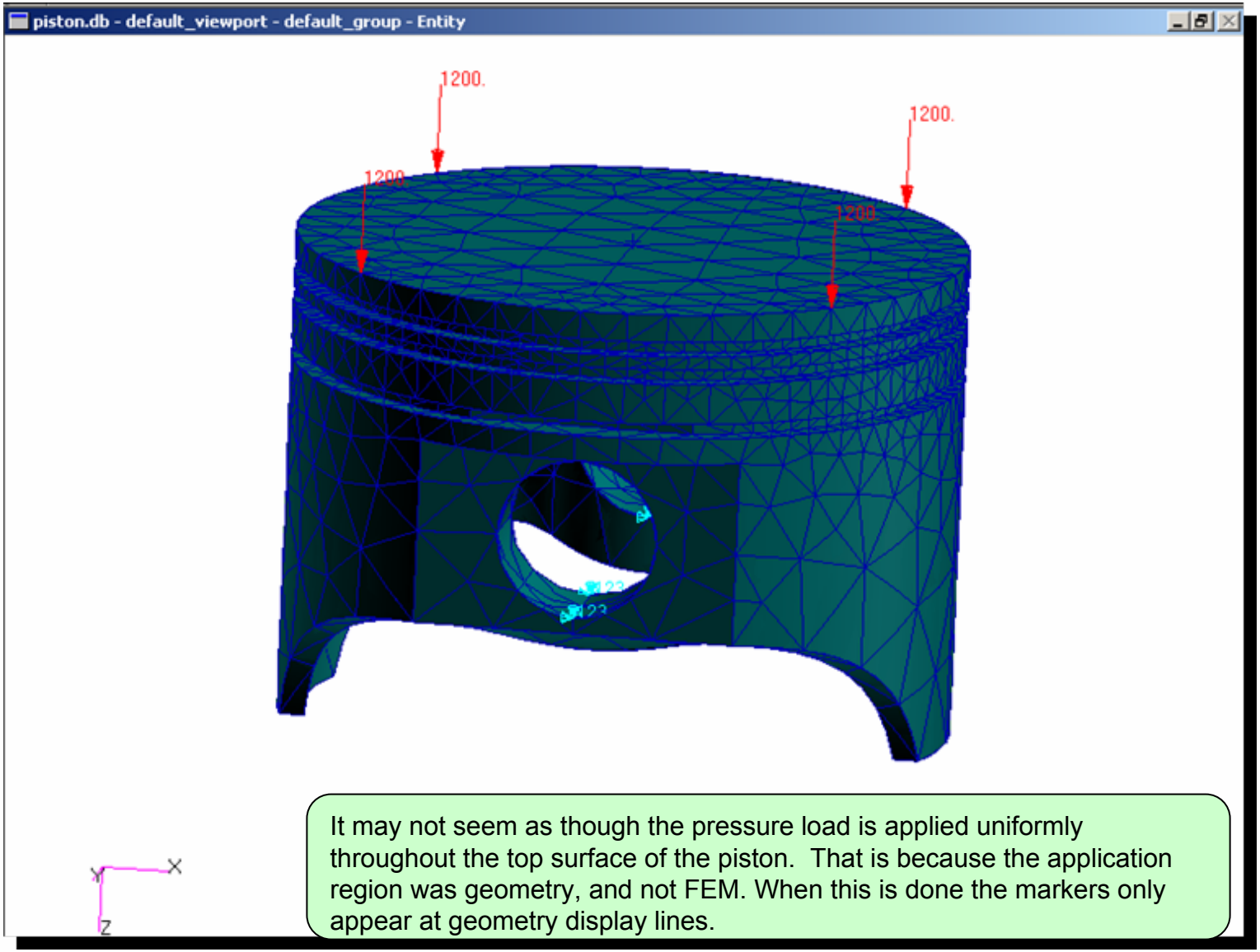


Shown above is the application region for the pressure load.



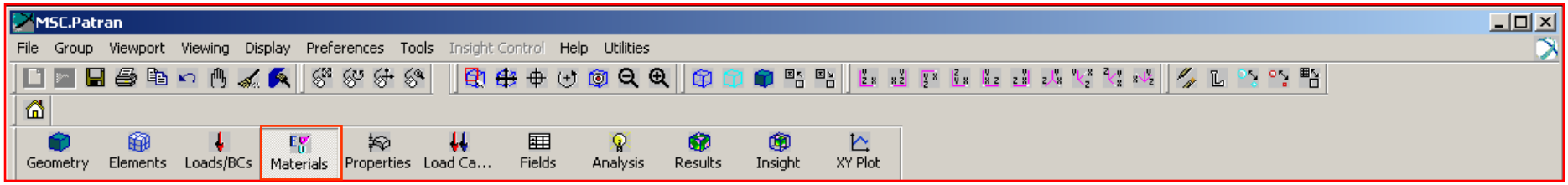


Step 6. Create a Pressure Load (Cont.)



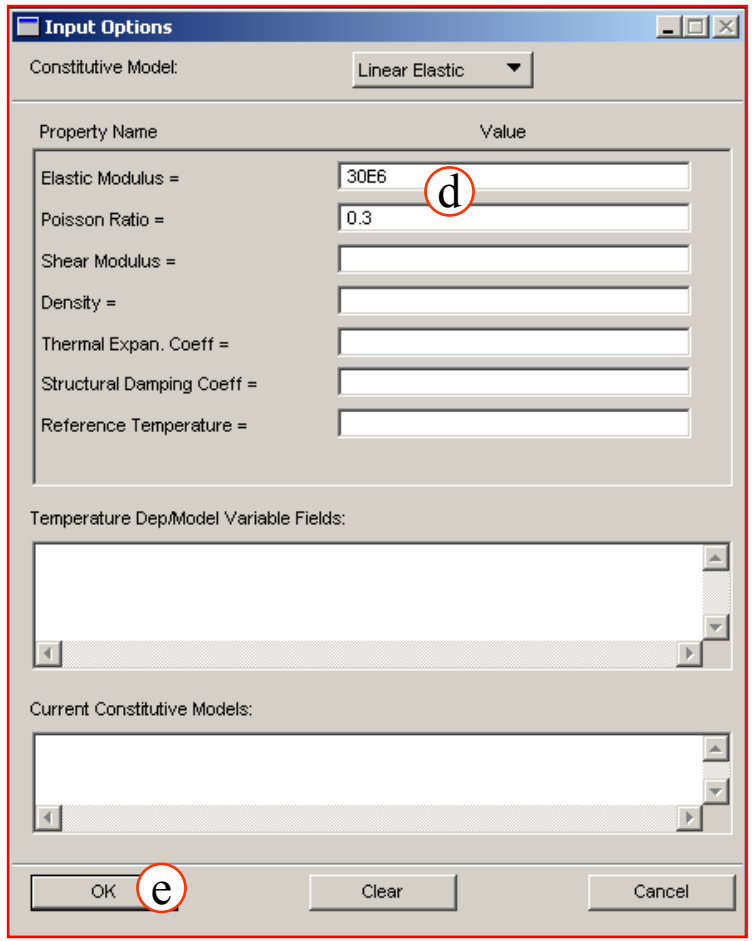
It may not seem as though the pressure load is applied uniformly throughout the top surface of the piston. That is because the application region was geometry, and not FEM. When this is done the markers only appear at geometry display lines.

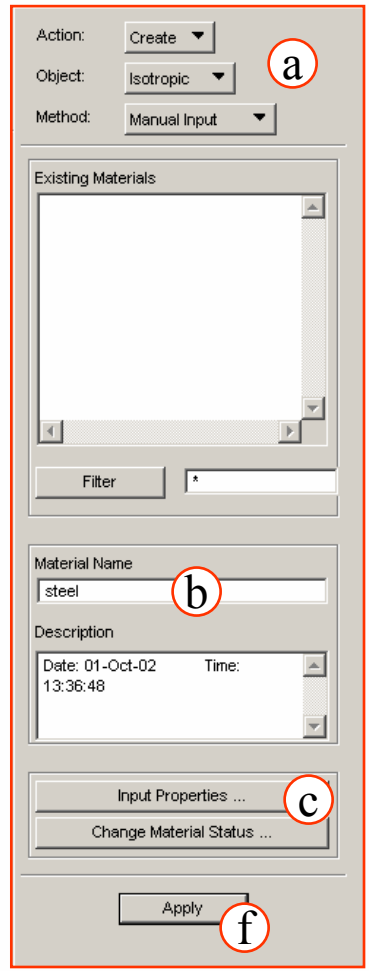
Step 7. Create Material Properties



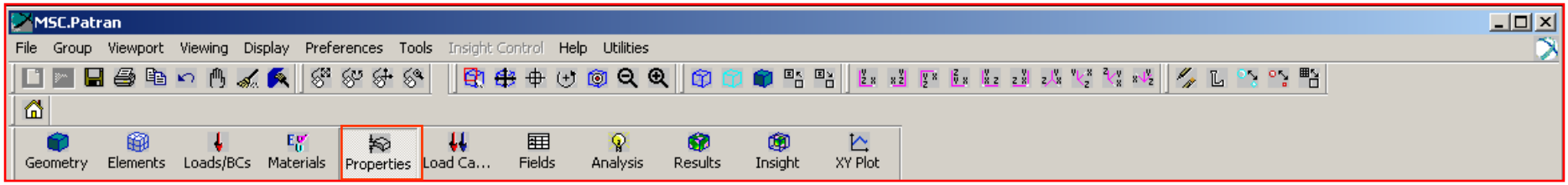
Create a material property for the piston.

a. Materials : Create / Isotropic/ Manual Input.
b. Enter **steel** for the *Material Name*.
c. Click on **Input Properties…**
d. Enter **30E6** and **0.3** for the *Elastic Modulus* and *Poisson Ratio*, respectively.
e. Click on **OK**.
f. Click on **Apply**.



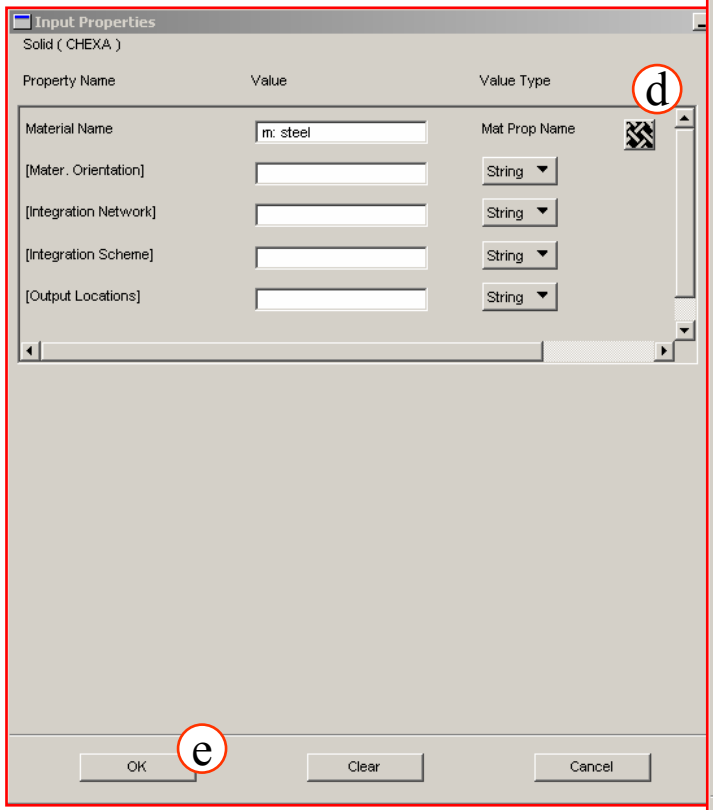


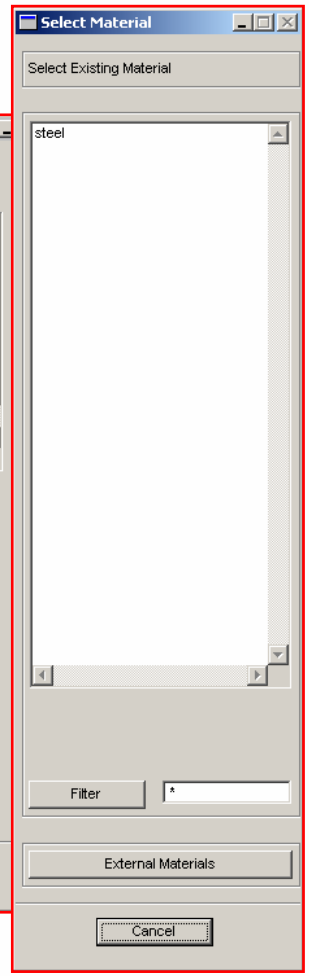
Step 8. Create Element Properties

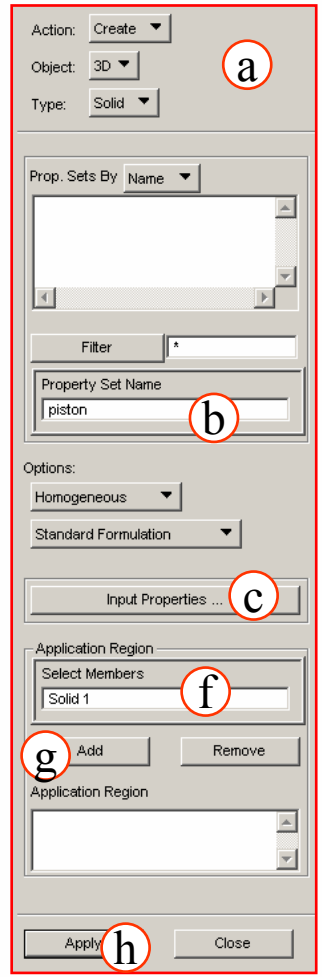


Create a 3D element property for the model.

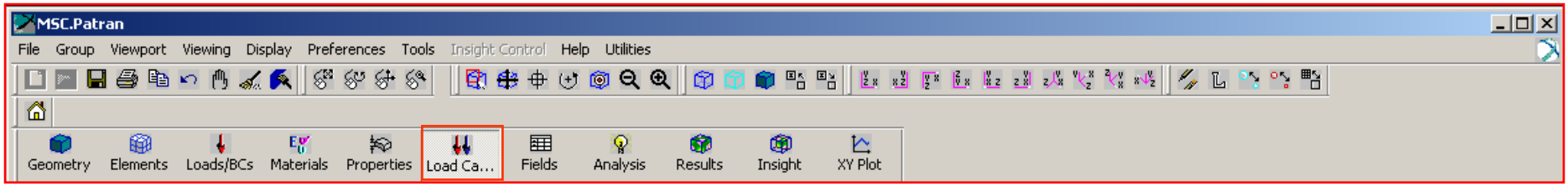
a. Properties : Create / 3D / Solid.
b. Enter **piston** for *Property Set Name*.
c. Click on **Input Properties…**
d. Click on **Mat Prop Name** icon, Choose *steel* from **Select Material**.
e. Click on **OK**.
f. Click on *Select Members* and select **Solid 1**.
g. Click on **Add**.
h. Click on **Apply**.





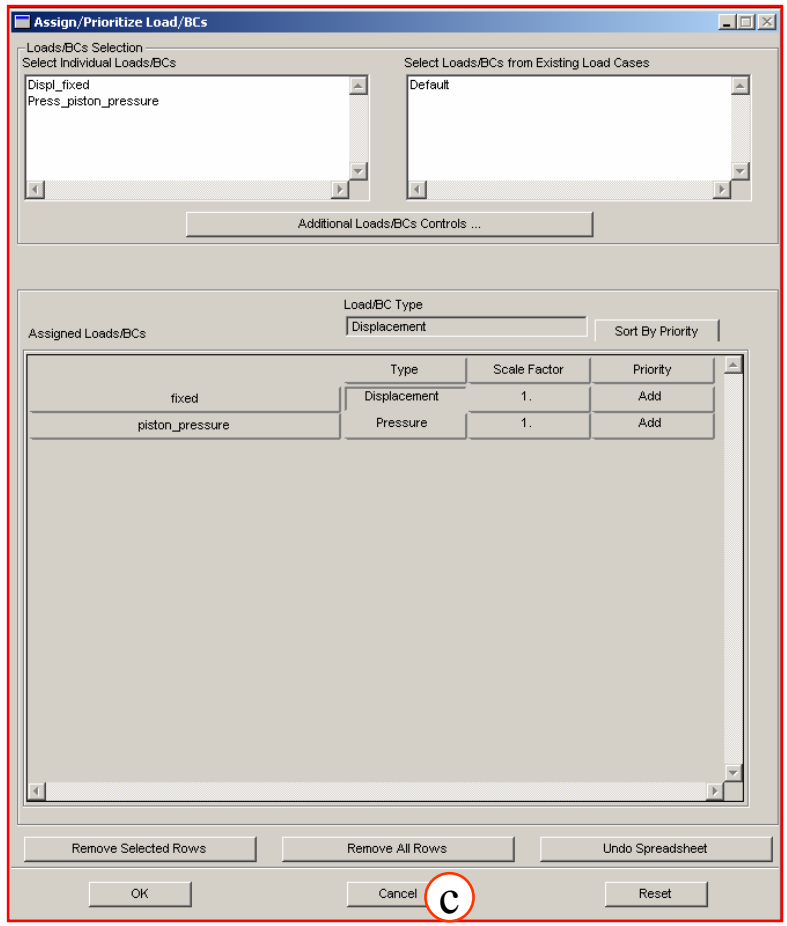


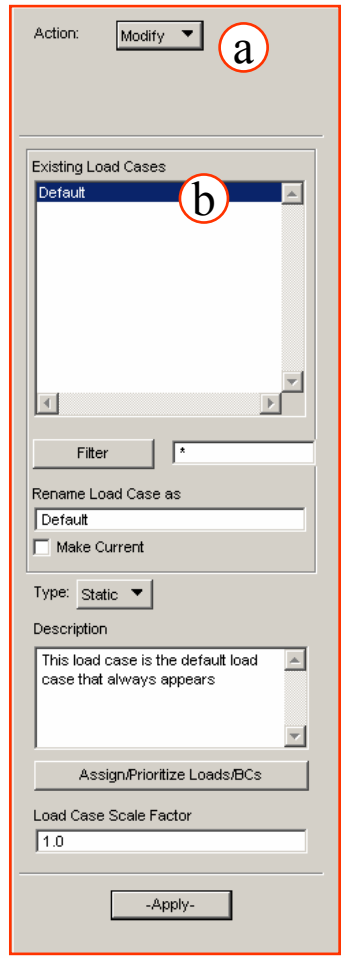
Step 9. Check the Load Cases



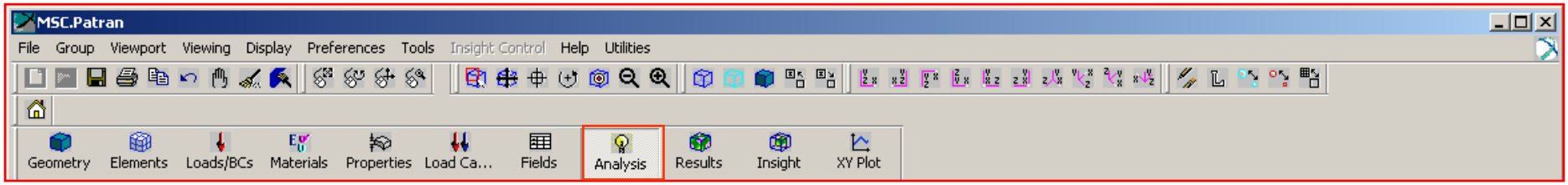
Check the load case Default and verify that the correct loads and boundary conditions are being applied.

a. Load Cases : Modify
b. Click on the **Default** load case.
c. Check to see that the correct loads are assigned and click **Cancel**.



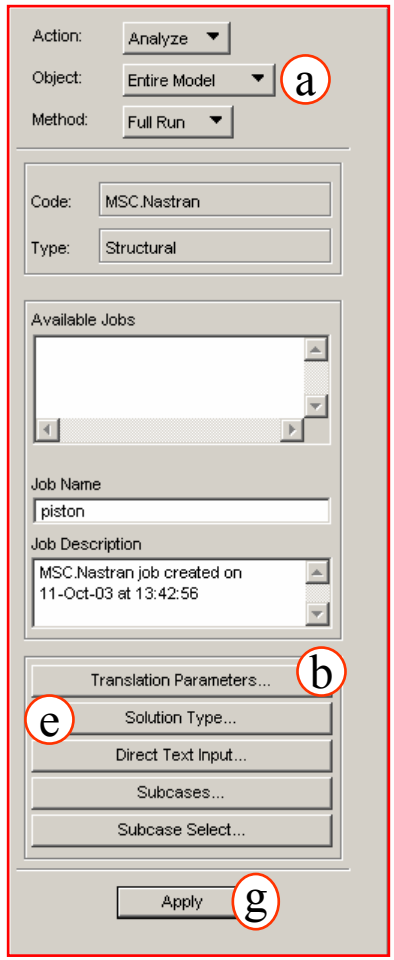


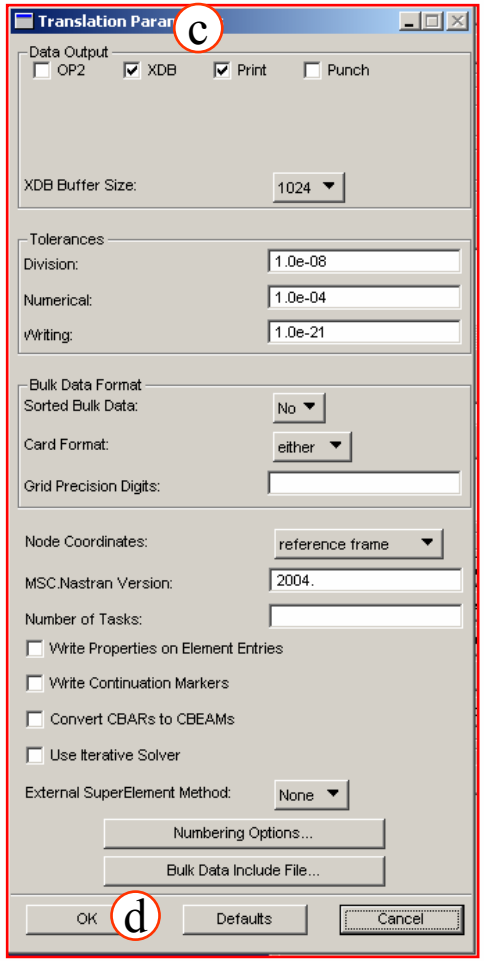
Step 10. Run the Analysis

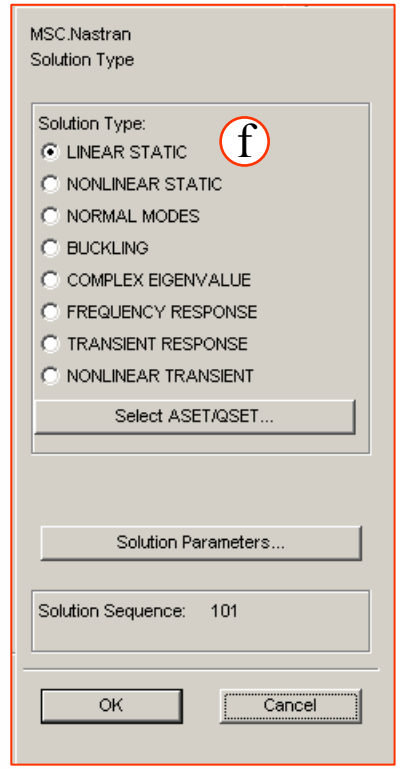


Send the model to MSC.Nastran and analyze the model.

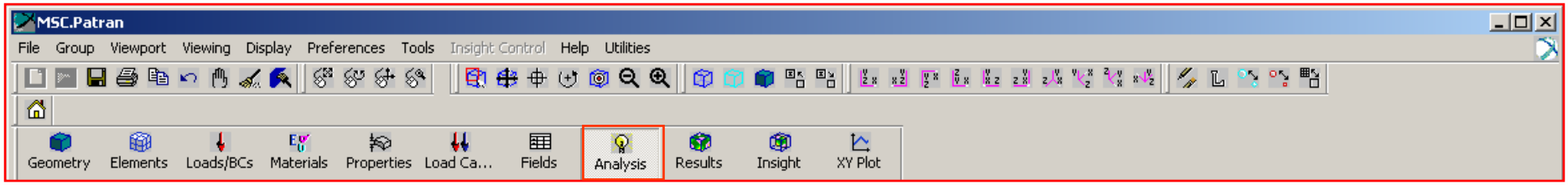
a. Analysis : Analyze / Entire Model / Full Run.
b. Click on **Translation Parameters…**
c. Set *Data Output* to **XDB** and **Print**.
d. Click on **OK**.
e. Click on **Solution Type…**
f. Set *Solution Type* to **Linear Static** and click **OK**.
g. Click on **Apply**.







Step 11. Read the Results



Read in the results file into MSC.Patran

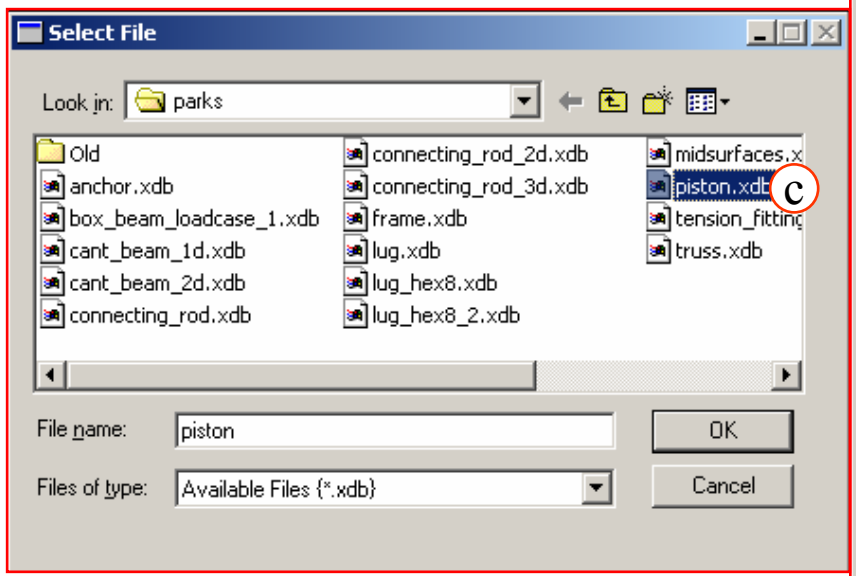
by attaching the XDB file.

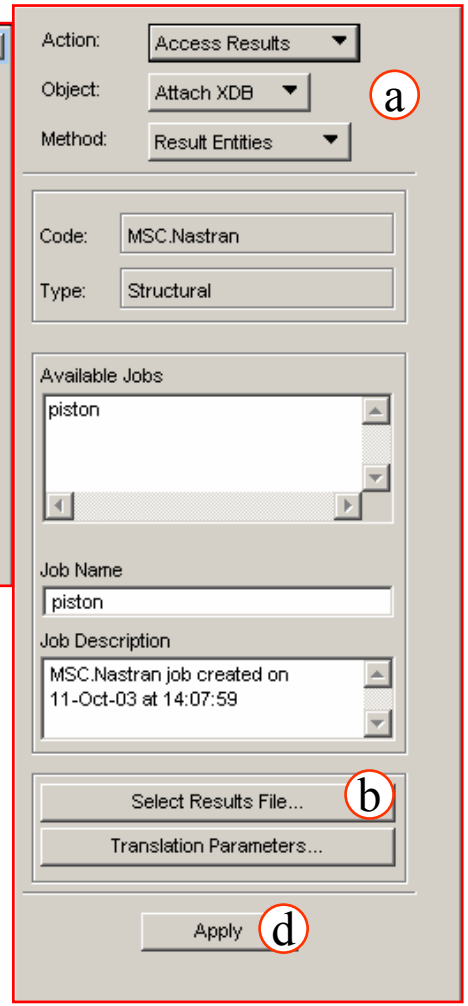
a. Analysis : Access Results / Attach XDB / Result Entities.

b. Click on **Select Results File…**

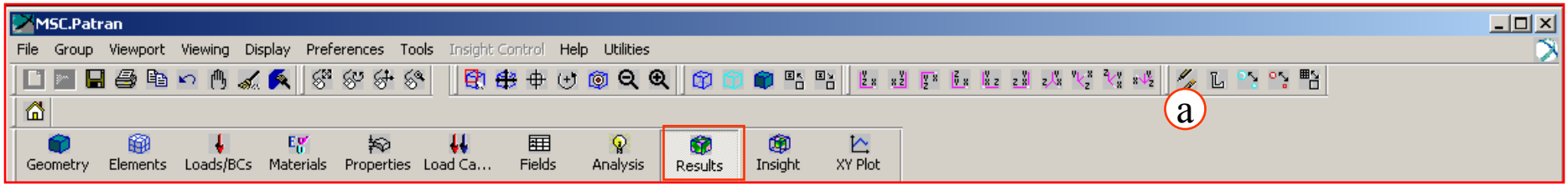
c. Select **piston.xdb** and click **OK**.

d. Click on **Apply.**



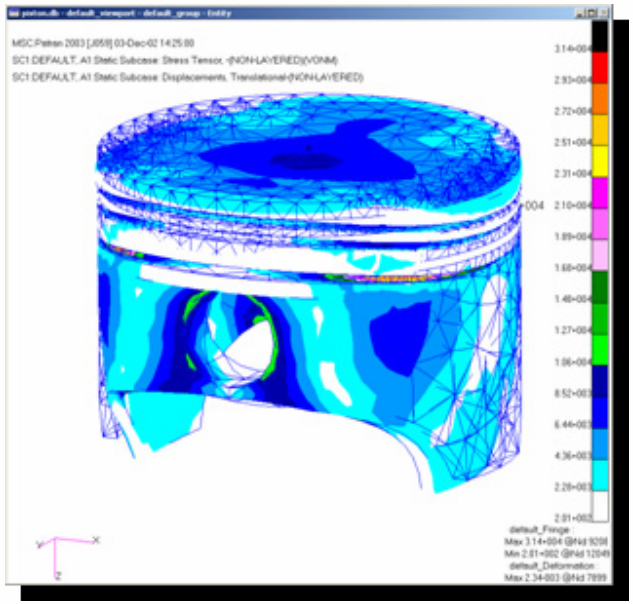


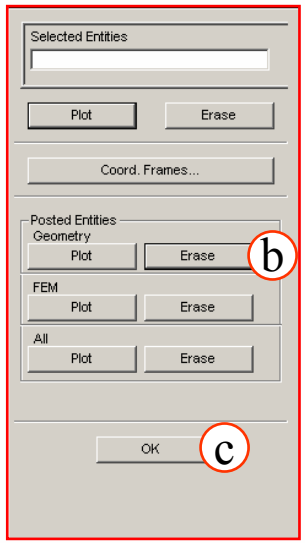
Step 11. Read the Results (Cont.)

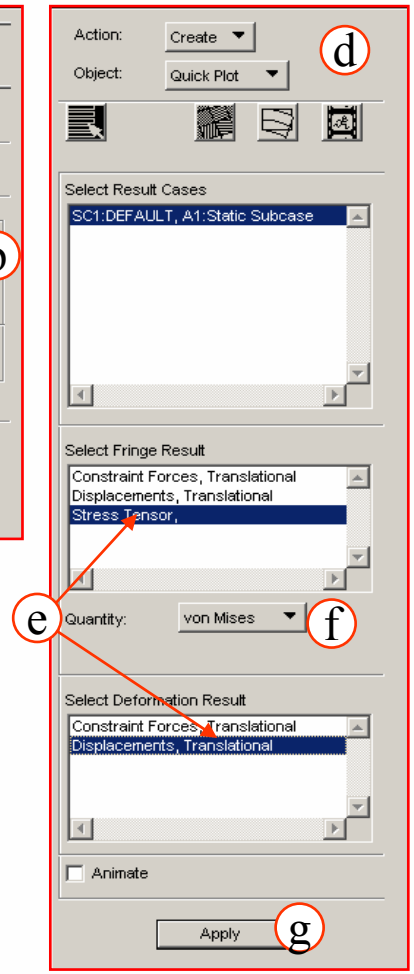


Plot both the deformation and the stress tensor using quick plot.

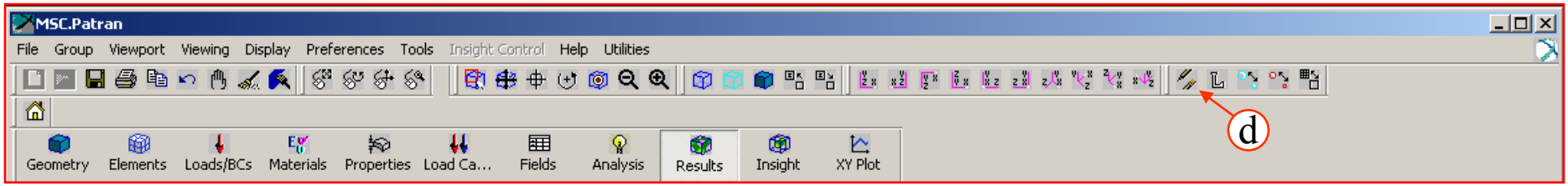
a. Click on the **Plot/Erase** icon.
b. Click on **Erase** under *Geometry*.
c. Click on **OK**.
d. Results : Create / Quick Plot.
e. Select **Stress Tensor** and **Displacements, Translational** from *Select Fringe Result* and *Select Deformation Result*, respectively.
f. Make sure that the *Quantity* set to **von Mises**.
g. Click on **Apply**.





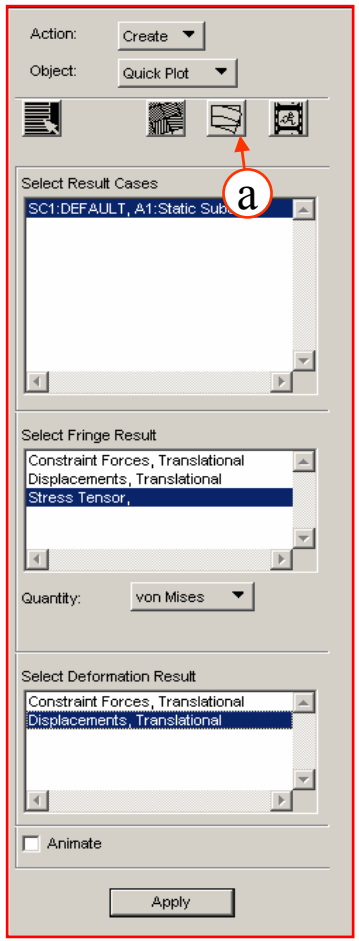


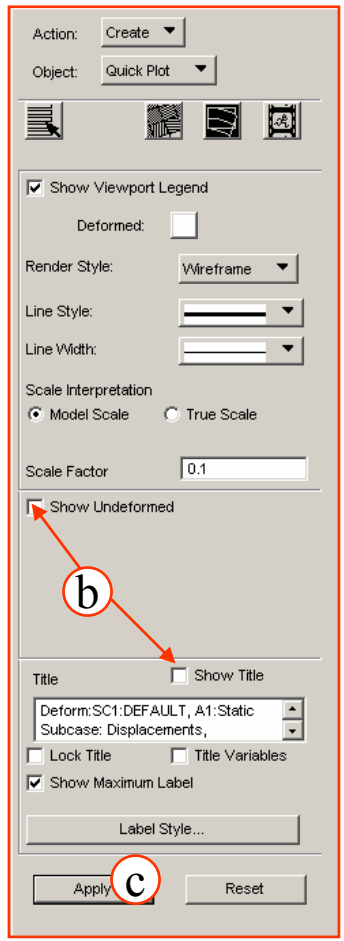
Step 11. Read the Results (Cont.)



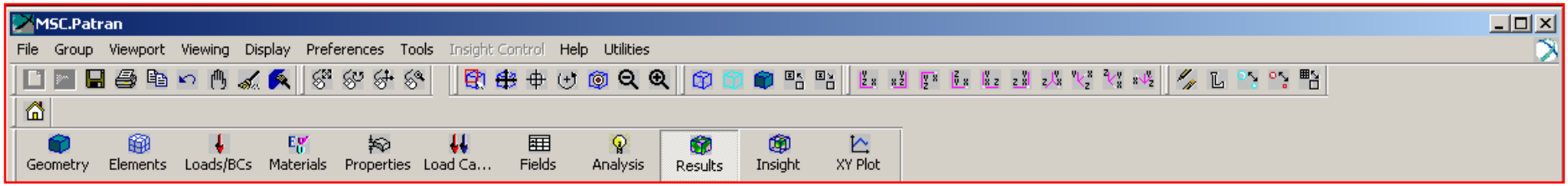
Modify the deformation attributes.

a. Click on the **Deform Attributes** icon.
b. Uncheck **Show Undeformed** and **Show Title**.
c. Click **Apply**.
d. Click on the **Plot/Erase** icon.
e. Click on **Erase** under *Geometry*.
f. Click on **OK**.



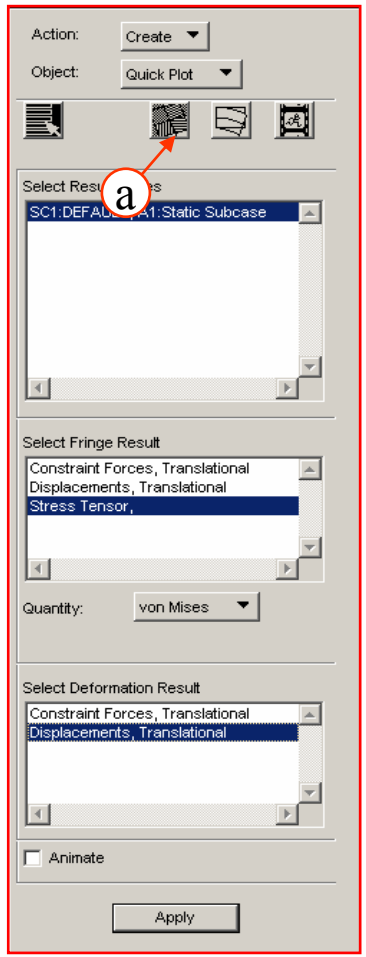


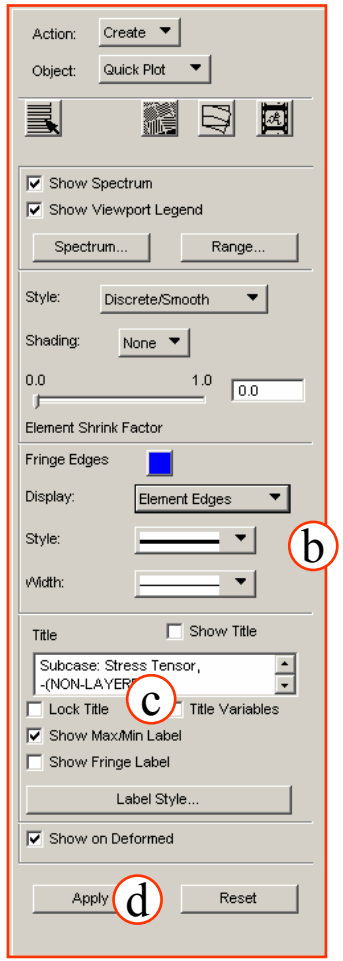
Step 11. Read the Results (Cont.)



Modify the fringe attributes.

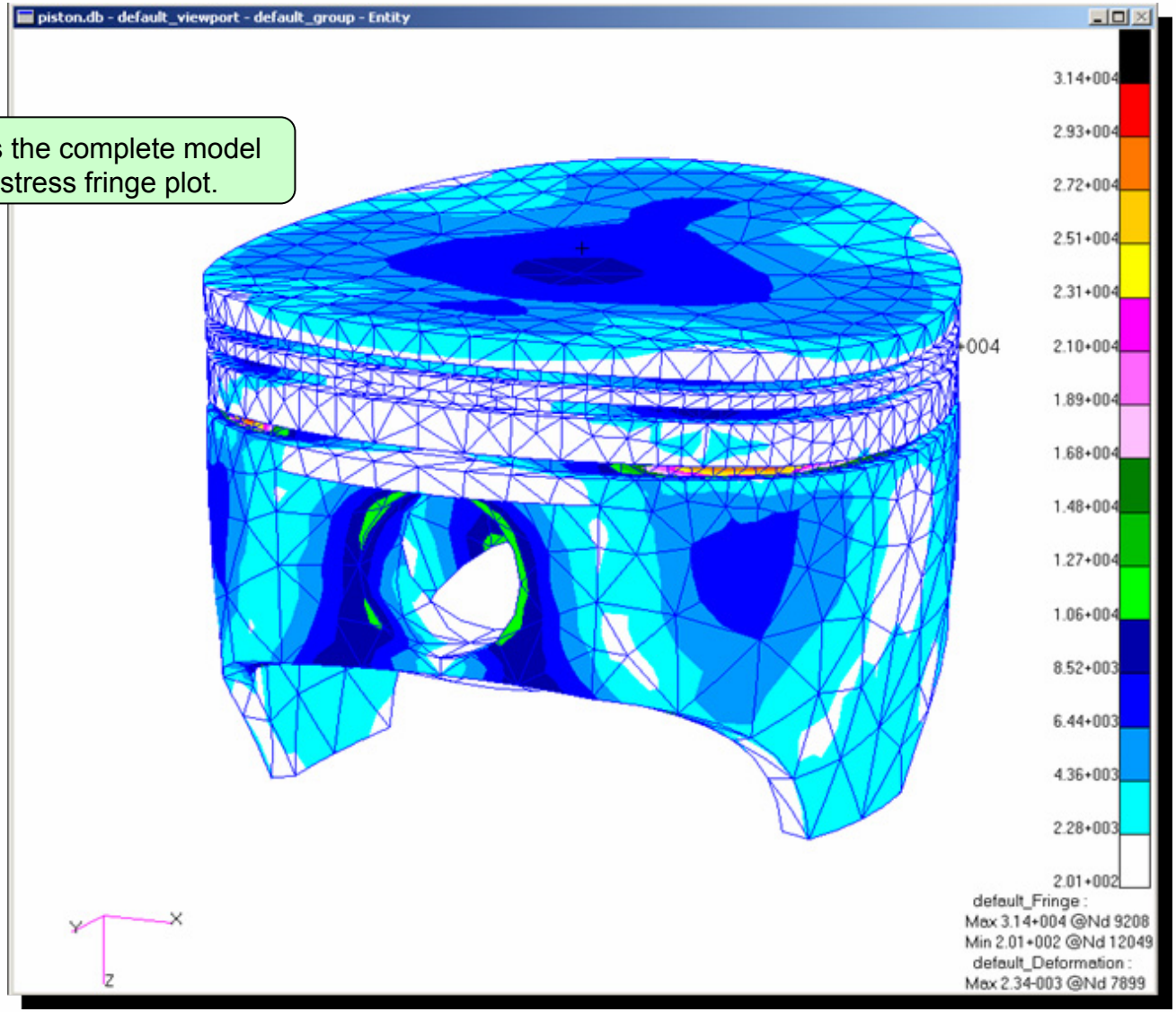
a. Click on the **Fringe Attributes** icon.
b. Set *Display* to **Element Edges**.
c. Uncheck **Show Title**.
d. Click on **Apply**.



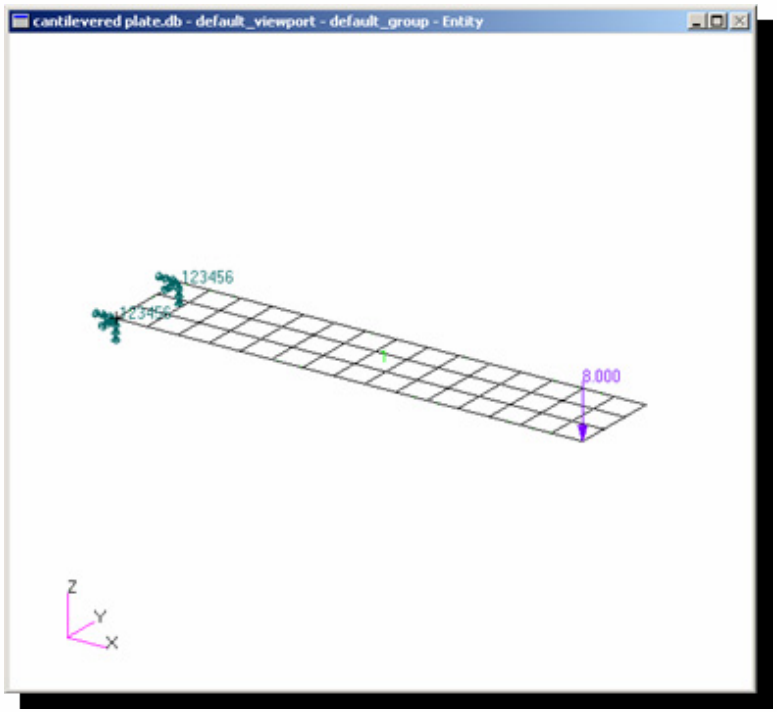


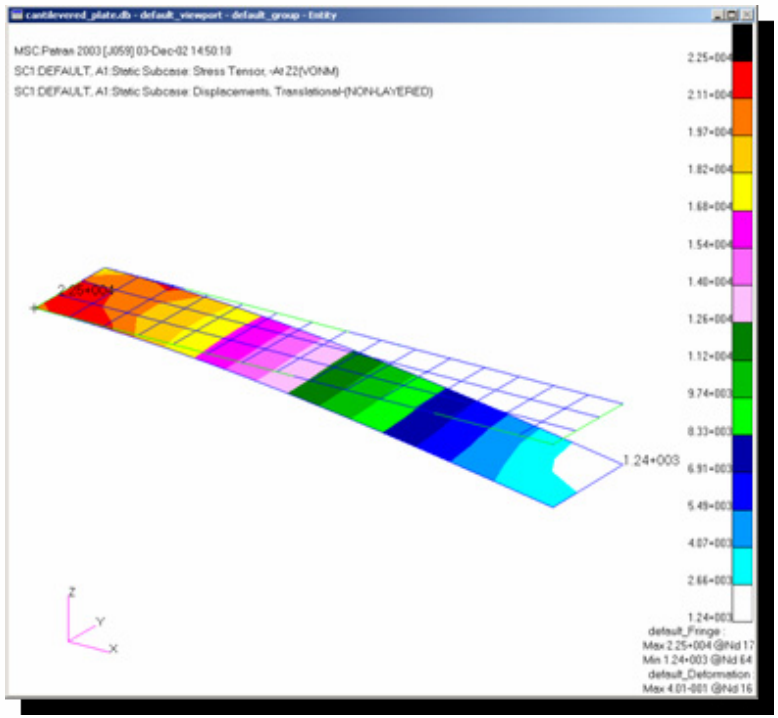
Step 11. Read the Results (Cont.)

Illustrated here is the complete model deformation and stress fringe plot.



WORKSHOP 2
CANTILEVERED PLATE

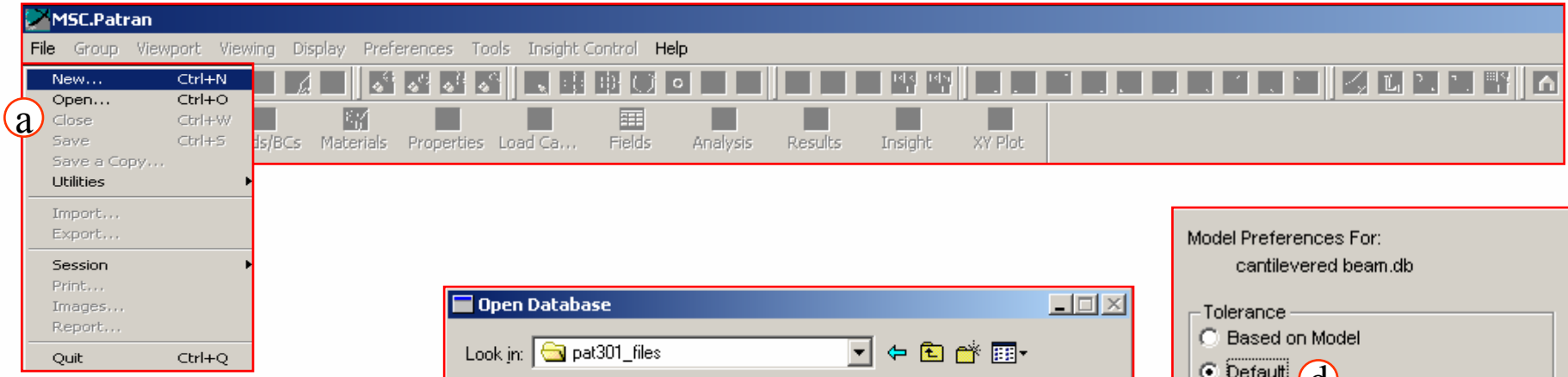


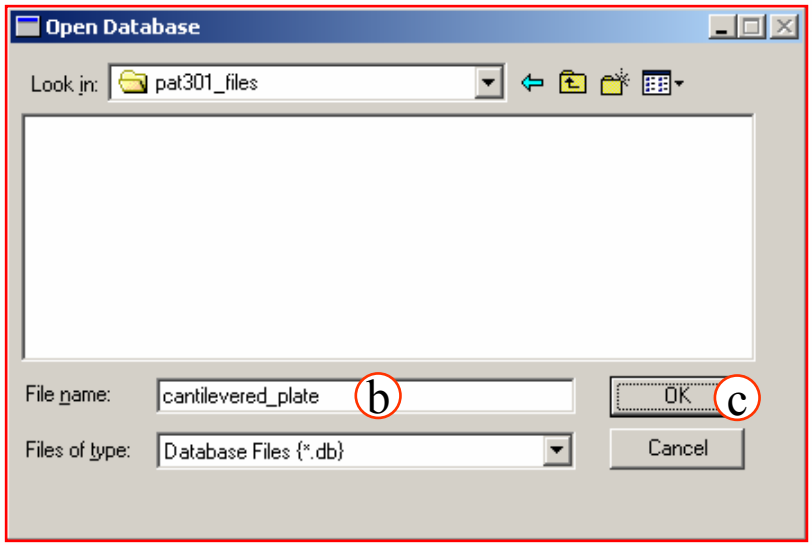


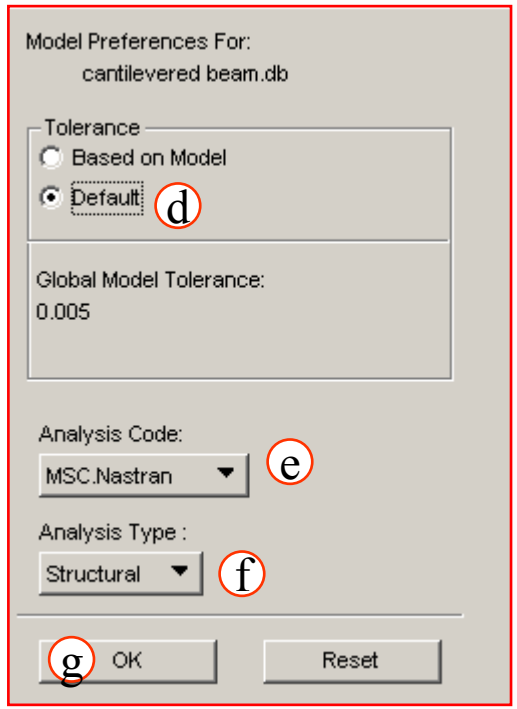
- **Workshop Objectives**
  - Model a cantilevered beam, using plate elements, subjected to a constant tip loading. The plate elements will undergo bending. MSC.Patran and MSC.Nastran will be used to create or analyze the linear static model.
- **Problem Description**
  - Display the component of stress in the axial direction (direction along the length of the plate structure) and the deformed shape of the plate model.
  - Plate material: Aluminum with $E = 10 \times 10^6$ psi and $\nu = 0.3$
  - Load on tip of plate = 8 lbf
- **Software Version**
  - MSC.Patran 2005r2
  - MSC.Nastran 2005r2b

- **Key Concepts and Steps:**
  - **Database:** create a new database with Analysis Code = MSC.Nastran and Analysis Type = Structural
  - **Geometry:** create a geometric surface to represent the plate
  - **Elements:** mesh the surface with plate elements
  - **Loads/BCs:** constrain one end of the plate model, and apply the constant tip load to the other end of the plate model.
  - **Materials:** specify an isotropic material for Aluminum
  - **Properties:** create a 2D shell/plate property
  - **Analysis:** Solution Type = Nastran Linear Static, Solution Sequence = 101, Method = Full Run
  - **Analysis:** access analysis results by attaching the XDB file to database
  - **Results:** plot X-component of stress and displacement results

Step 1. Create a Database



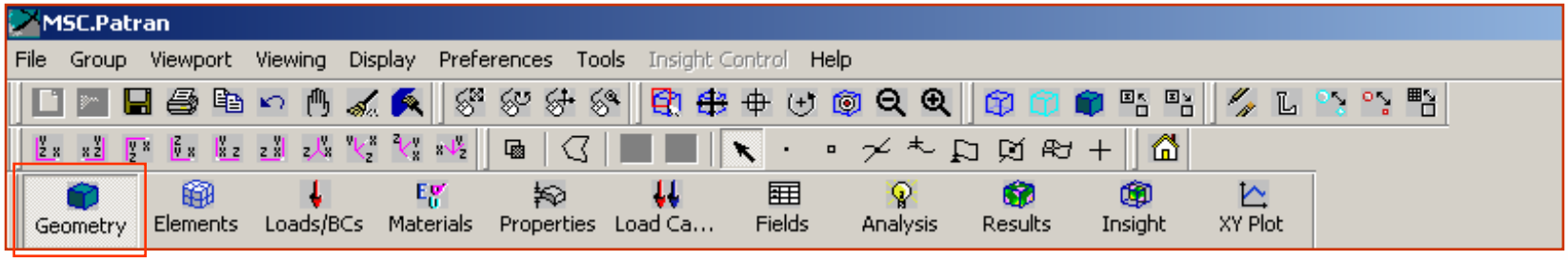




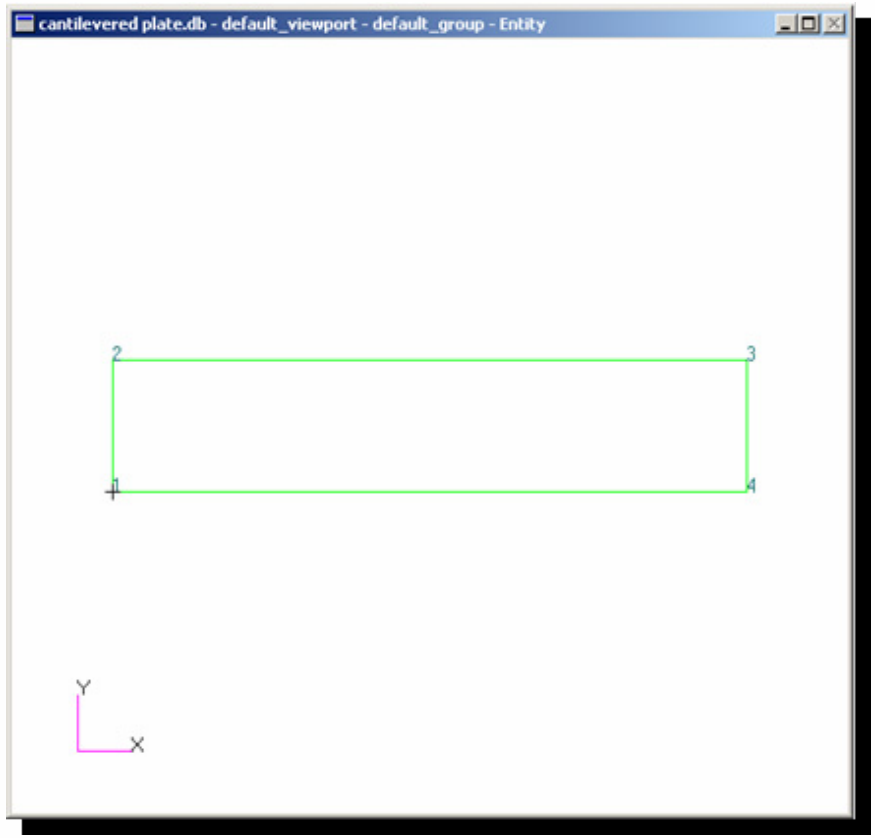
Create a new database.

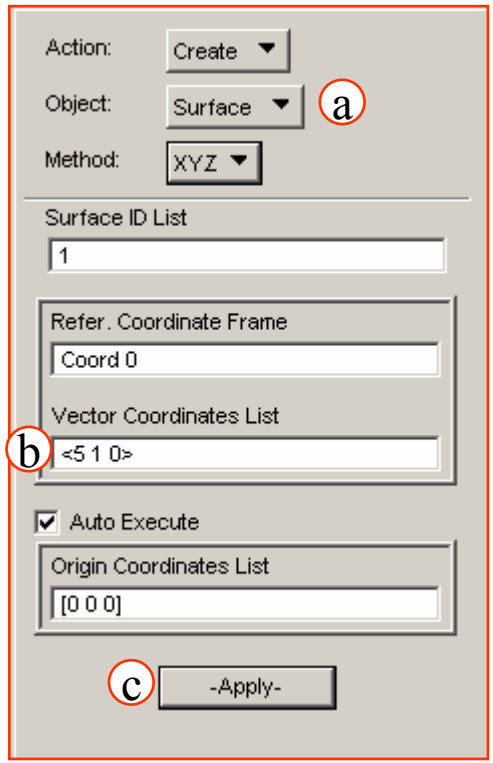
a. File / New.
b. Enter **cantilevered_plate** as the file name.
c. Click **OK**.
d. Choose **Default** Tolerance.
e. Select **MSC.Nastran** as the Analysis Code.
f. Select **Structural** as the Analysis Type.
g. Click **OK**.

Step 2. Create Geometry of the Plate

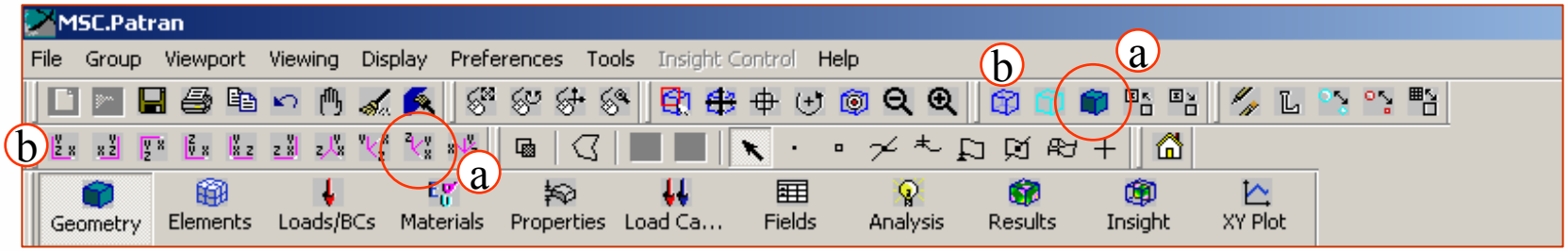


a. Geometry: Create / Surface / XYZ.
b. Select on **Vector Coordinates List** and enter **< 5 1 0 >**.
c. Apply.

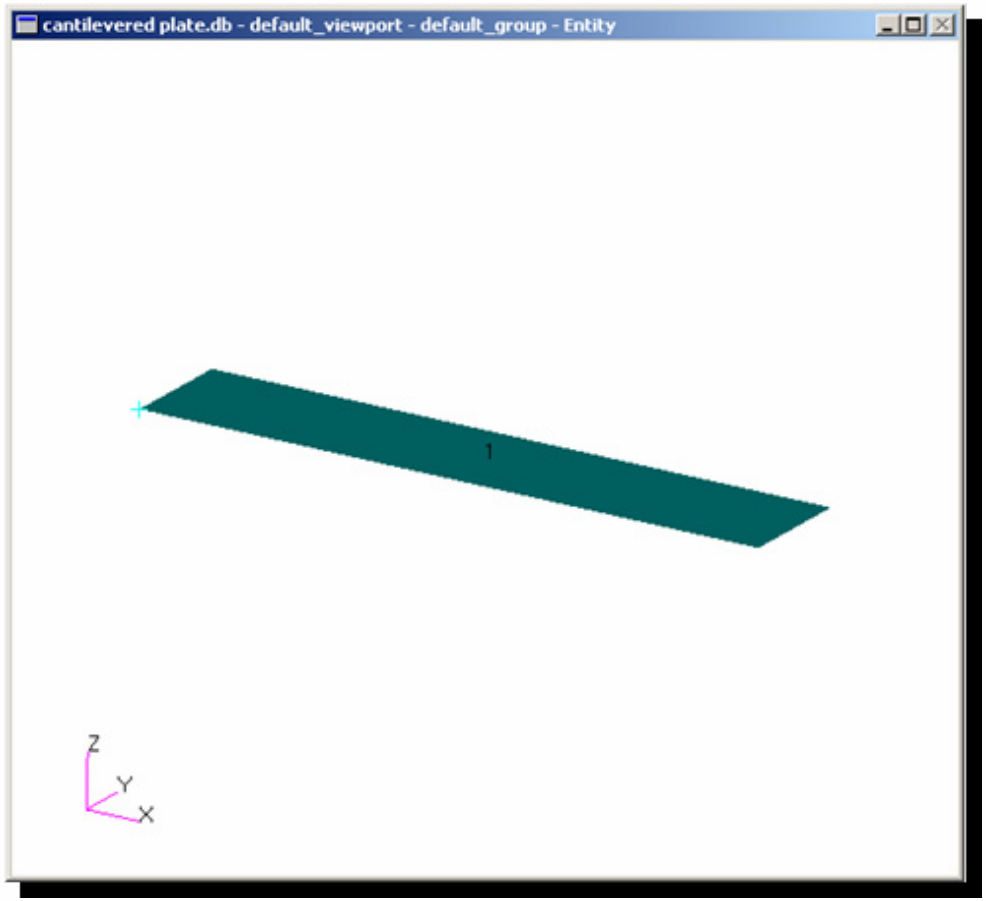




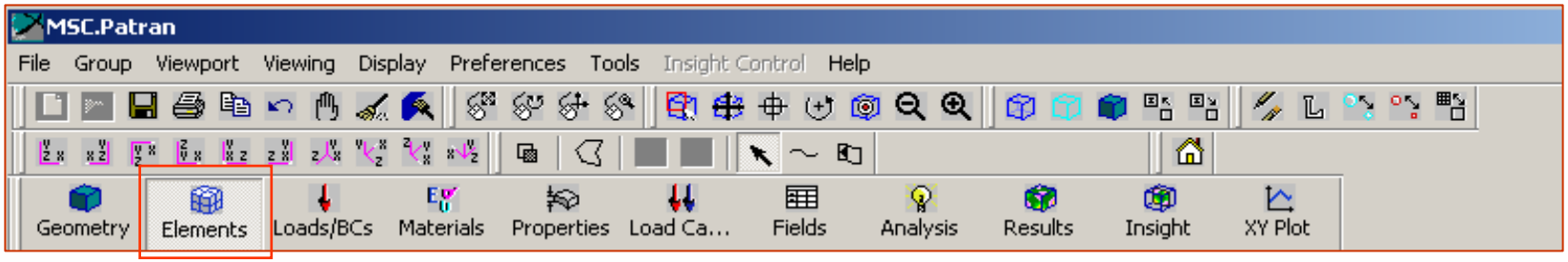
Step 2. Create Geometry of the Plate (Cont.)



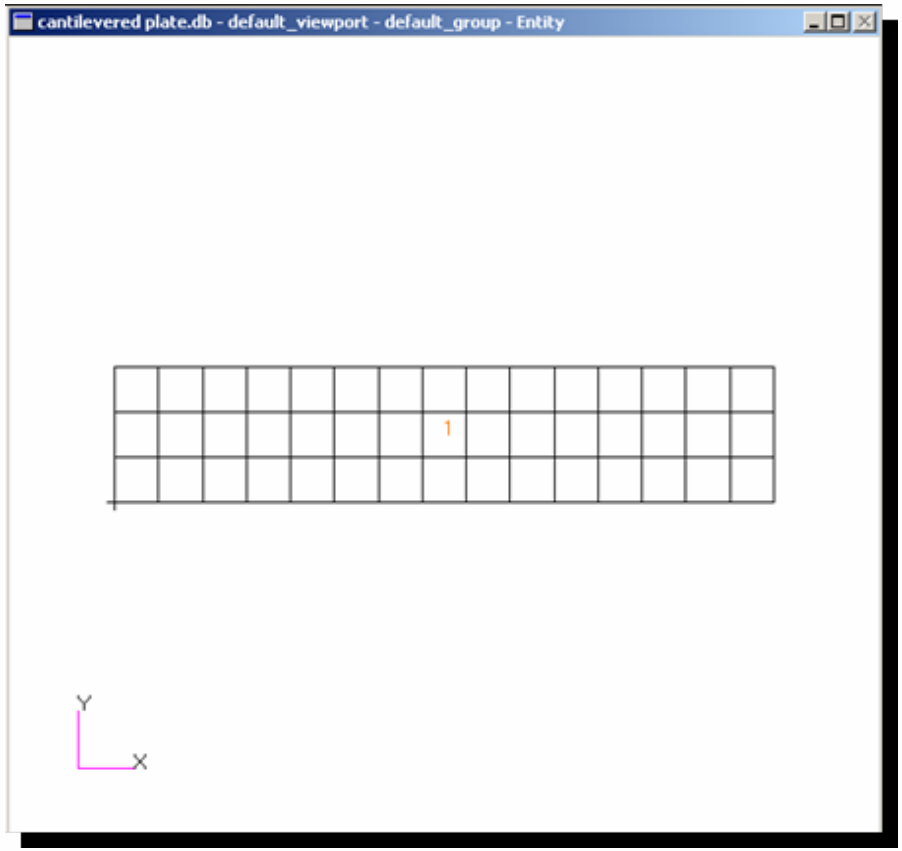
a. Select **Smooth Shade** and **Iso 3** View.
b. Change back to **Wireframe** and **Front view**.

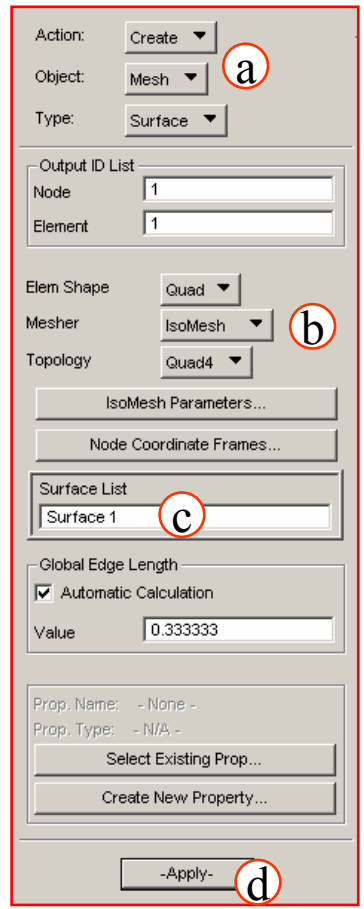


Step 3. Meshing with Quad4 Elements

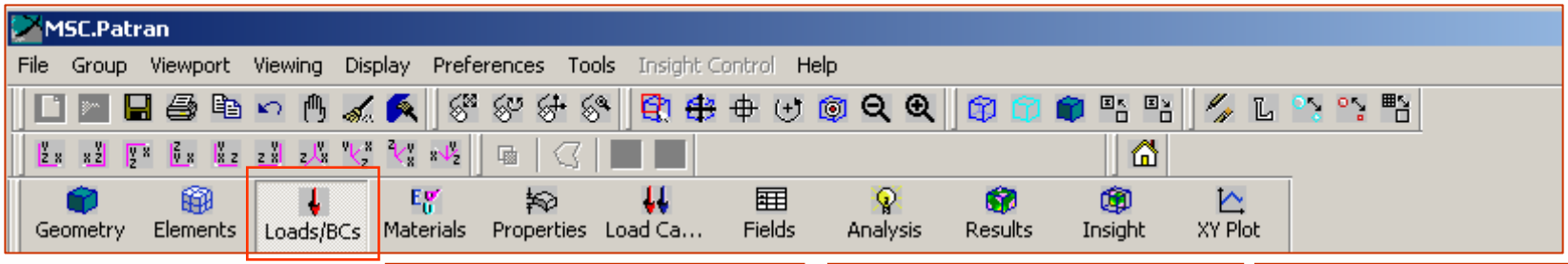


a. Elements: Create / Mesh / Surface.
b. Select
   Elem Shape: **Quad**.
   Mesher: **IsoMesh**.
   Topology: **Quad4**.
c. Click on *Surface List* and select **Surface 1**.
d. Apply.



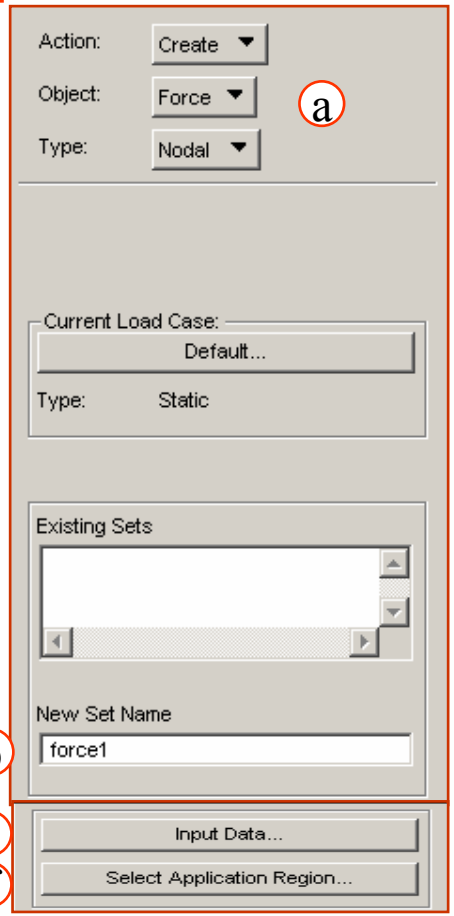


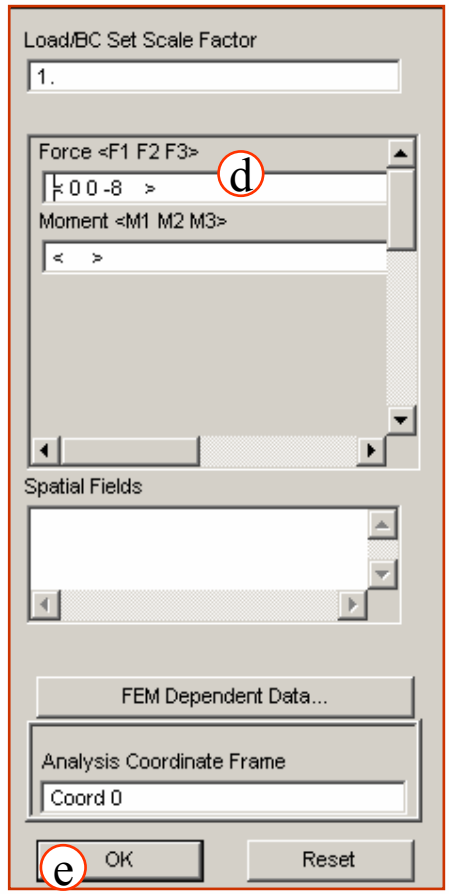
Step 4. Create a Force at Free End

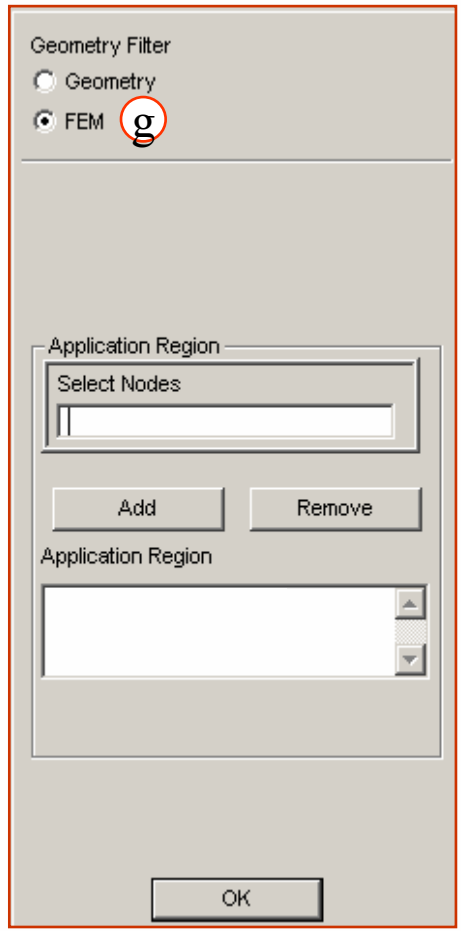


A force will be applied to a node at the end of the cantilever plate.

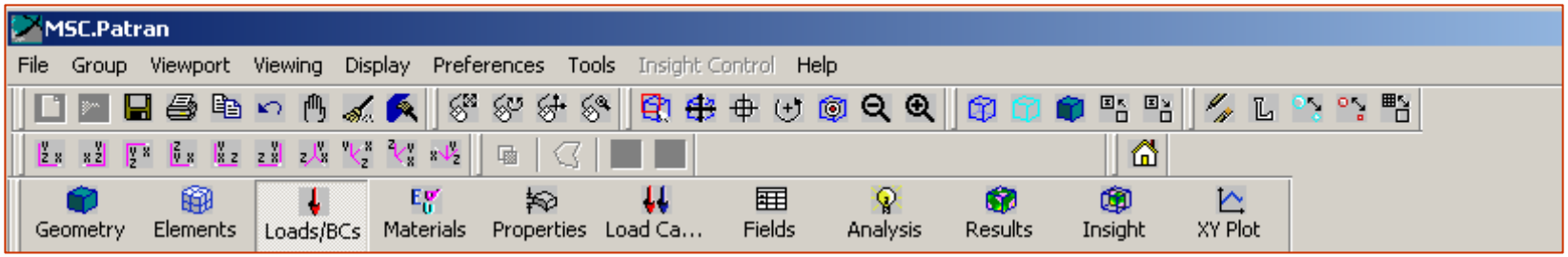
a. Loads / BCs: Create / Force/ Nodal.
b. Select on *New Set Name* and enter **force1**.
c. Select **Input Data**.
d. Enter **< 0 0 –8 >** on *Force < F1 F2 F3 >*.
e. OK.
f. Click **Select Application Region**.
g. Select **FEM** on *Geometry Filter*.



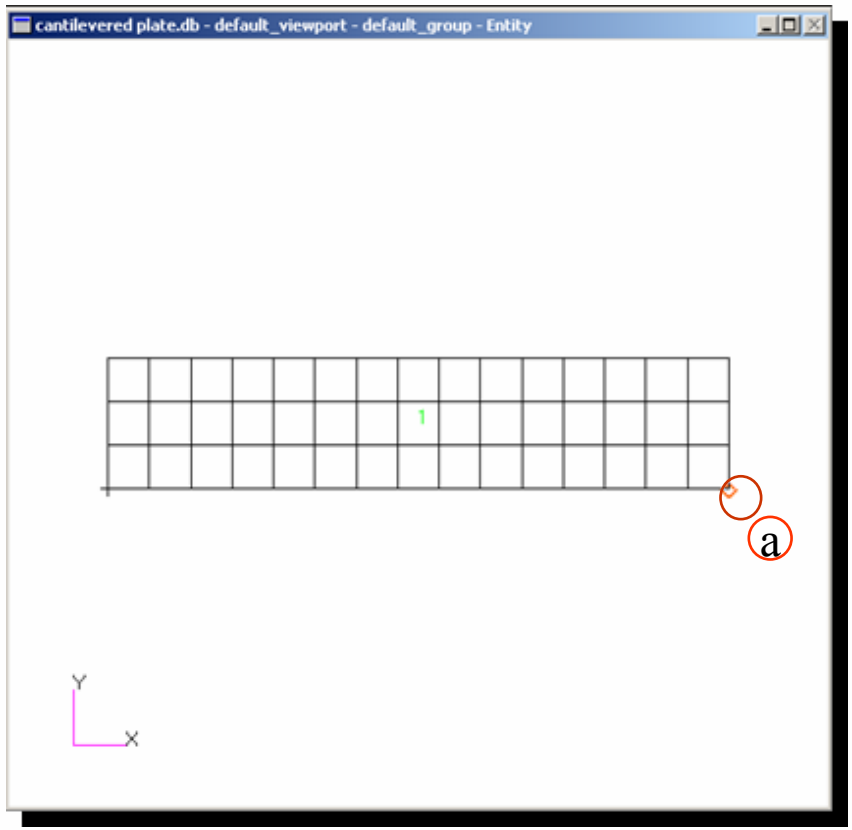


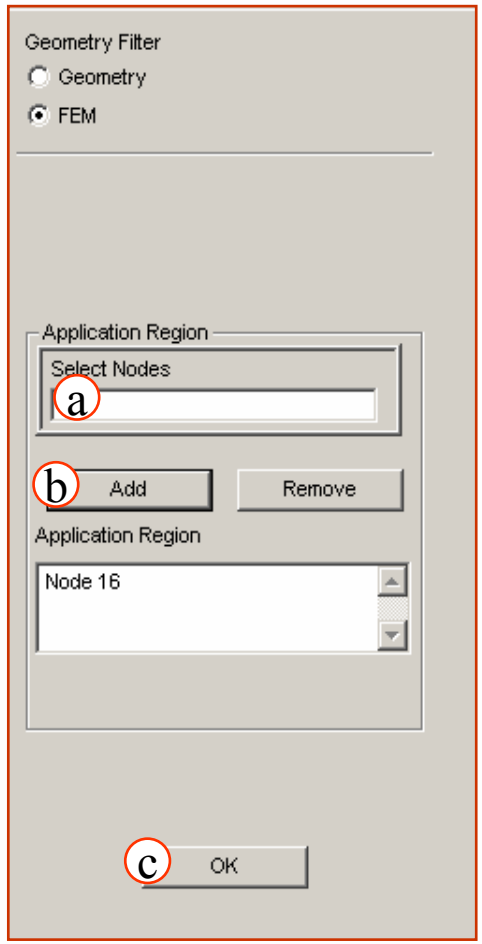


Step 4. Create a Force at Free End (Cont.)

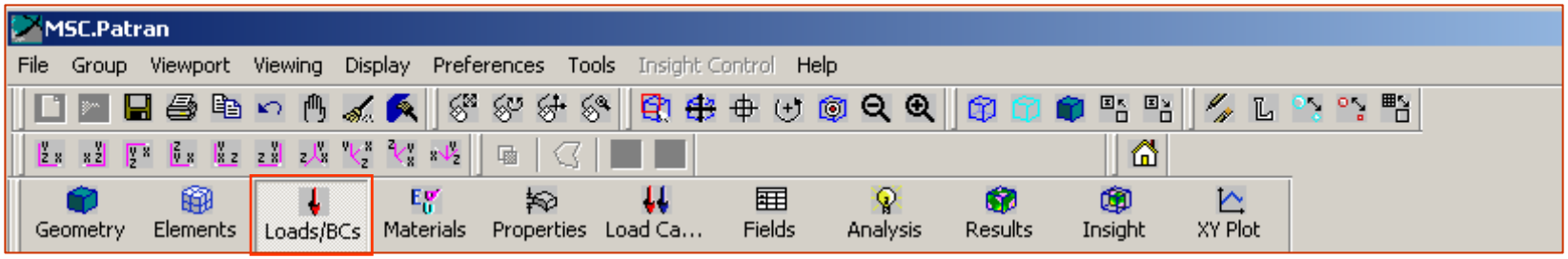


a. Click on *Select Nodes* and select the **lower right corner** node as shown in the figure.
b. Add.
c. OK.
d. Apply.



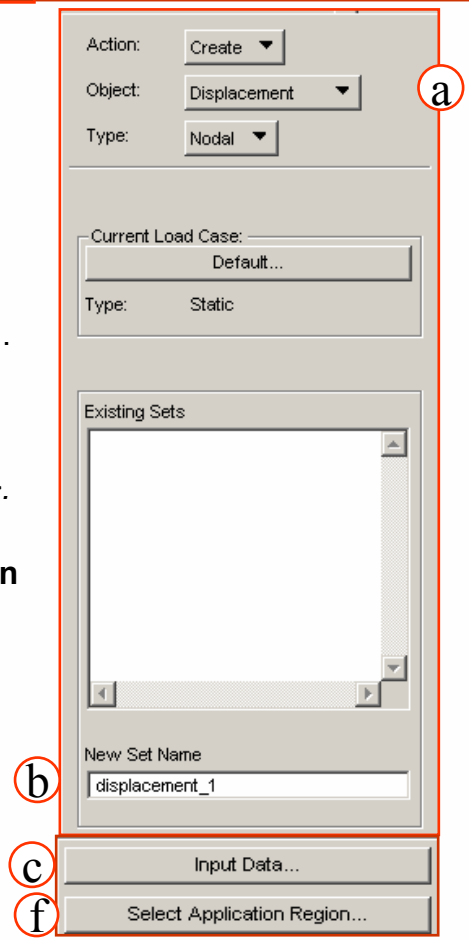


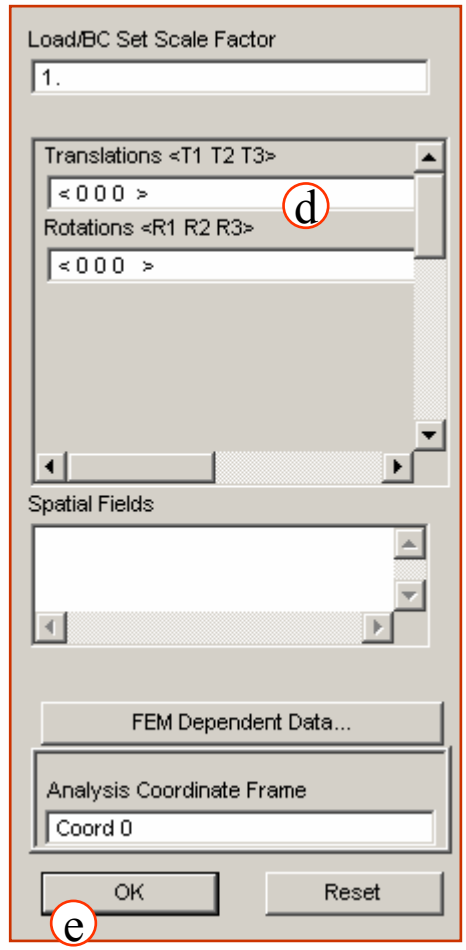
Step 5. Create Constraints on the Plate

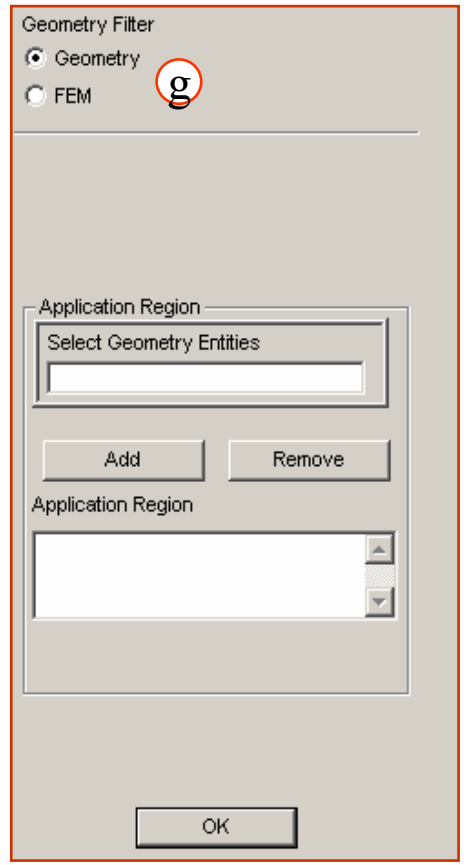


Constrain the other end of the plate, fixing all six degrees of freedom at each node.

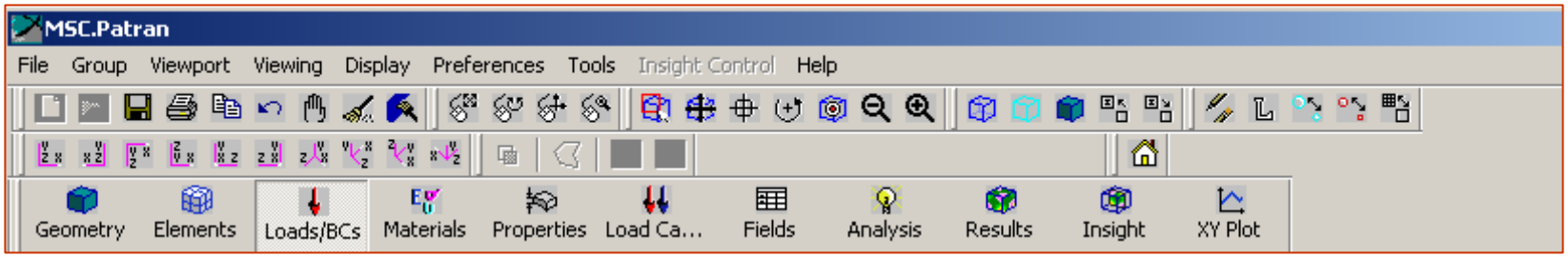
a. Loads / BCs: Create / Displacements / Nodal.
b. Select on *New Set Name*: and enter **displacement_1**.
c. Select **Input Data**.
d. Enter **<0 0 0>** for *Translations <T1 T2 T3 >* and *Rotations <R1 R2 R3>.*
e. OK.
f. Click on **Select Application Region**.
g. Select **Geometry** for *Geometry Filter*.



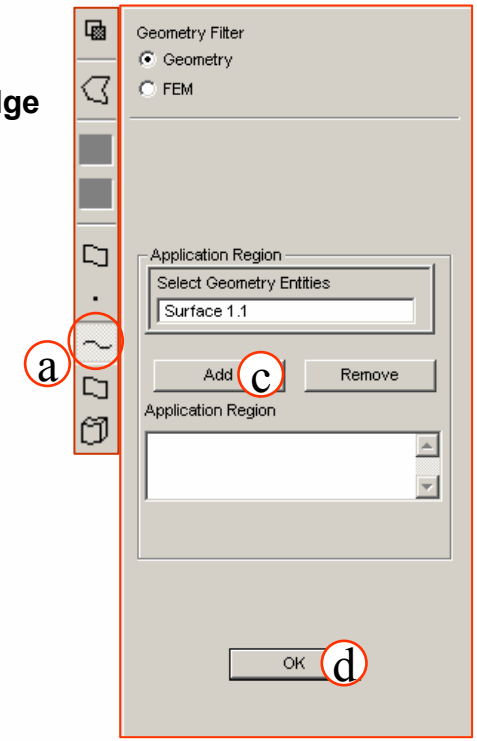


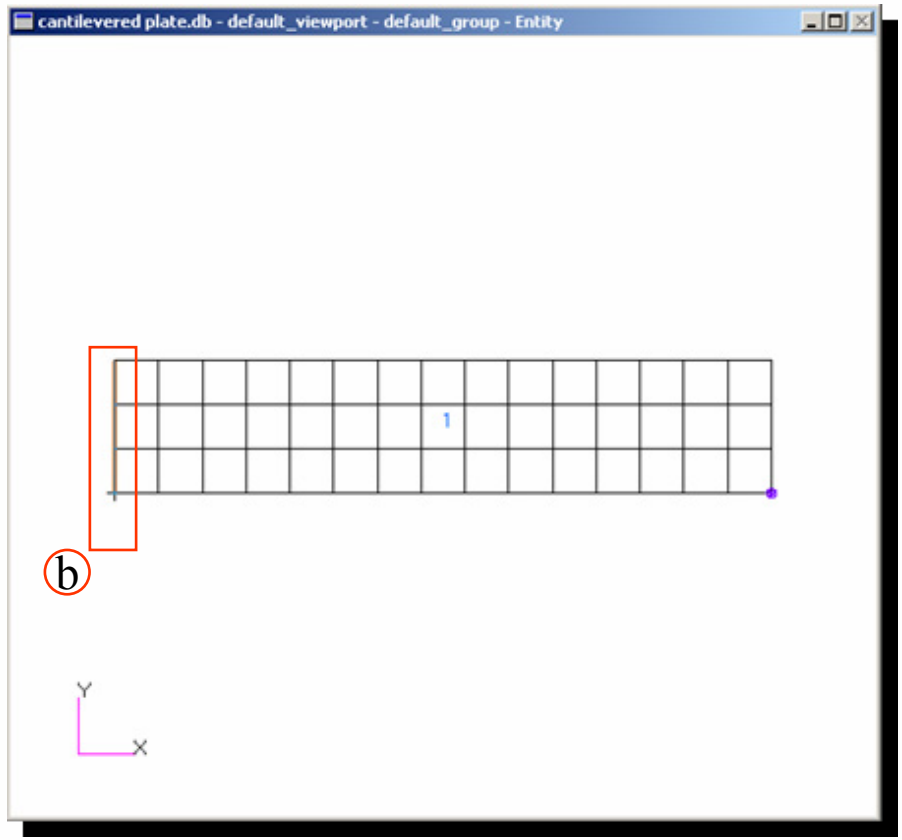


Step 5. Create Constraints on the Plate (Cont.)

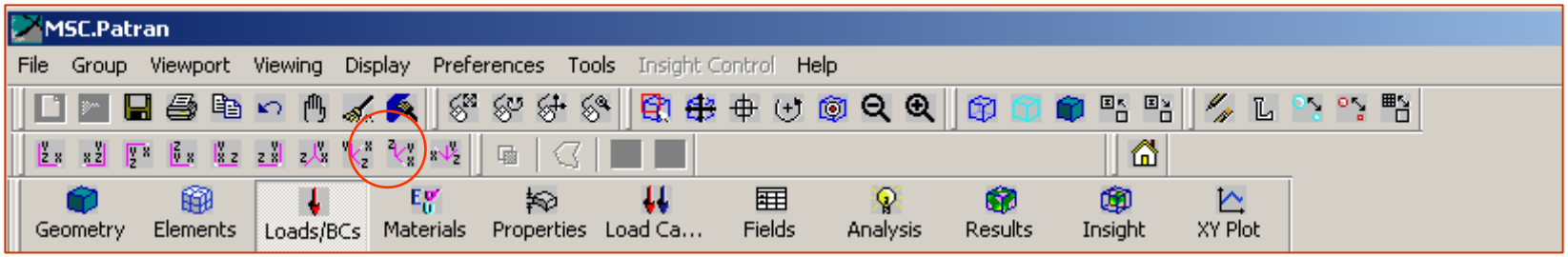


a. Pick the **Curve or Edge** icon.
b. Select on the edge shown in the figure.
c. Add.
d. OK.
e. Apply.

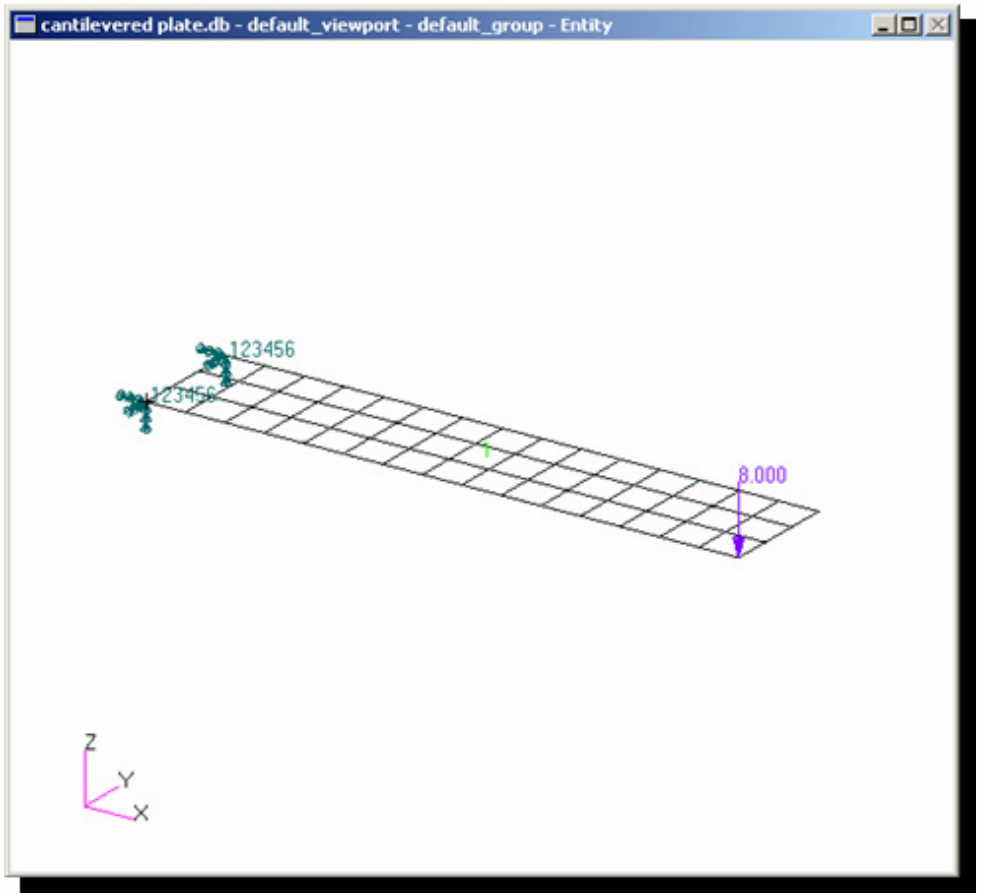




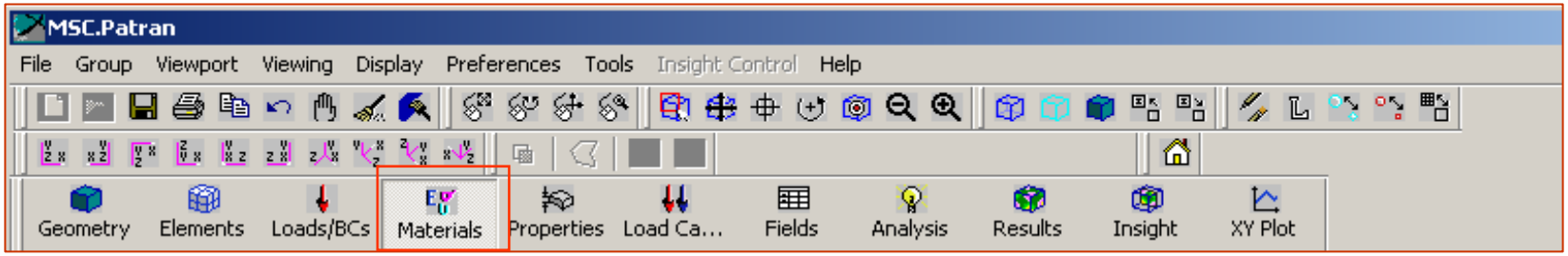
Step 5. Create Constraints on the Plate (Cont.)



a. Select on **Iso3 view** from the tool bar. Your model should look like the following.

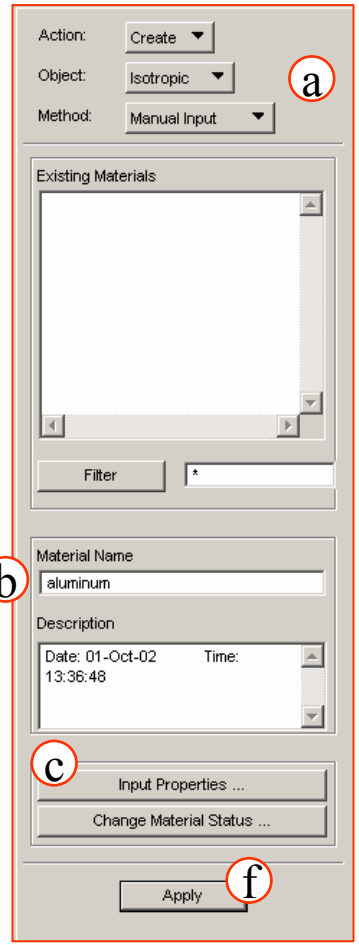


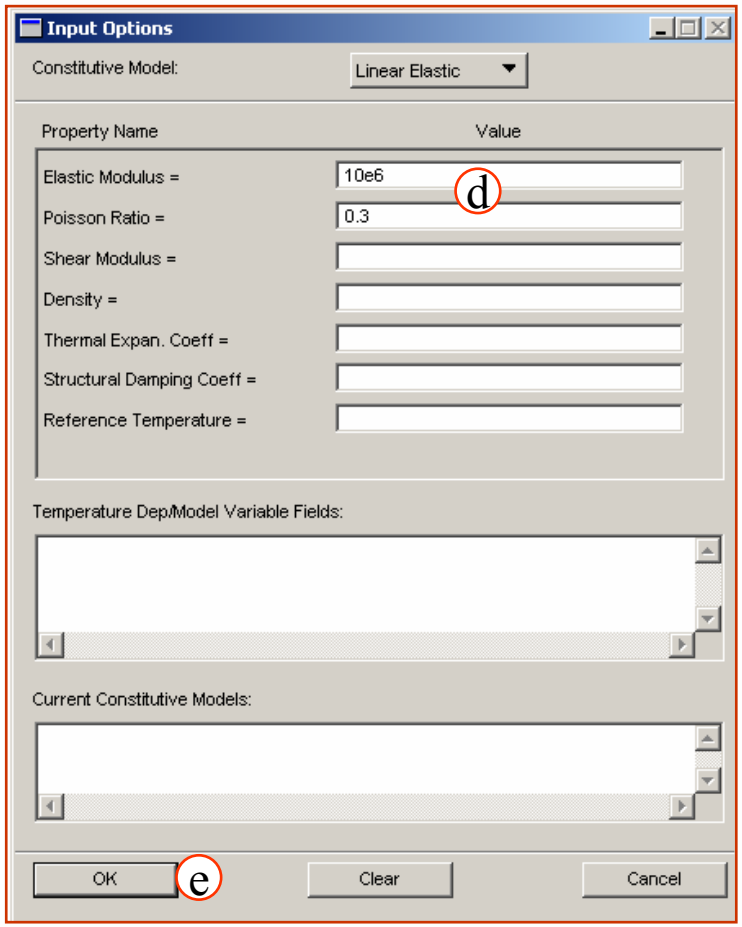
Step 6. Defining the Material



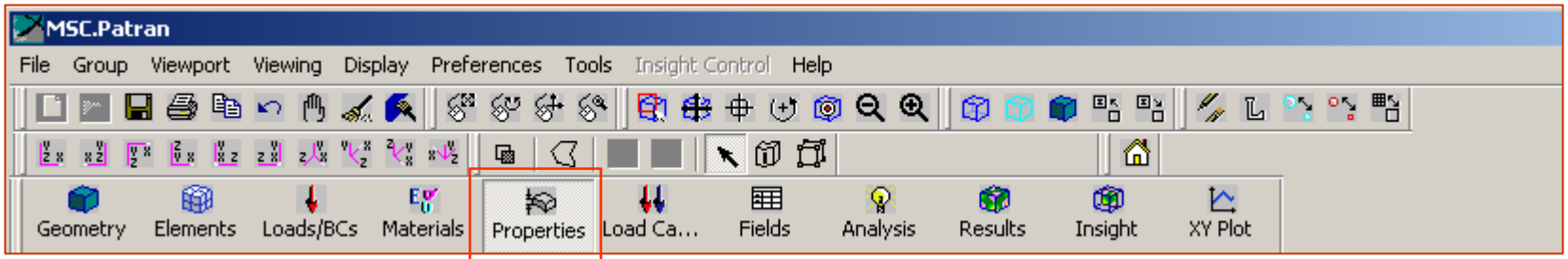
We will set aluminum as the material of the plate.

a. Materials: Create / Isotropic / Manual Input.
b. Select on *Material Name* and enter **aluminum**.
c. Select **Input Properties**.
d. Enter:
   Elastic Modulus: **10e6**.
   Poisson Ratio: **0.3**.
e. OK.
f. Apply.

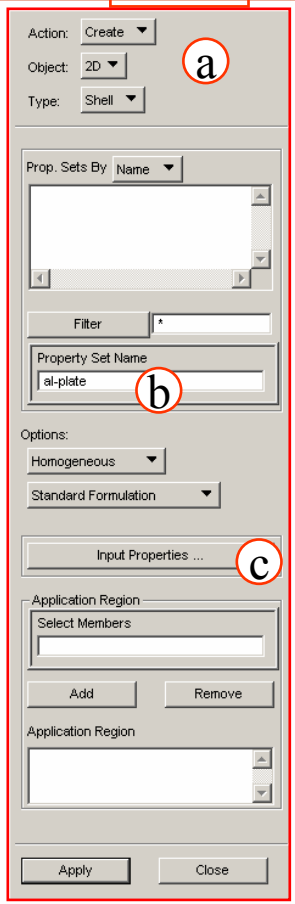


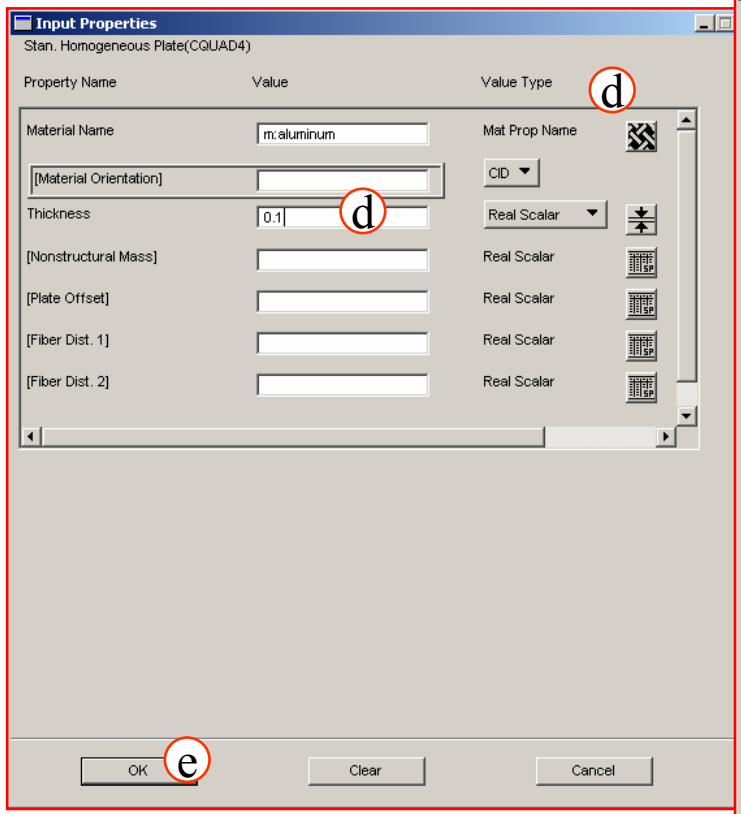


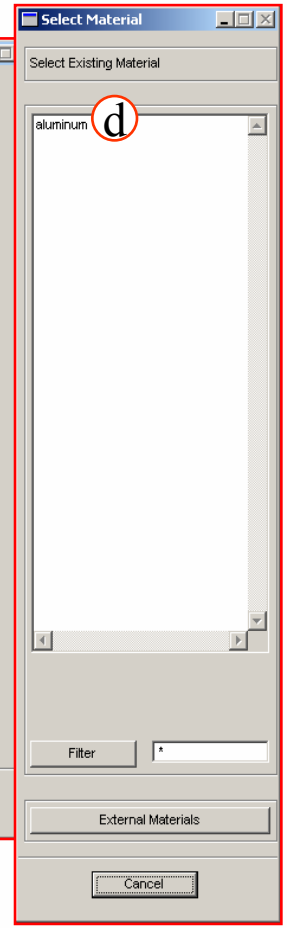
Step 7. Defining the Element Properties



a. Properties: Create / 2D / Shell.
b. Select *Property Set Name* and enter **al-plate**.
c. Select **Input Properties**.
d. Click **Mat Prop Name** icon, choose *aluminum* and enter **0.1** as the *Thickness*.
e. OK.

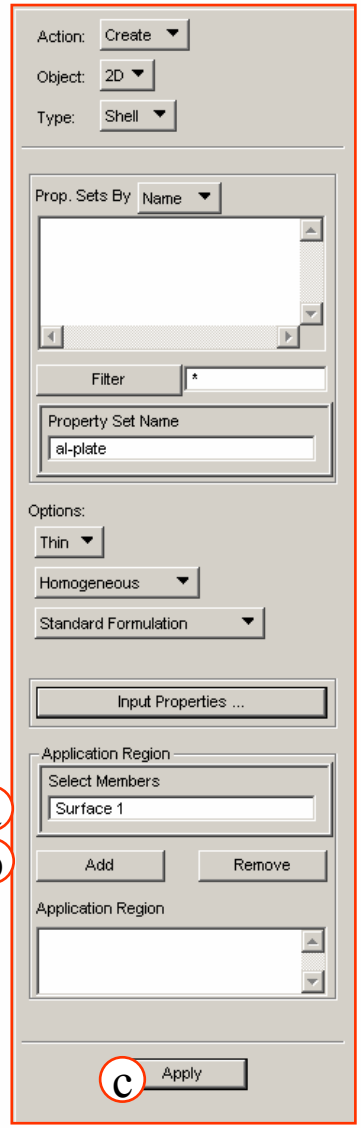


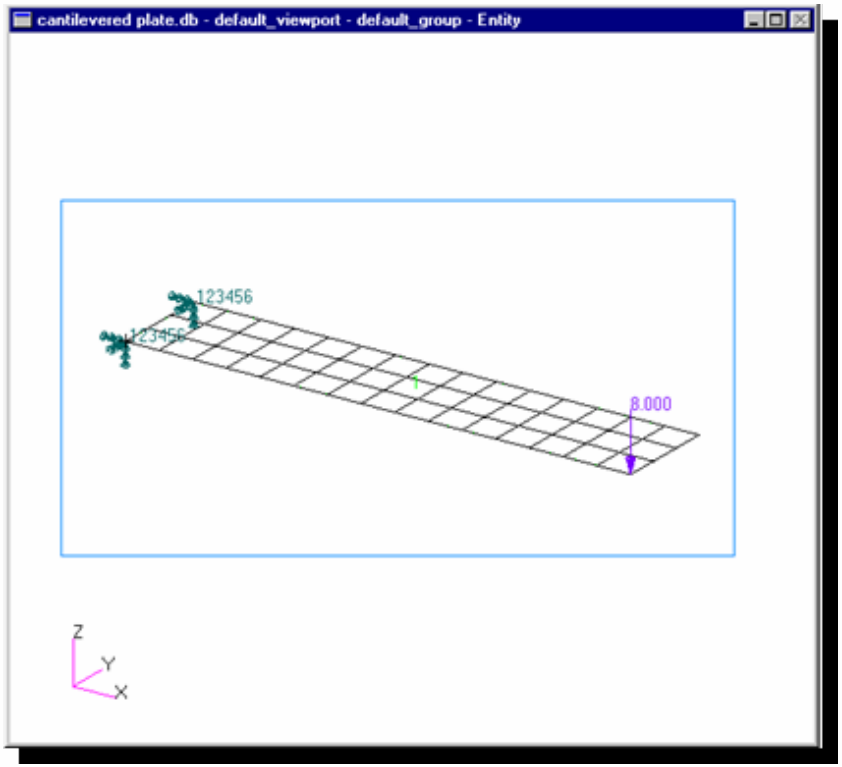




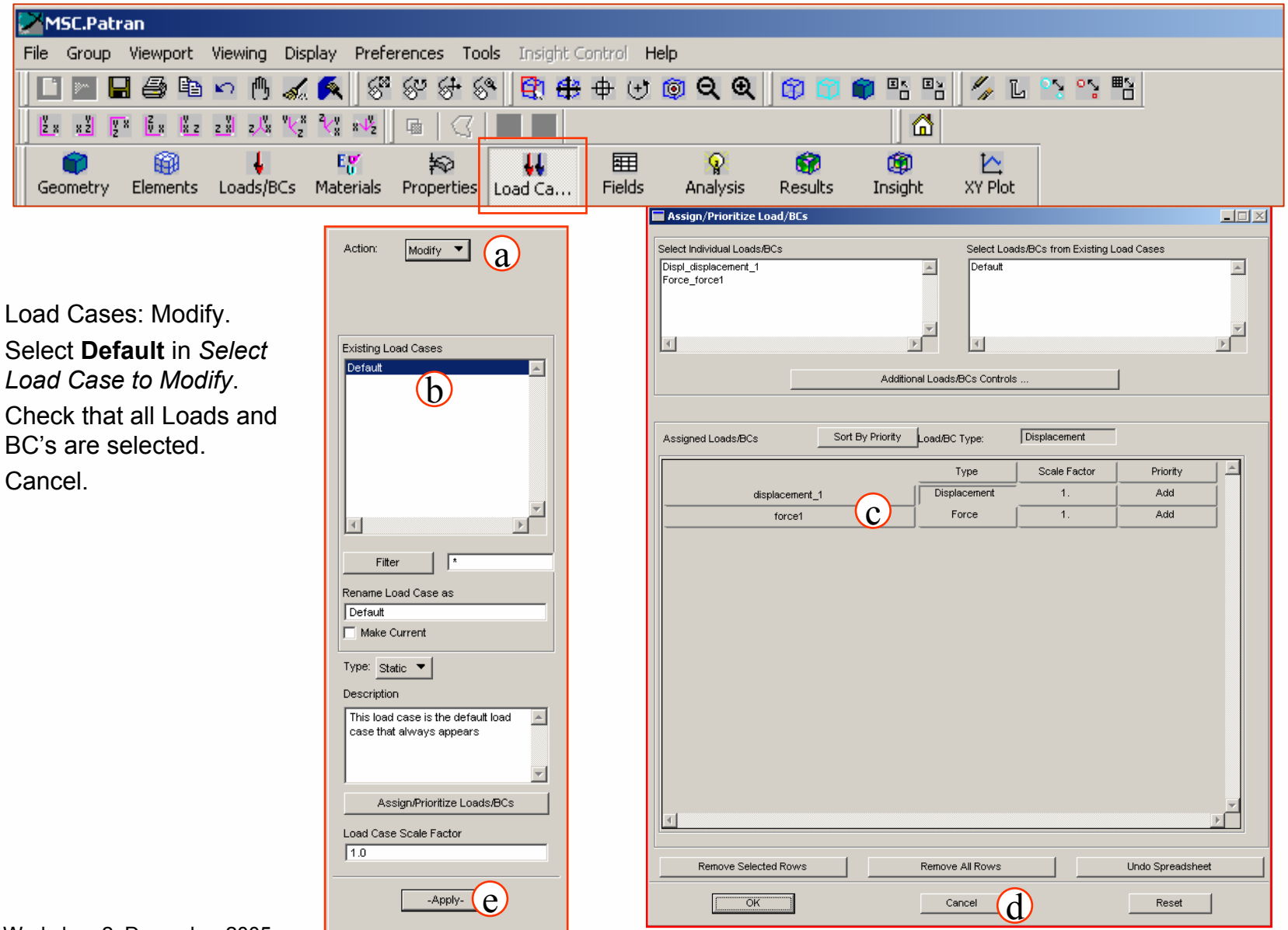
Step 7. Defining the Element Properties (Cont.)

a. Select on *Application Region* and pick to include all geometry as shown in the figure.
b. Add.
c. Apply.



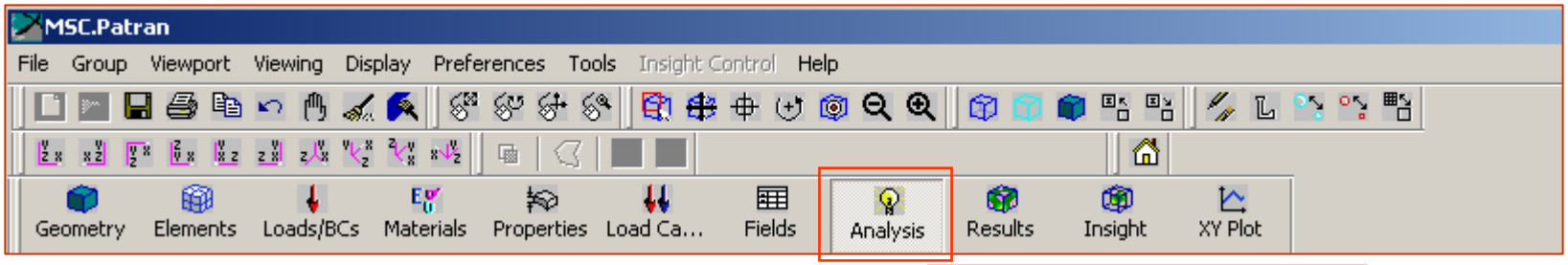


Step 8. Verify all Loads and BC's for Selection



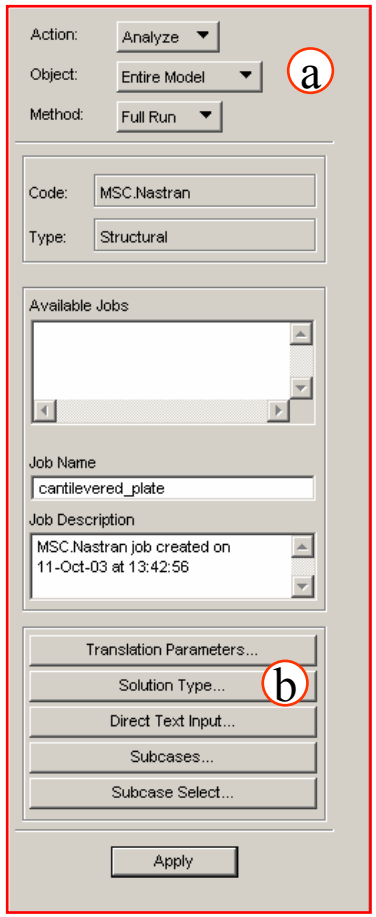
a. Load Cases: Modify.
b. Select **Default** in *Select Load Case to Modify*.
c. Check that all Loads and BC's are selected.
d. Cancel.

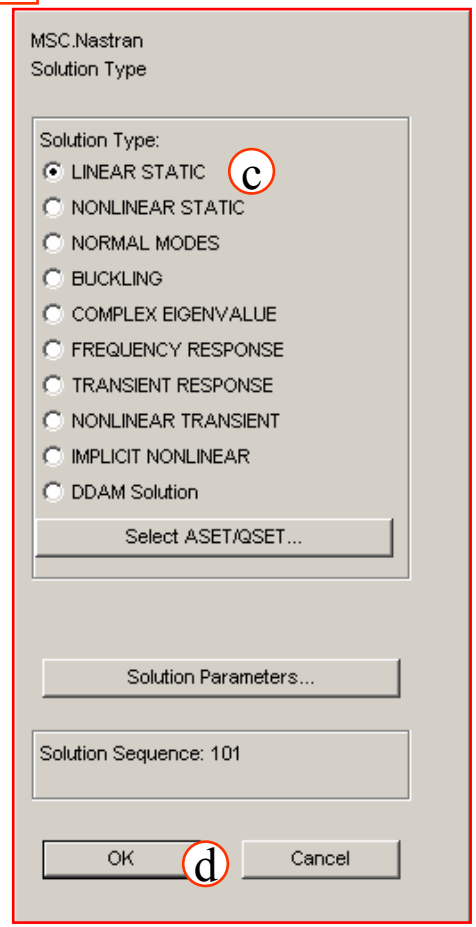
Step 9. Analysis



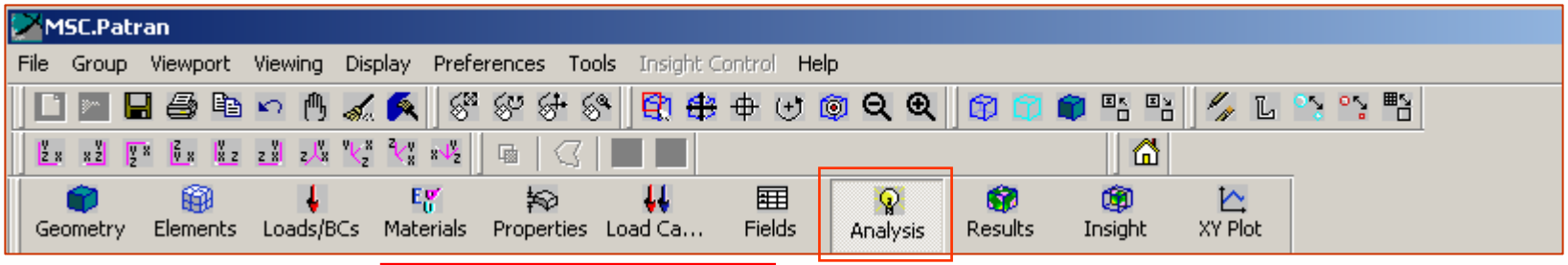
Run the analysis of the model.

a. Analysis: Analyze / Entire Model / Full Run.
b. Select **Solution Type**.
c. Choose **LINEAR STATIC** for *Solution Type*.
d. OK.
e. Apply.



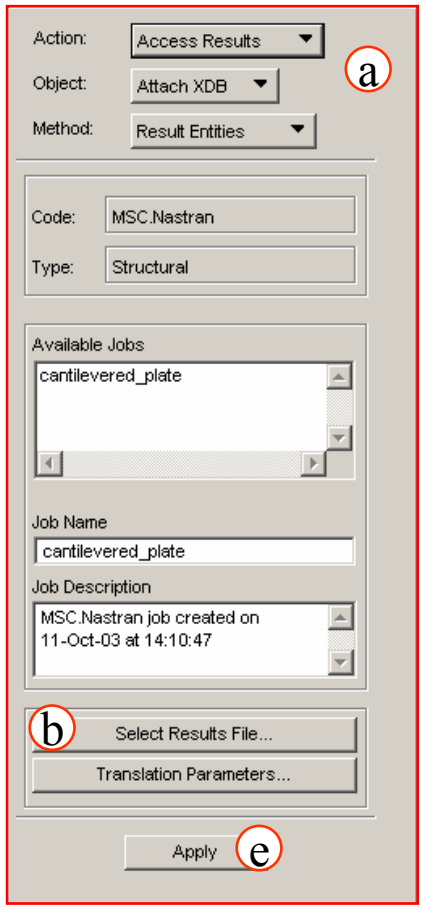


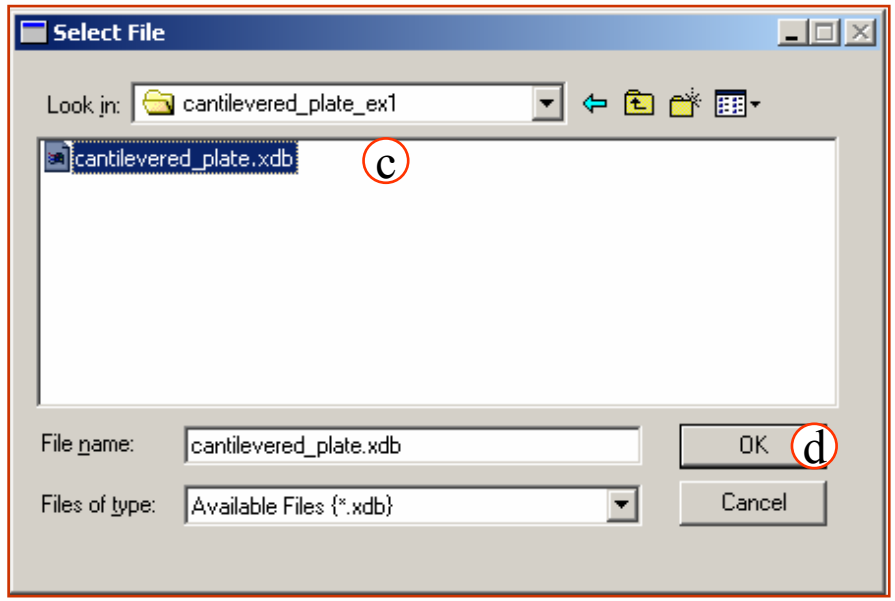
Step 10. Read Results Under Analysis



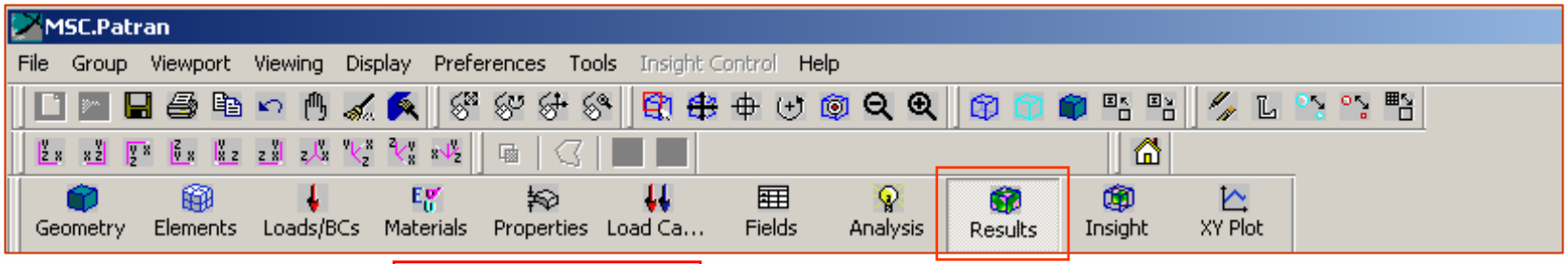
We will attach the .xdb file in order to read the results.

a. Analysis: Access Results / Attach XDB / Result Entities.
b. Click on **Select Results File**.
c. Select **cantilevered_plate.xdb**.
d. OK.
e. Apply.



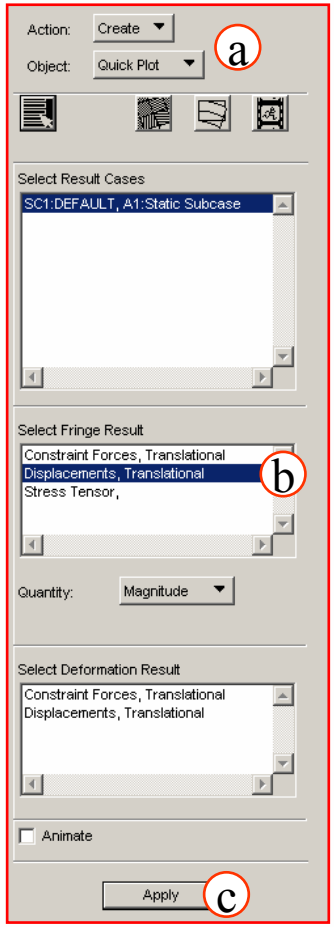


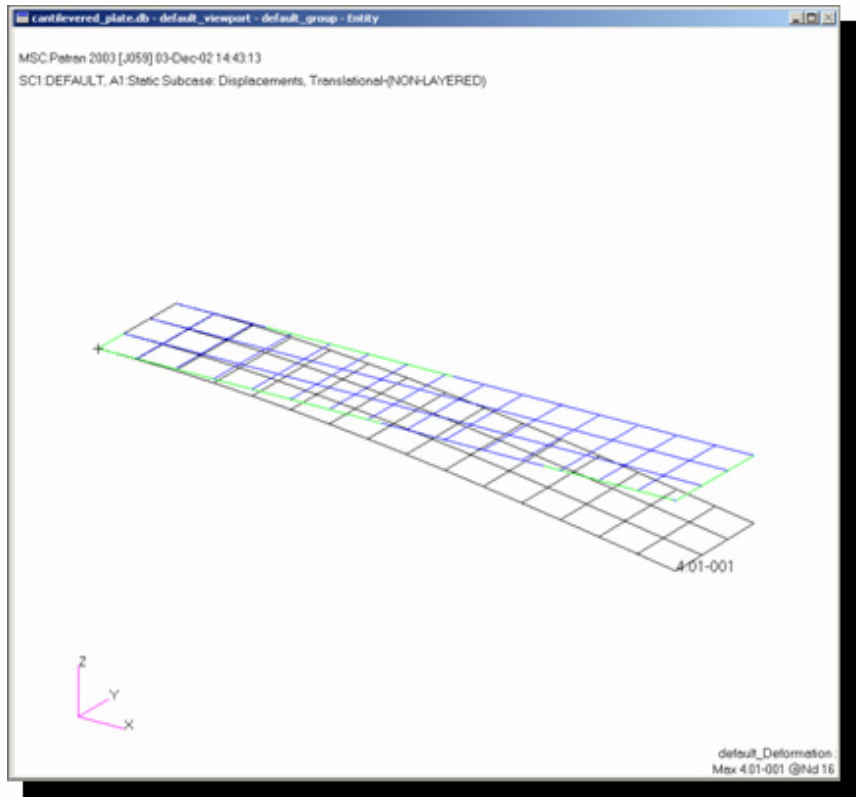
Step 11. Results



Create a Quick Plot .

a. Results: Create / Quick Plot.
b. Select **Displacement, Translational** under *Select Deformation Result*.
c. Apply.



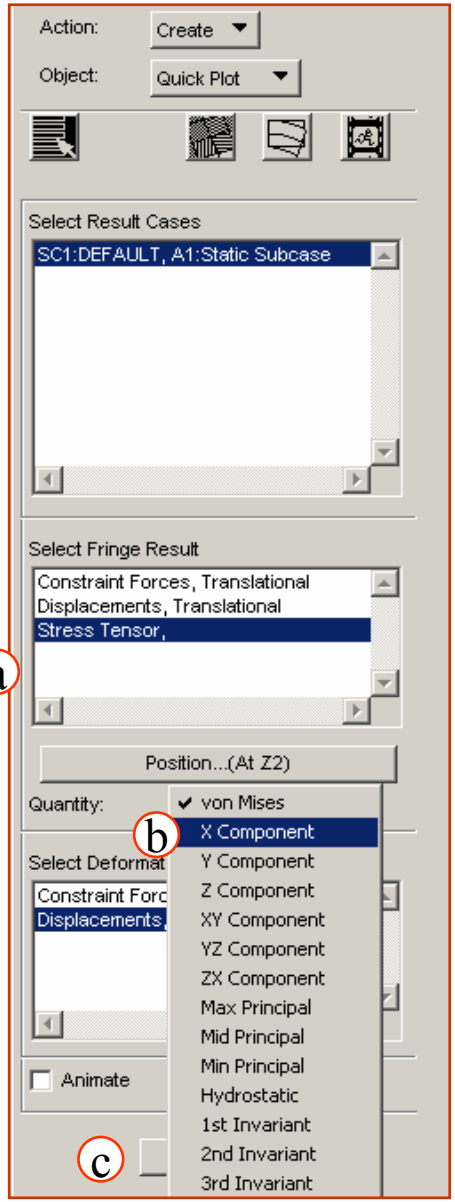


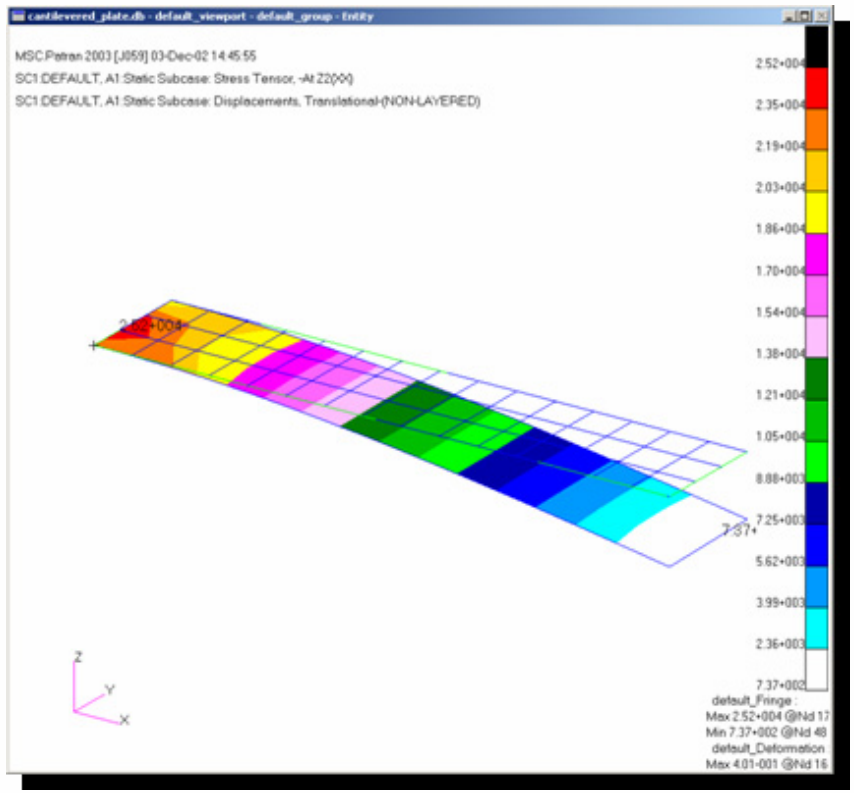
Step 11. Results (Cont.)

a. Select **Stress Tensor** under *Select Fringe Result*.
b. Choose **X Component** in *Quantity*.
c. Apply.

This ends this exercise

d. File / Close.





http://www.mscsoftware.com/Academia/Student-Edition/Default.aspx

Egregio Professore Mucchi

Da oggi può aiutare i suoi studenti a rafforzare le loro competenze nella progettazione multidisciplinare (CAE)

con la Student Edition di MSC.Software, scaricabile gratuitamente da tutti gli di studenti iscritti regolarmente

ad un corso di studi.

La Student Edition è stata pensata specificamente per gli studenti universitari e forniscono funzionalità di analisi multidisciplinare

e agli elementi finiti (FEA), simulazioni strutturali e termiche, dinamiche multicorpo, integrazione di sistemi di controllo e analisi

lineare e non-lineare avanzata.

Utilizzare la Student Edition permette agli studenti di accedere alle soluzioni standard utilizzate nell'industria da più di 40 anni con

i prodotti MD Nastran, Patran, MD Adams e Marc. Le Student Edition sono infatti versioni limitate, non nelle funzionalità ma nel

numero di elementi modellabili, dei più diffusi software MSC, specificatamente pensate per i progetti scolastici e lo studio individuale.

Scaricando e installando la Student Edition gli studenti avranno inoltre accesso ad un'ampia serie di tutorials, webinars ed esempi

con i quali potranno esercitarsi senza difficoltà.

Per scaricare la Student Edition è sufficiente registrarsi sul sito attestando la propria iscrizione ad un corso universitario.

Nel pdf allegato troverà le istruzioni dettagliate per la procedura di registrazione e il *download*.

Le aziende sono impegnate a ridurre sempre più i lenti e costosi test fisici, sostituendoli e integrandoli con simulazioni numeriche

in grado di prevedere il comportamento reale dei prodotti. Apprendere le tecniche di simulazione con la Student Edition di MSC.Software

darà agli studenti il vantaggio necessario per emergere nel mondo del lavoro, sempre più competitivo.

La invitiamo a proporre ai suoi studenti di visitare il sito web www.mscsoftware.com

e scaricare la loro Student Edition.

Cordiali saluti

Daniele Catelani

daniele.catelani@mscsoftware.com

Daniele Catelani

Project Manager

MSC.Software srl (http://www.mscsoftware.com)

Via F.d'Assisi,22

10121 Torino

Office phone: +39 0115151840

Mobile phone: +39 3489011012

Skype: daniele_catelani

daniele.catelani@mscsoftware.com

MSC Software

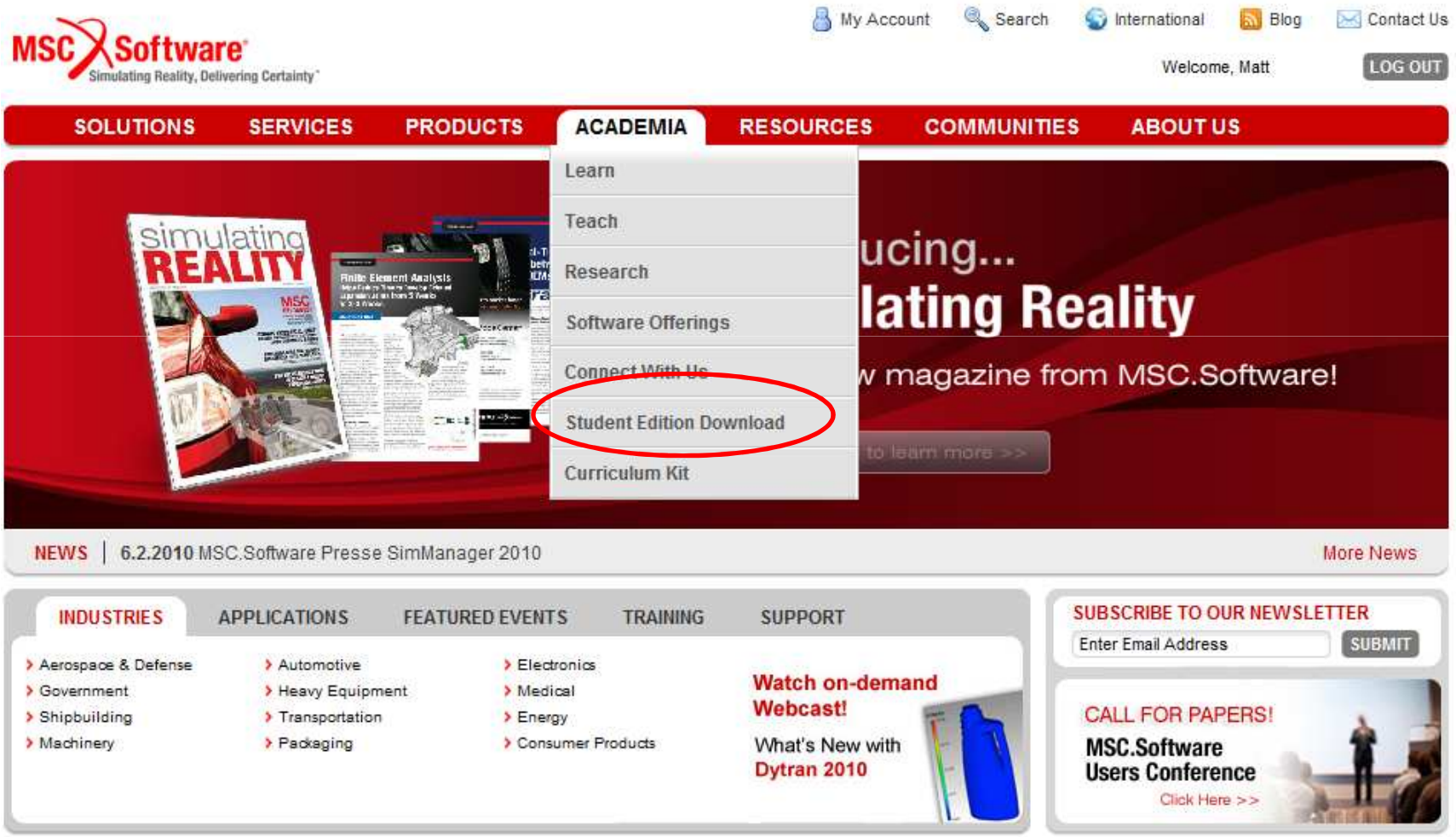
MSC Student Editions

Istruzioni per registrazione e Download

MSC Software
Student Edition

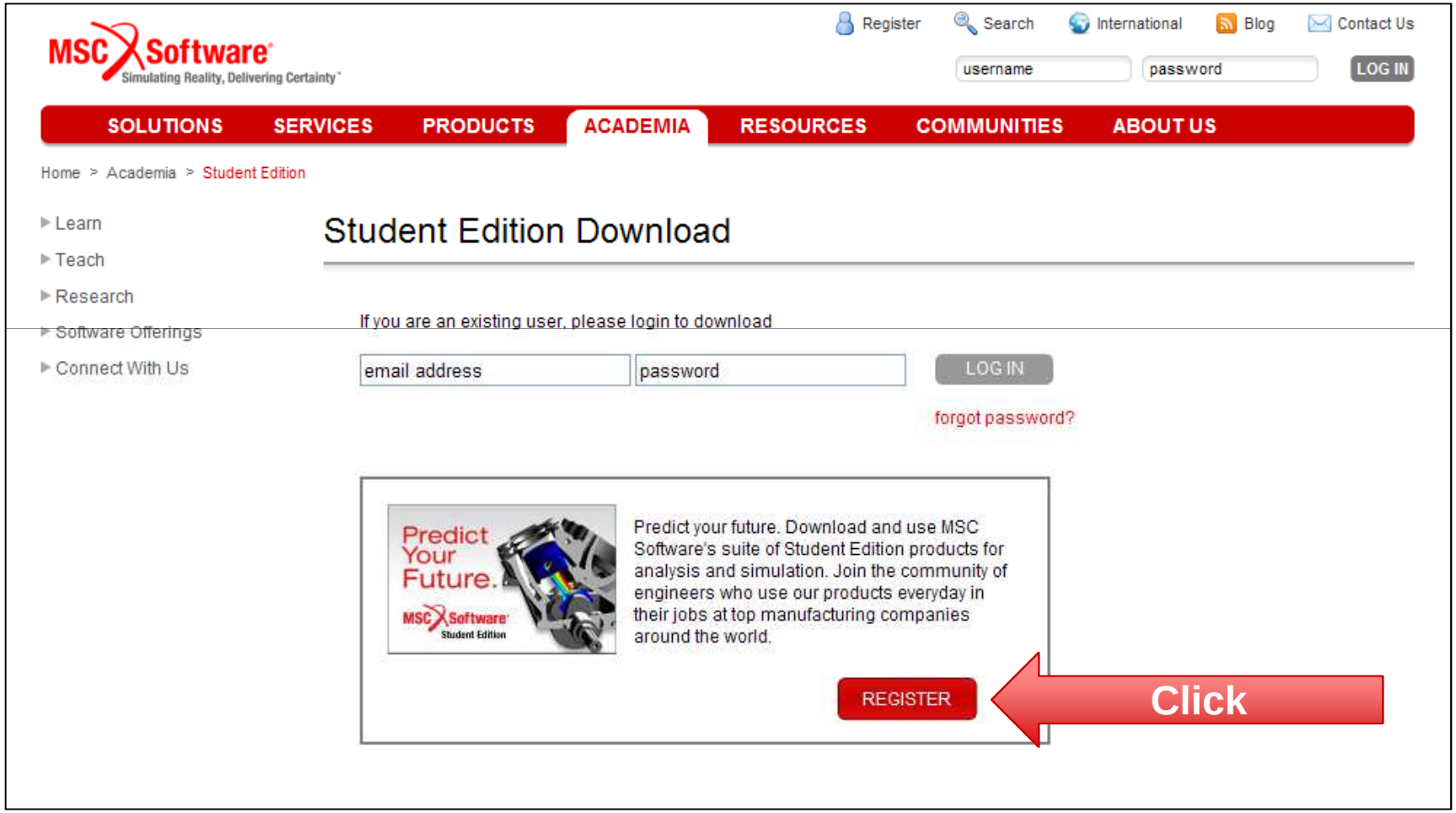
Student Edition Download Location

Dal sito www.mscsoftware.com cliccare su Academia > Student Edition Download



Student Edition Landing Page

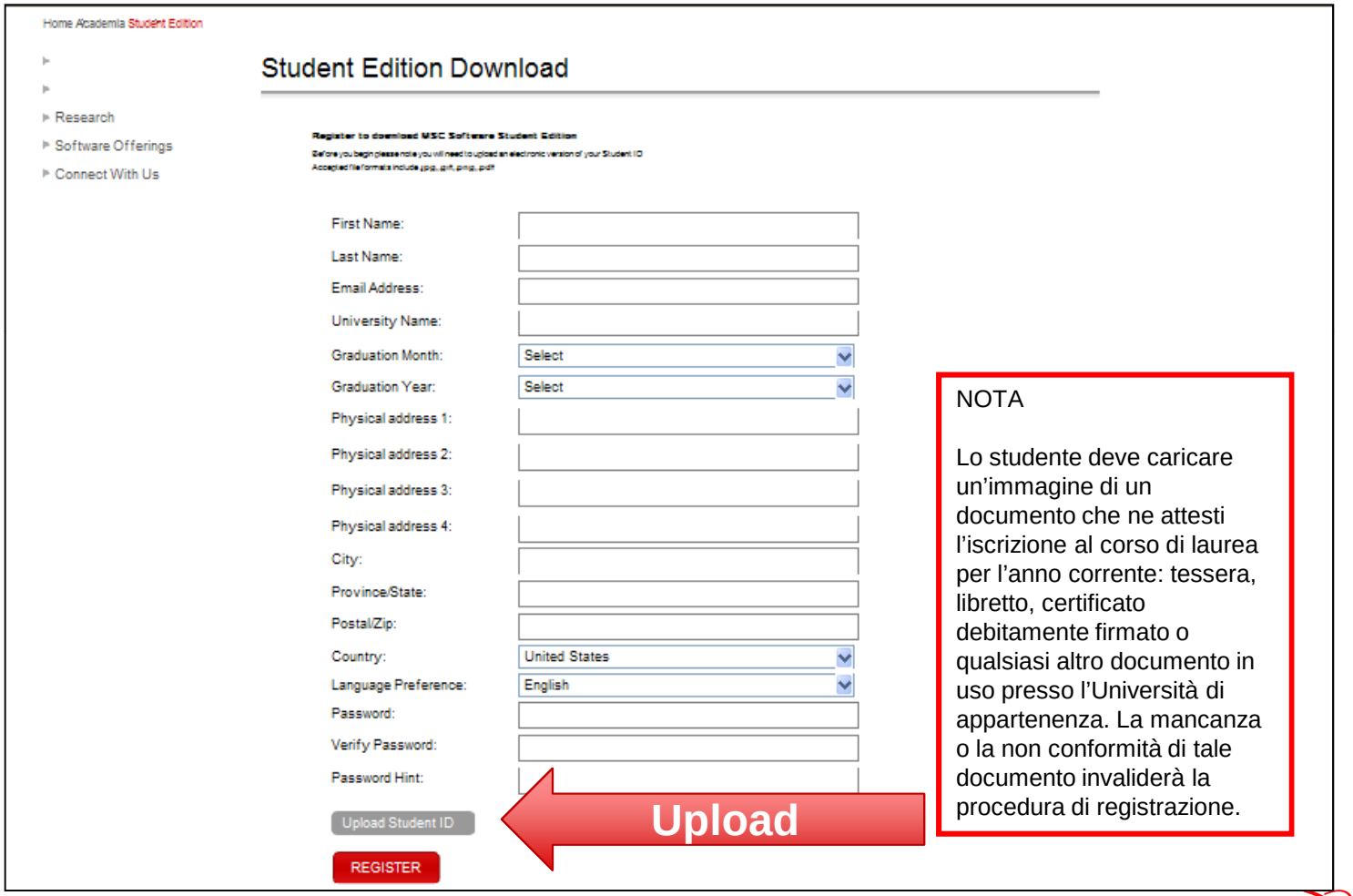
Se non si è già utenti registrati, seguire la procedura di registrazione



MSC Software
Simulating Reality, Delivering Certainty

Student Edition Registration Form

Inserire completamente i propri dati e una immagine (Student ID)



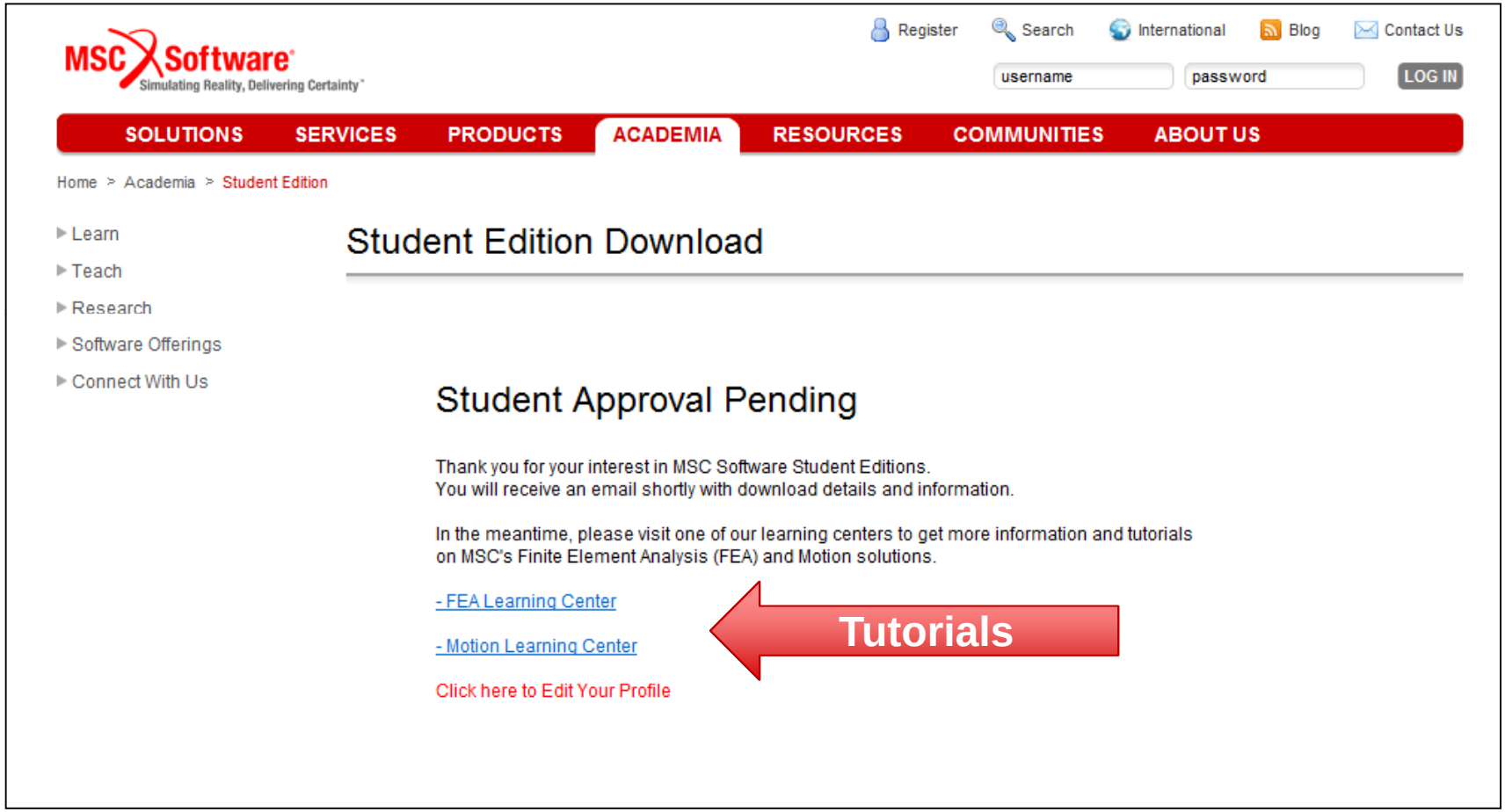
NOTA

Lo studente deve caricare un'immagine di un documento che ne attesti l'iscrizione al corso di laurea per l'anno corrente: tessera, libretto, certificato debitamente firmato o qualsiasi altro documento in uso presso l'Università di appartenenza. La mancanza o la non conformità di tale documento invaliderà la procedura di registrazione.

MSC Software
Simulating Reality, Delivering Certainty™

Pending Approval Message

La richiesta deve essere approvata



MSC Software
Simulating Reality, Delivering Certainty

Approval Email

Lo studente riceverà un'e-mail di conferma con il link per il download

MSC Software
Simulating Reality, Delivering Certainty™

Dear Matt,

Congratulations and welcome to MSC Software's Student Editions!

You're about to experience some of the best applications for engineering analysis and simulation. Download today at:
http://www.mscsoftware.com/Student_Edition.

Please also visit our website for tutorials and demos of MSC Software applications. Click on FEA and Motion & Systems Learning Centers:
http://www.mscsoftware.com/Learn

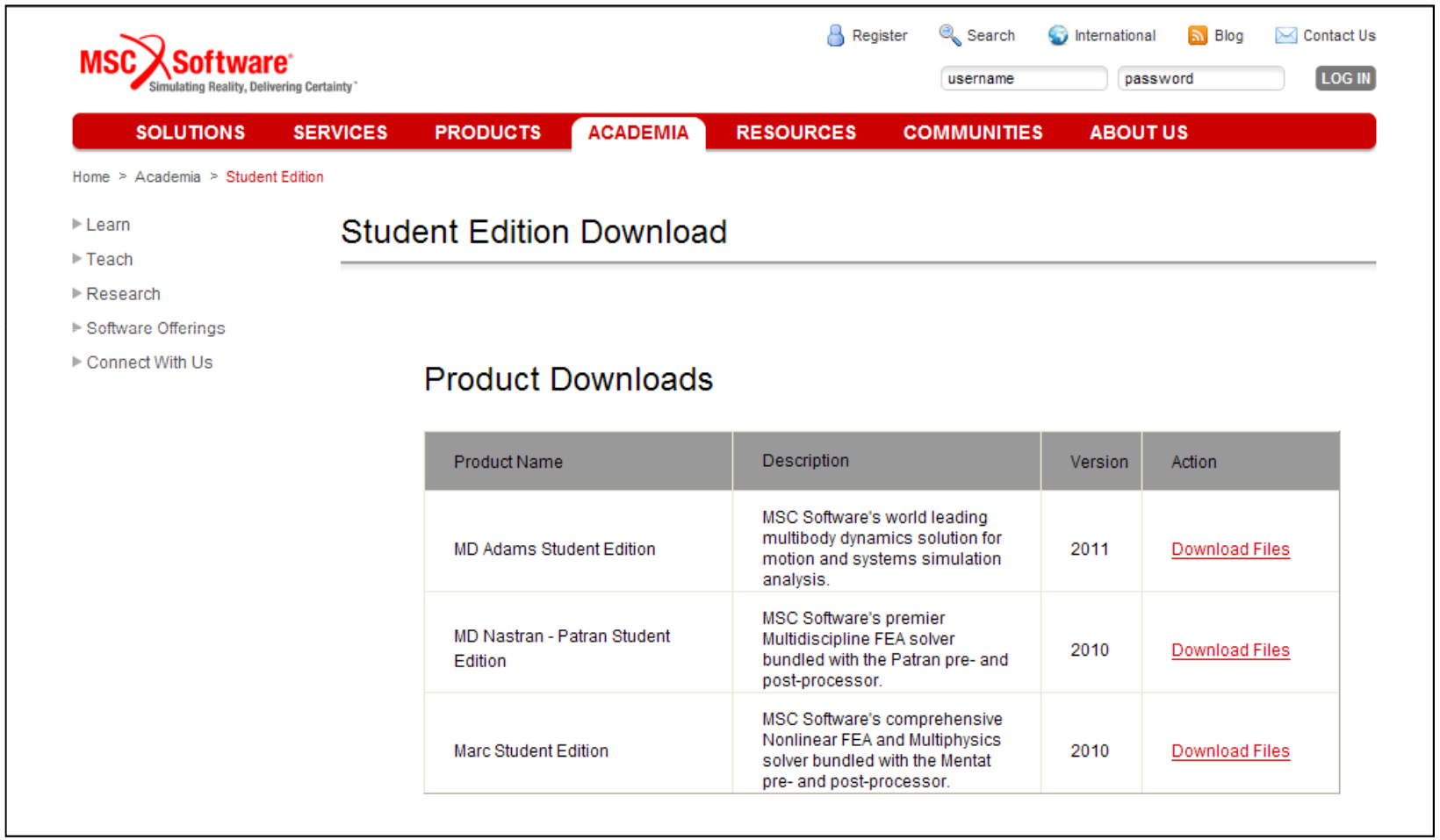
Tutorials

Thank you,
MSC Software

MSC Software
Simulating Reality, Delivering Certainty™

Software Download Page

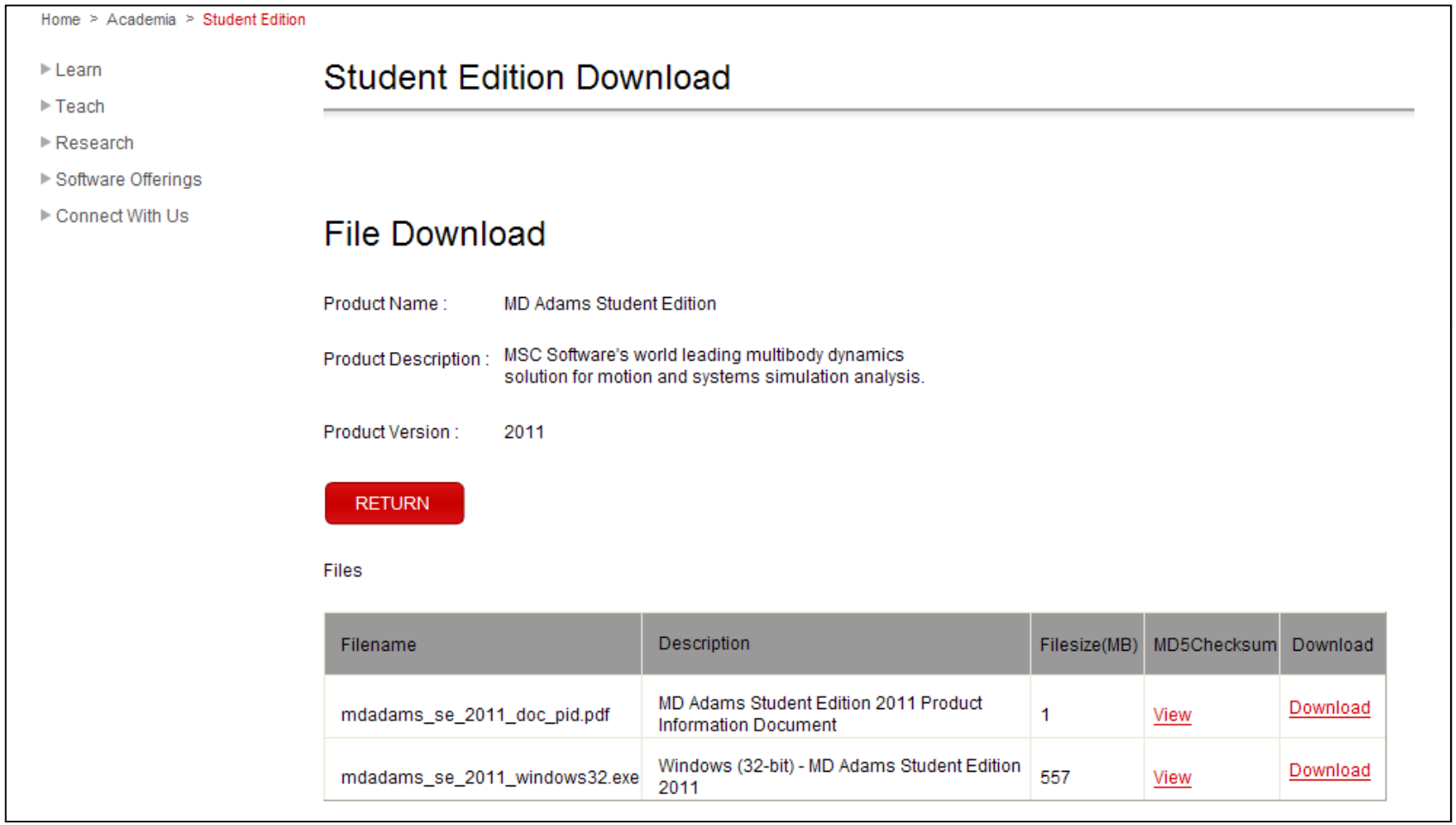
Sono disponibili 3 Student Edition (per piattaforme Windows 32 o 64 bit)



Software Download Page

Ogni pacchetto contiene il software con licenza inclusa

Sono disponibili anche Product Information Document



MSC Software
Simulating Reality, Delivering Certainty