MSC/NASTRAN

Análise Estática de Estruturas

Eliseu Lucena Neto

2014

Introdução

Esperamos que estas notas sejam úteis num primeiro contato do aluno com o programa de elementos finitos *MSC/NASTRAN*, utilizando a versão 7.01 do FEMAP. A apresentação do programa será feito por meio de exemplos envolvendo a análise estática de estruturas.

A MacNeal-Schwendler Corporation, fundada em 1963, pesquisa, desenvolve e dá suporte a softwares CAE (Computer Aided Engineering) ligados à modelagem e análise por elementos finitos. Participou, junto à NASA (National Aeronautics and Space Administration), no desenvolvimento do programa *NASTRAN* (NAsa STRuctural ANalysis), tornando-se cedo proprietária da versão *MSC/NASTRAN*. A primeira versão comercial do *MSC/NASTRAN* é de 1971.

Dentre as áreas de aplicação do *MSC/NASTRAN*, é a análise estrutural o seu lugar-comum, seguida de aplicações em transferência de calor. Além da evolução natural que vem sofrendo ao longo dos anos, hoje se acha disponível para computadores que variam desde os micros até os supercomputadores.

Em linhas gerais, o *MSC/NASTRAN* realiza:

- *Análise Estática Linear*: é o tipo de análise mais básica. O termo "linear" significa que a resposta da estrutura – os deslocamentos e as tensões, por exemplo – é linearmente relacionada com as cargas aplicadas. O termo "estática" significa que as cargas aplicadas não variam com o tempo – ou que a variação no tempo é insignificante, podendo ser seguramente ignorada.

- *Análise Estática Não-Linear*

- *Flambagem*: faz uso do "problema linearizado no deslocamento" para determinação da carga crítica (problema de autovalor).

- *Análise Modal*: calcula as frequências naturais e os correspondentes modos de vibração de uma estrutura (problema de autovalor).

- *Análise Harmônica*: determina a resposta de uma estrutura quando sujeita a carregamentos que variam harmonicamente com o tempo (carregamentos com frequência definida).

- *Análise Dinâmica Transiente*: Determina a resposta de uma estrutura quando sujeita a carregamentos que variam arbitrariamente com o tempo. Todas as cargas aplicadas são conhecidas em qualquer instante.

- *Análise Dinâmica Não-Linear*

- *Transferência de Calor em Regime Estacionário*

- *Transferência de Calor em Regime Transiente*

- *Otimização*

As variáveis nodais dos elementos utilizados na análise estrutural são "deslocamentos". Quantidades como deformação e tensão são derivadas posteriormente.

Elementos

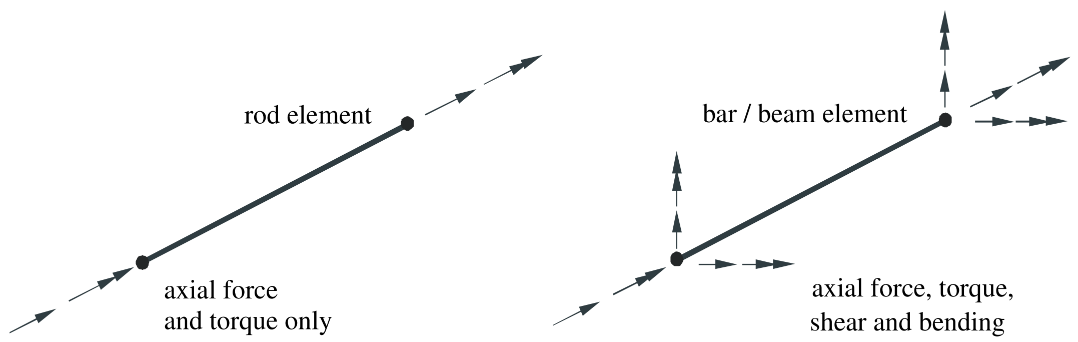
As formas geométricas dos elementos comumente utilizados no *MSC/NASTRAN* para a análise estrutural são:

- Elementos unidimensionais: usados em treliças e pórticos.

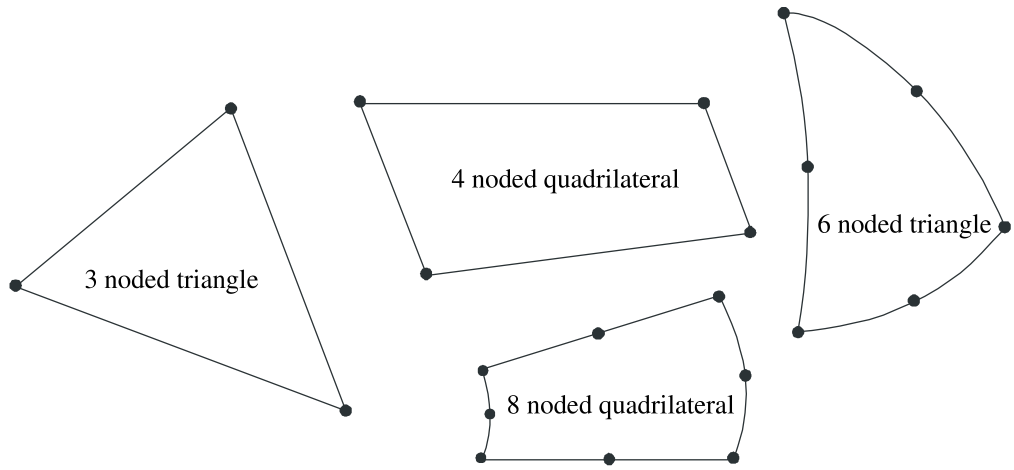
  ROD: resiste a esforço normal e torção; graus de liberdade de um nó no sistema local: TX (translação na direção de X), RX (rotação em torno de X).

  BAR: resiste a todos os esforços; graus de liberdade de um nó no sistema local: TX, TY, TZ, RX, RY, RZ; prismático.

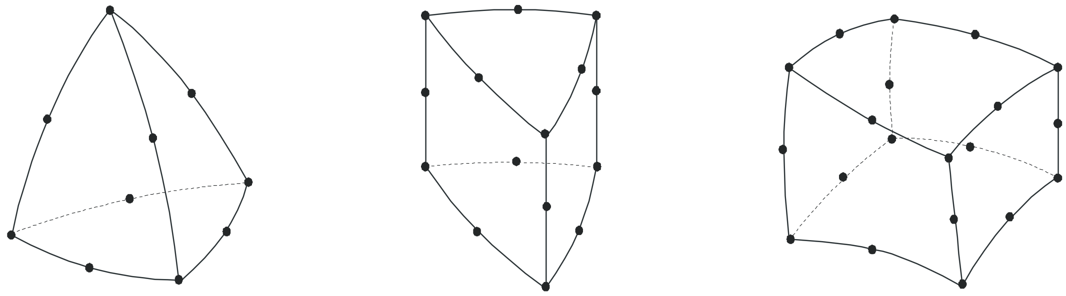
  BEAM: resiste a todos os esforços; graus de liberdade de um nó no sistema local: TX, TY, TZ, RX, RY, RZ; seção transversal variável; o eixo neutro e o de cisalhamento não precisam coincidir; pode levar em conta o empenamento da seção transversal na rigidez à torção; etc.



- Elementos bidimensionais: são triângulos ou quadriláteros planos ou curvos; usados em membranas, placas e cascas; graus de liberdade de um nó no sistema local: TX, TY, TZ, RX, RY.



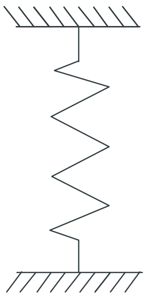
- Elementos tridimensionais: são tetraedros, pentaedros e hexaedros; usados em sólidos; graus de liberdade de um nó no sistema local: TX, TY, TZ.



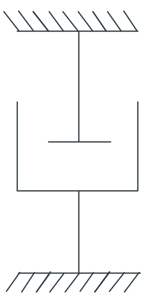
4 or 10 noded TETRA   6 or 15 noded PENTA   8 or 20 noded HEXA

(with and without mid-side nodes)

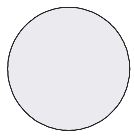
- Elementos especiais: molas, amortecedores, massas concentradas, etc.



spring



damper



concentrated mass

Aplicações Numéricas

O programa *MSC/NASTRAN* empregado recorre à versão 7.01 do FEMAP como processador dos dados de entrada e saída dos resultados. Dentre os arquivos criados e deixados em disco, destacamos:

- *xxx.DAT* dados que podem ser executados a qualquer momento.
- *xxx.F06* saída de resultados em ASCII.
- *xxx.OP2* saída de resultados em binário.

- *xxx.MOD* contém a parte gráfica de *xxx.DAT*. Quando se faz uma execução, pode-se adicionar em *xxx.MOD* a saída de resultados (parte gráfica de *xxx.F06* ou *xxx.OP2*).

São apresentados dez exemplos denominados Workshop 1, 2, $\cdots$, 10, sendo o primeiro deles escrito mais detalhadamente. Alguns foram adaptados da página

> *http://www.mscsoftware.com ⇒ mechanical solutions ⇒ support ⇒ application examples ⇒ example exercises ⇒ msc.nastran for windows*

e outros foram aqui desenvolvidos.

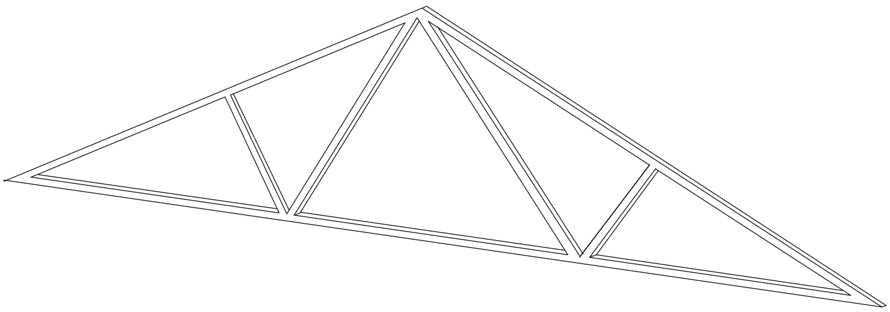
Recomendamos a reprodução de todos os dez exemplos no *MSC/NASTRAN*, experimentando de próprio punho a potencialidade de um programa dessa natureza. Perceba como é possível automatizar a análise estrutural e reservar ao engenheiro unica e exclusivamente a parte interpretativa dos resultados. Sobrará assim mais tempo para dedicação à parte criativa do projeto.

> "Who, in practice nowadays, would conduct an elastic analysis of a single-bay portal frame other than by feeding it into the office program? ...university libraries contain shelves of structural textbooks devoted to complex and impenetrable hand-methods for analysing such structures." (D. A. Nethercot, "On the Teaching of Structural Engineering", *Proceedings of the Conference on Civil and Structural Engineering in the 21st Century*, University of Southampton, 26–28 April 2000, p. 157).

However, beware of computers. And, especially beware of developers of engineering software. Regardless of the source of trouble, the engineer who uses the software is held responsible for the results.

Workshop 1

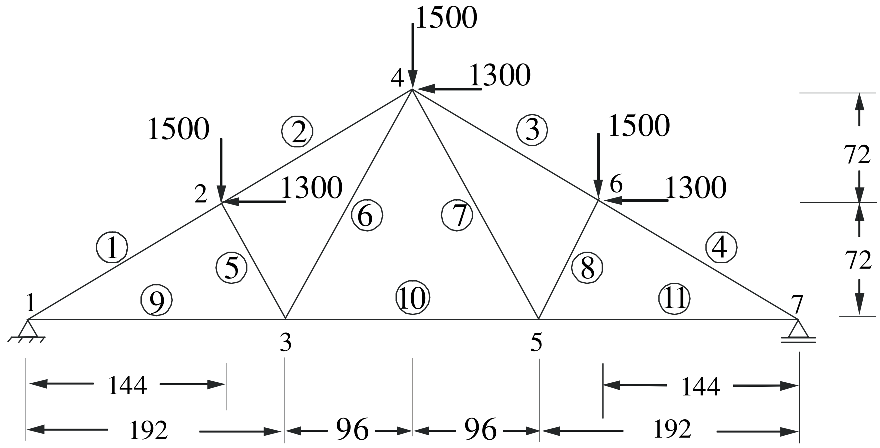
Linear Static Analysis of a Simply-Supported Truss



Objectives

- Create a finite element model by explicitly defining node locations and element connectivities.
- Define a MSC/NASTRAN analysis model comprised of rod elements.
- Run a MSC/NASTRAN linear static analysis.
- View analysis results.

Model Description



Above is a finite element representation of the truss structure shown on the title page. The nodal coordinates provided are defined in the global cartesian coordinate system (MSC/NASTRAN Basic System). The structure is comprised of truss segments connected by smooth pins such that each segment is either in tension or compression. The structure is pinned at node 1 and supported by a roller at node 7. Point forces are applied at nodes 2, 4 and 6.

Young's Modulus	1.76×10^6 psi
Poisson's Ratio	0.3
Cross-Sectional Area	5.25 in^2

Suggested Exercise Steps

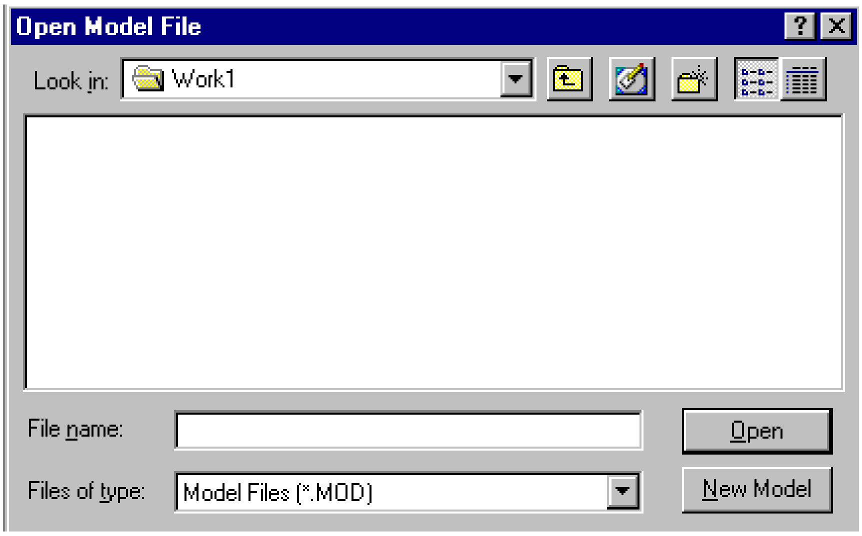
- Define a material.
- Define a rod property using the newly defined material.
- Create the nodes for the truss model in the global cartesian coordinate system.
- Create the truss segments using the newly defined property.
- Define the relevant constraints for the model.
- Create the constraint at node 1 by fixing the 1 and 2 directions (corresponding to TX and TY).
- Create the constraints at node 7 by fixing the TY direction.
- Apply a -1300 lbf in the FX direction and a -1500 lbf in the FY direction at nodes 2, 4 and 6.
- The model is now ready for analysis.
- List the results of the analysis and compare with expected answers at the end of the exercise.
- Display the deformation of the truss and remove all labels and markers.

Exercise Procedure

1. Start up **MSC/NASTRAN for Windows 4.5** and begin to create a new model.

   Double click on the icon for the **MSC/NASTRAN for Windows V4.5**.

   On the *Open Model File* form, select **New Model**.

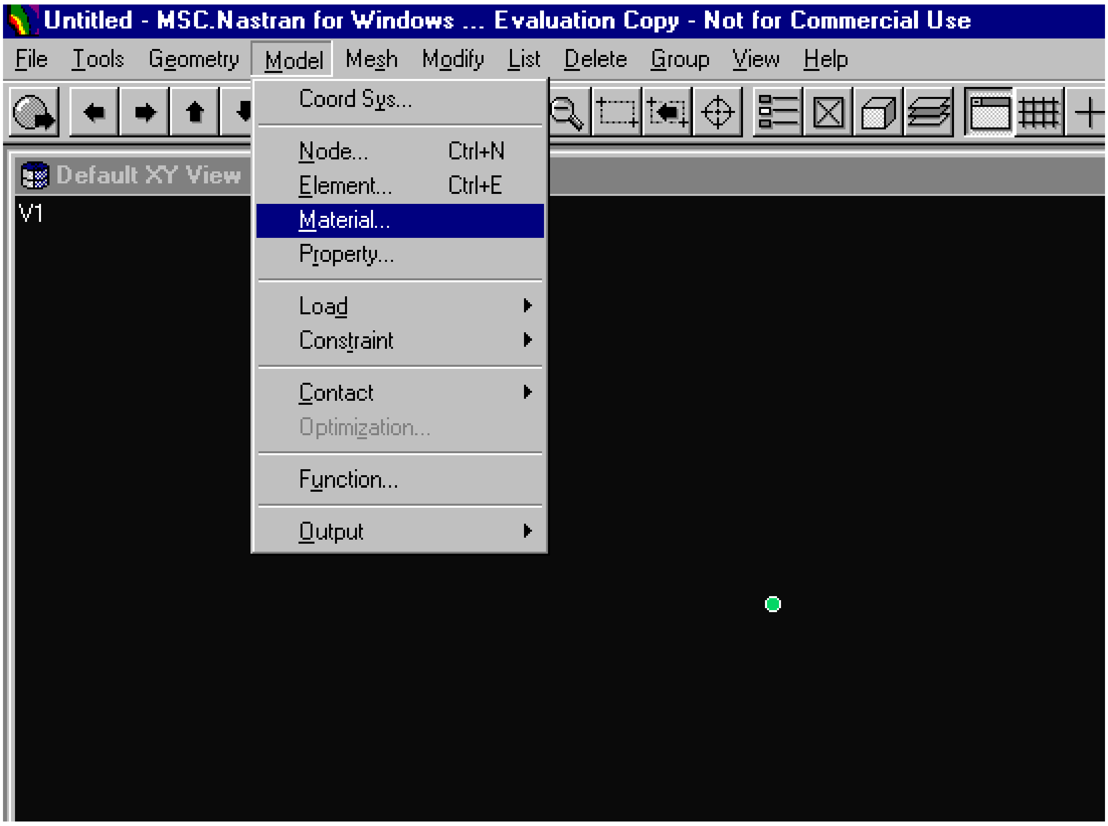
   Turn off the workplane:

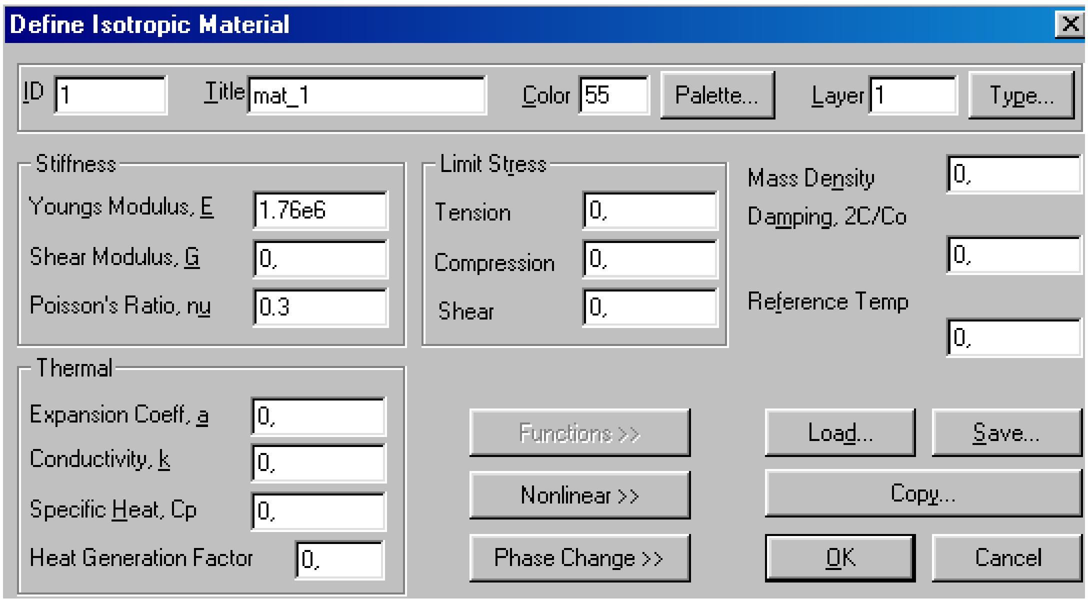
   Tools / Workplane (or **F2**) / □ **Draw Workplane / Done**

   View / Regenerate (or **Ctrl G**).

2. Create a material called **mat_1**.

   From the pulldown menu, select **Model / Material**.





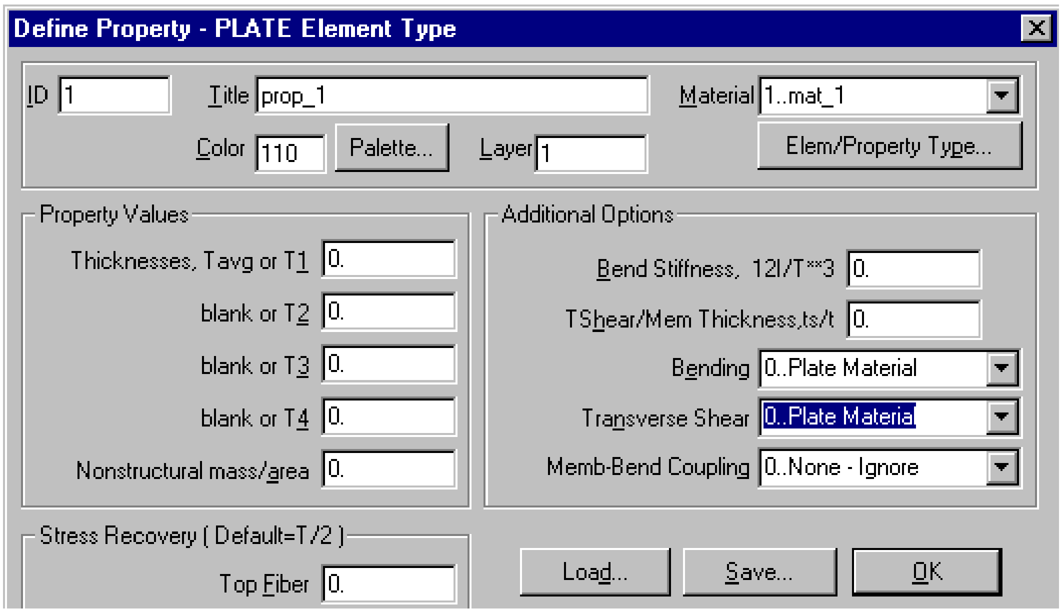
Title	**mat_1**
Young's Modulus	**1.76e6**
Poisson's Ratio	**0.3**

Select **OK** / **Cancel**.

NOTE: In the *Messages Window* at the bottom of the screen, you should see a verification that the material was created. You can check here throughout the exercise to both verify the completion of operations and to find an explanation for errors which might occur.

3. Create a property called **prop_1** to apply to the members of the truss.

From the pulldown menu, select **Model / Property**.

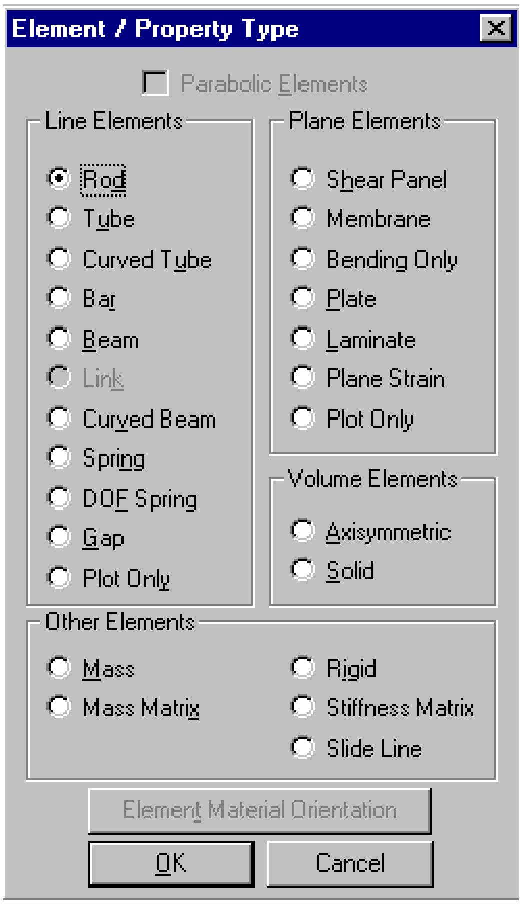


Title	**prop_1**

To select the material, click on the list icon next to the databox and select **mat_1**.

Material	**mat_1**

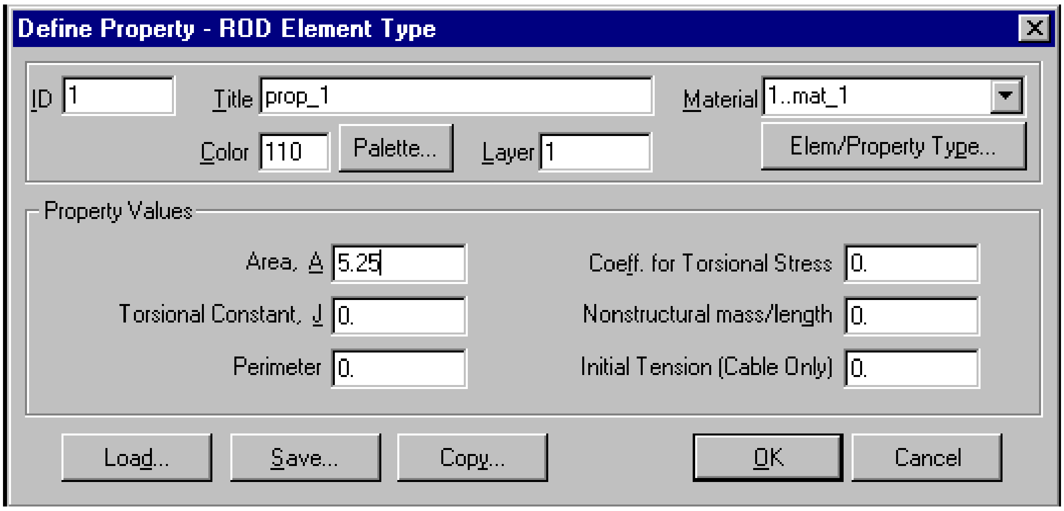
Elem / Property Type



Change the property type from **Plate** element (default) to **Rod** element.

Line Elements **Rod**

Select **OK**.

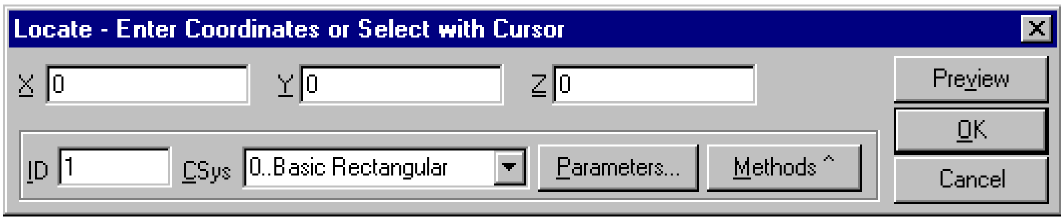


Area	**5.25**

Select **OK** / **Cancel**.

4. Create the nodes for the truss model.

   Create the first node of the model by selecting **Model** / **Node** (or **Ctrl N**).



X:	*Y*:	*Z*:	
0	**0**	**0**	select **OK**

Repeat the process for the other 6 nodes:

Node	X	Y	Z	Select
2	**144**	**72**	**0**	**OK**
3	**192**	**0**	**0**	**OK**
4	**288**	**144**	**0**	**OK**
5	**384**	**0**	**0**	**OK**
6	**432**	**72**	**0**	**OK**
7	**576**	**0**	**0**	**OK**

Select **Cancel**.

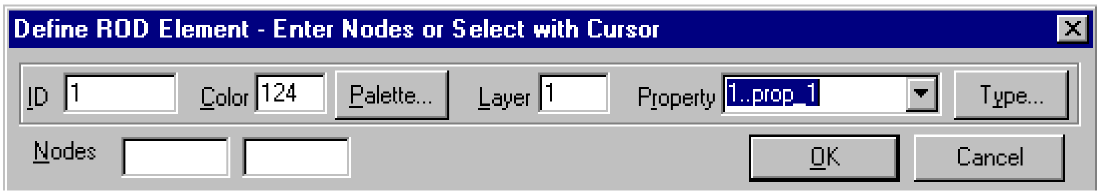
To fit the display onto the screen, select **View** / **Autoscale** / **Visible** (or **Ctrl A**)

5. Create the elements for the truss model.

   First, display the node numbers:

   View / **Options** / **Quick Options** (or **Ctrl Q**) / **Labels On** / **Done** / **OK**.

Choose **Model / Element** (or **Ctrl E**)

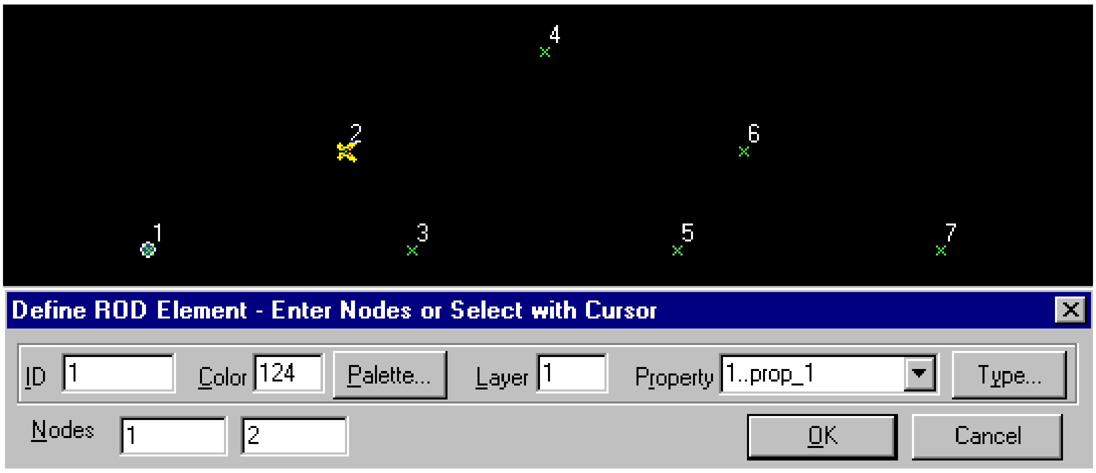


To select the property, click on the list icon next to the databox and select **prop_1**.

Property **prop_1**

When selecting the nodes, you may (if you wish) manually type in the endpoint nodes of the rod elements. However, it is easier to use the graphic interface and select the nodes on the screen using the mouse. Click in the first *Nodes* box and then select the nodes on the screen in the following order.

NOTE: The node nearest to the cursor is highlighted by a large yellow X - you don't have to click precisely on the node!

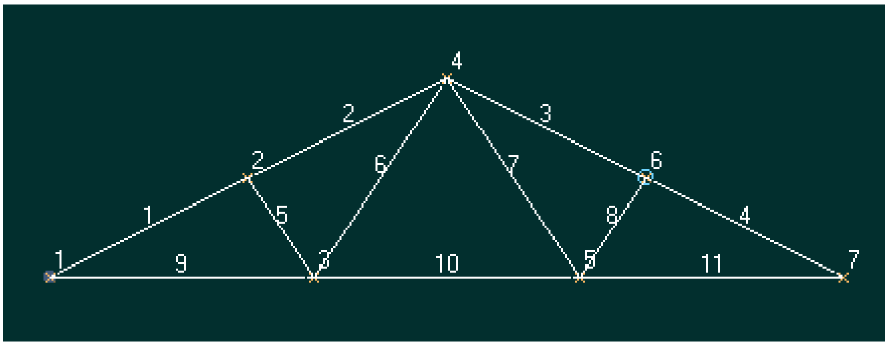


Nodes: **1** **2** select **OK**

Element 1 has now been created between the two nodes. Continue creating the rest of the elements by connecting the following nodes:

Nodes		Select
2	**4**	**OK**
4	**6**	**OK**
6	**7**	**OK**
2	**3**	**OK**
3	**4**	**OK**
4	**5**	**OK**
5	**6**	**OK**
1	**3**	**OK**
3	**5**	**OK**
5	**7**	**OK**

Select **Cancel**.



6. Create the model constraints.

Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following:

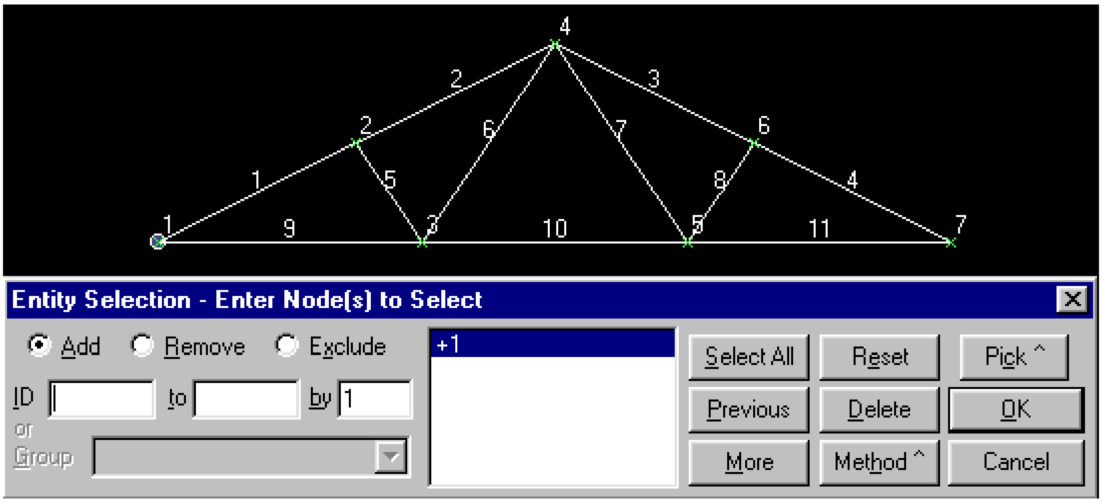
Model / Constraint / Set

Title	**constraint_1**

Select **OK**.

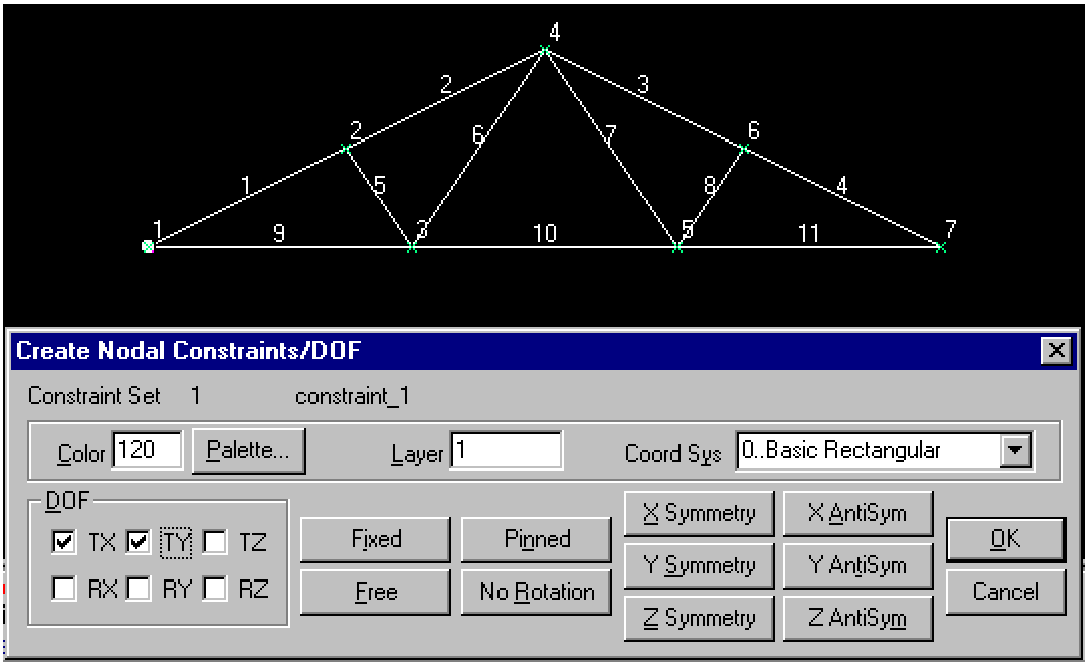
Now, define the relevant constraint for the model.

Model / Constraint / Nodal



Select **Node 1**. It will be marked with a white circle, a +1 will be added to the *Entity Selection* box, and you will be unable to highligh it anymore. These are all ways of checking which node you have selected.

Select **OK**.



On the *DOF* box, select

$\boxed{\checkmark}$ **TX** $\boxed{\checkmark}$ **TY**

Select **OK.**

Notice that the constraint appears on the screen at Node 1, fixing the 1 and 2 directions (corresponding to TX and TY). Create the constraint for the other side of the model.

Select **Node 7 / OK**

On the *DOF* box, select

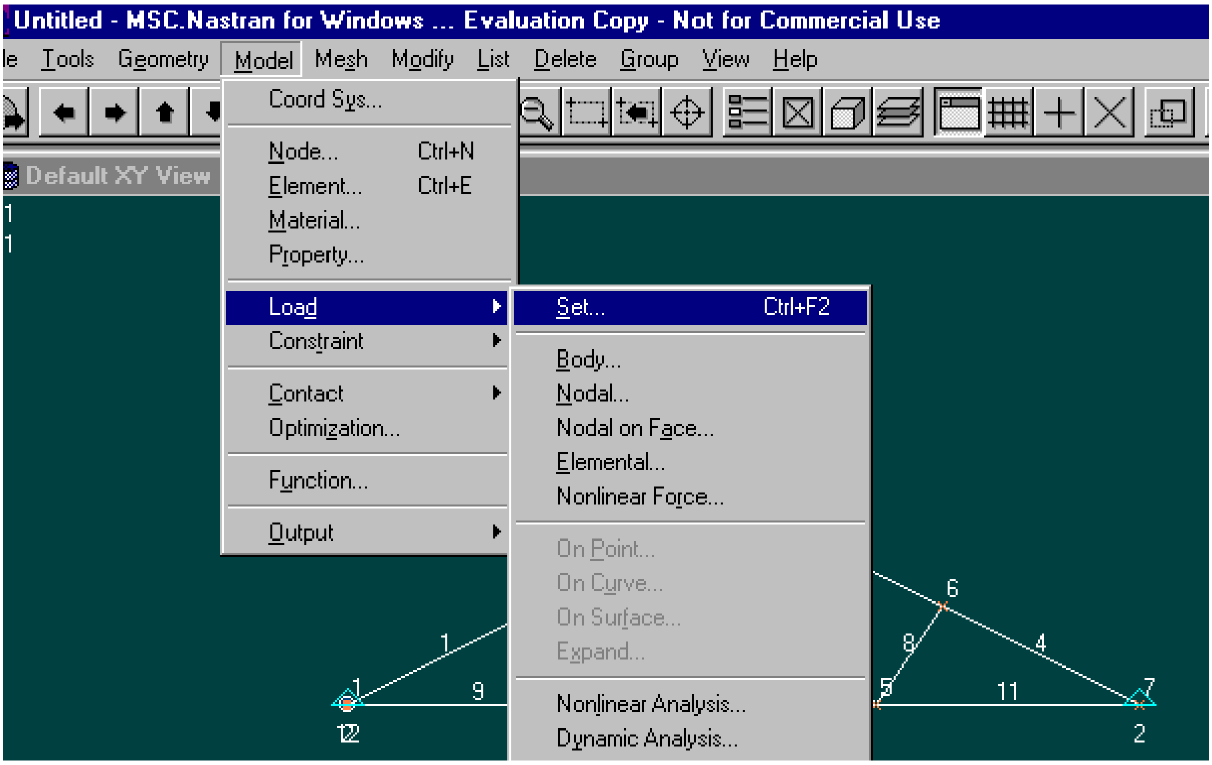
$\boxed{\checkmark}$ **TY**

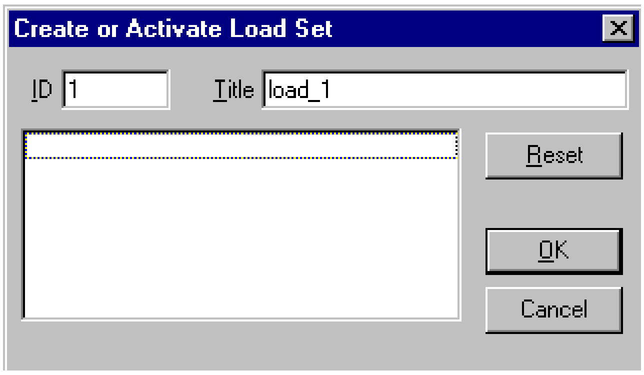
Select **OK / Cancel**.

7. Create the model loading.

Like the constraints, a load set must first be created before creating the appropriate model loading.

Model / Load / Set (or **Ctrl F2**)



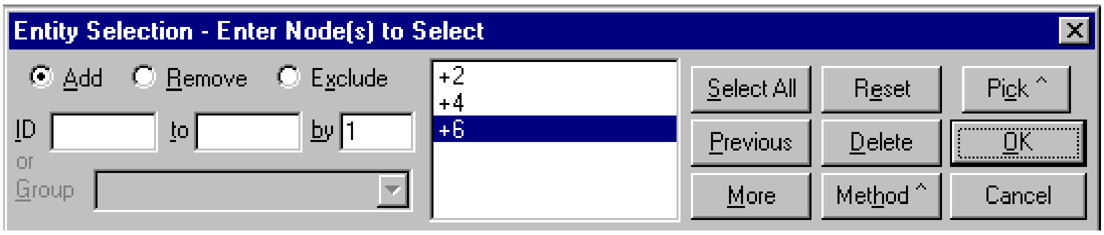


Title	load_1

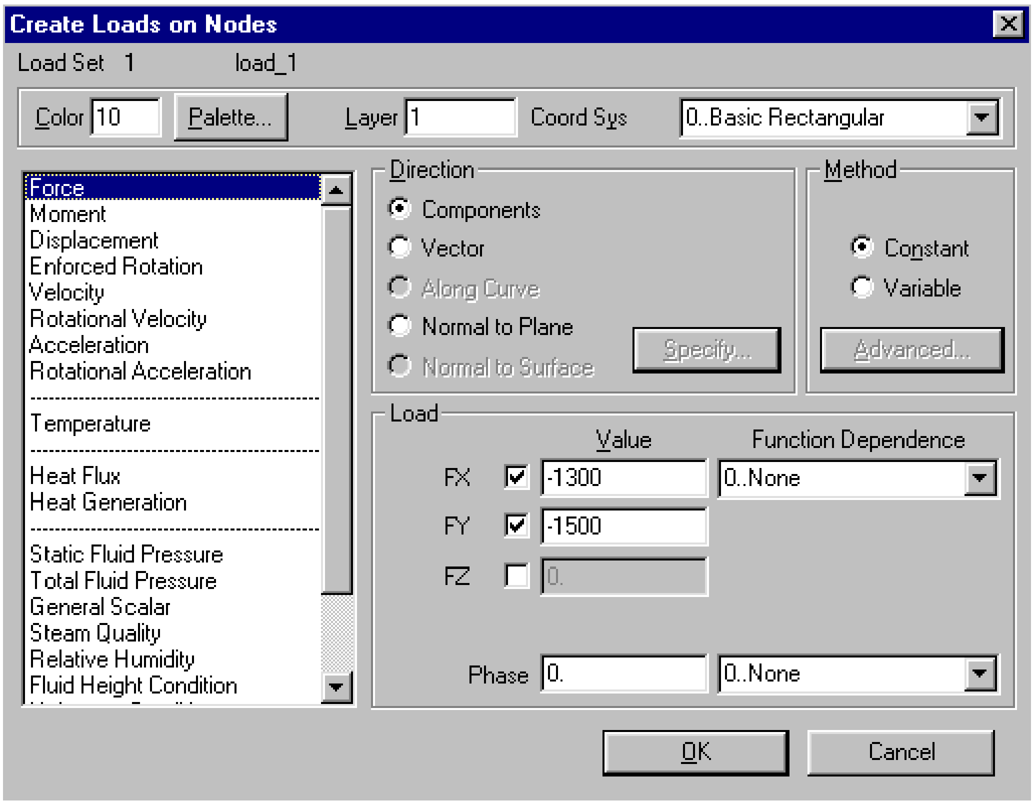
Select **OK**.

Now, define the relevant loading conditions.

Model / Load / Nodal



Select **Nodes 2**, **4** and **6 / OK**



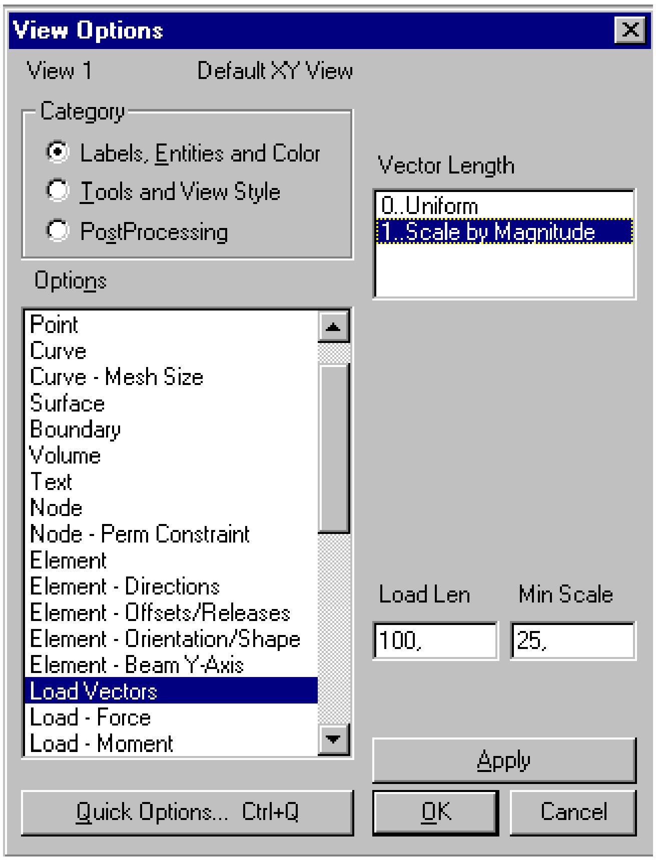
Highlight **Force**

Method	**Constant**	
Load	**FX** ✓	**-1300**
	FY ✓	**-1500**

Select **OK / Cancel**.

Notice that the component forces are combined. To view the component:

View / Options (or **F6**)

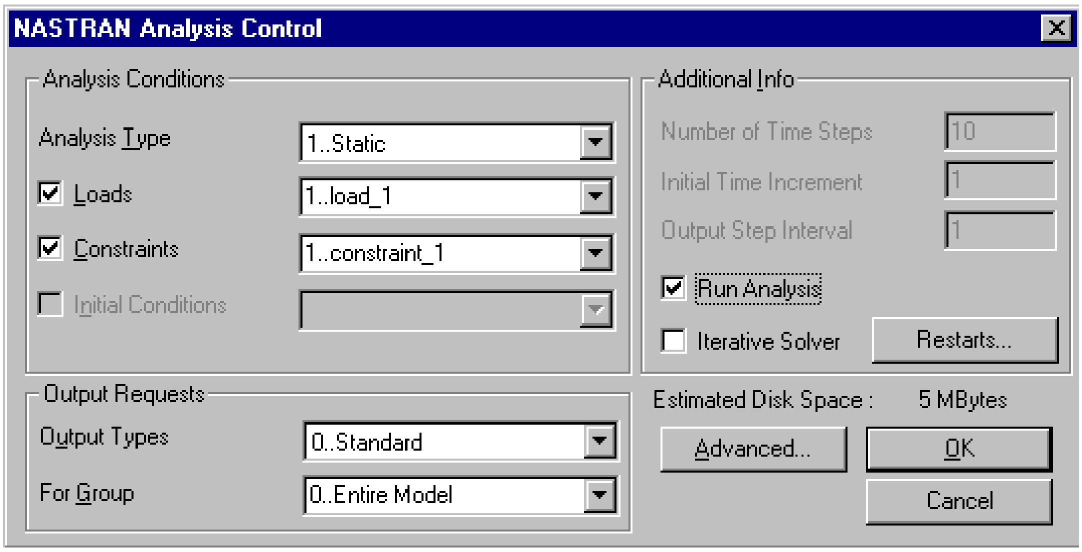


Options	**Load Vectors**
Vector Length	**Scale by Magnitude**
Options	**Load-Force**
Color / Component Mode	**Entity, Components**

Select **OK**.

8. Submit the model for analysis.

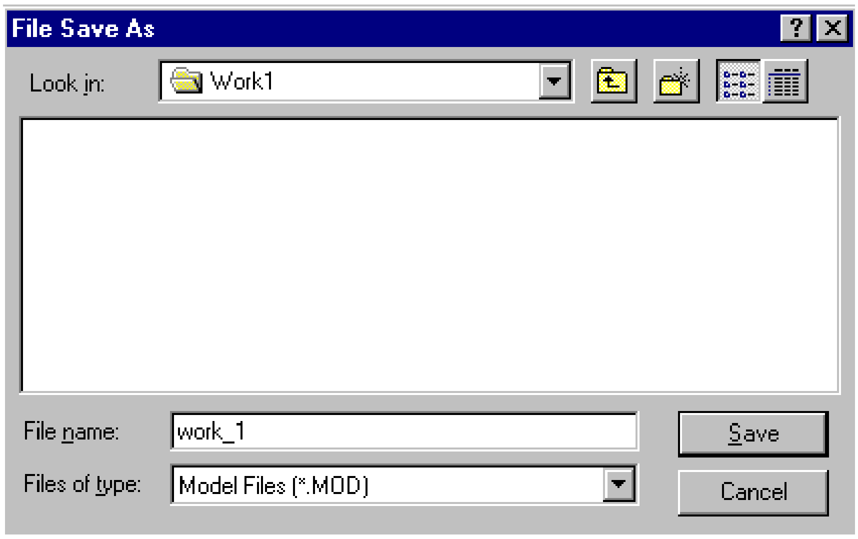
File / Analyze



Analysis Type		**Static**
Loads	✓	**load_1**
Constraints	✓	**constraint_1**
	✓	**Run Analysis**

Select **OK**.

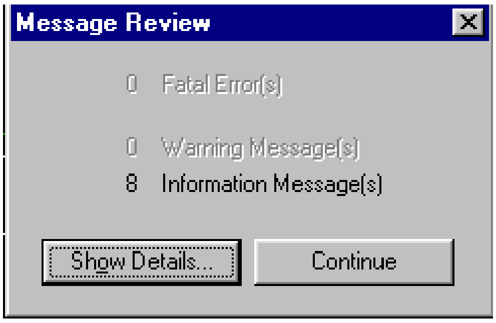
When asked if you wish to save the model, respond **Yes**.



Be sure to set the desirable working directory.

File Name work_1

Select **Save**.



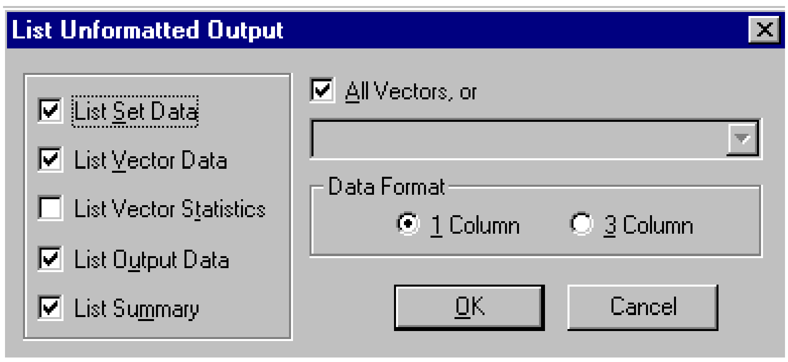
When the MSC/ NASTRAN manager is through running, MSC/ NASTRAN for Windows will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time. Then, select **Continue**.

9. List the results of the analysis.

To list the results, select the following:

List / Output / Unformatted

Select All / OK



NOTE: You may want to expand the message box in order to view the results.

Select **OK**.

Answer the following questions using the results. The answers are listed at the end of the exercise.

When there is a big list of results, a quick way to determine the results at a specified node or element is using the **List/ Output/ Query** command. The step required to answer the first question is listed below.

List / Output / Query

Output Set	**MSC / NASTRAN Case 1**
Category	**Any Output**
Entity	**Node**
ID	**7**

Select **OK**.

Double click at the bottom of the screen to see the results. Double click again to return.

What is the displacement at grid (node) 7?

Disp. X = ________________

Disp. Y = ________________

Disp. Z = ________________

What is the constraint force at grid (node) 1?

Force X = ________________

Force Y = ________________

Force Z = ________________

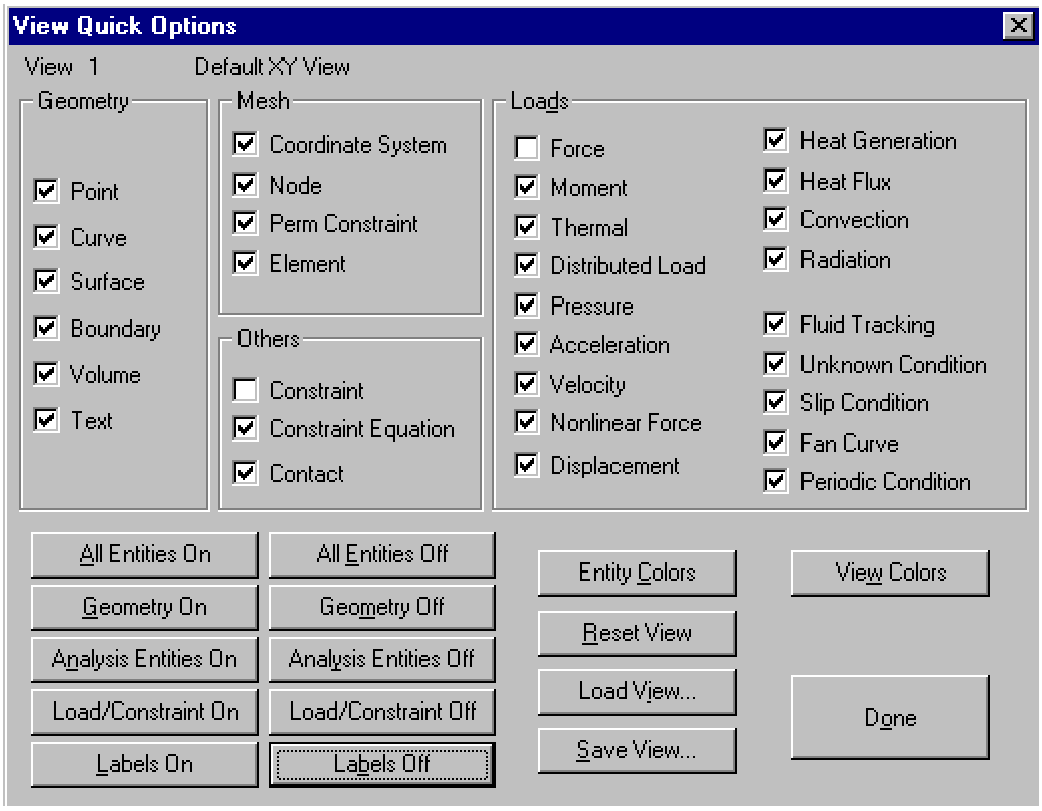
What is the axial stress for element 7?

Axial Stress = ________________

10. Display the deformed plot on the screen.

Finally, you may now display the deformed plot. First, however, you may want to remove the load and boundary constraint markers.

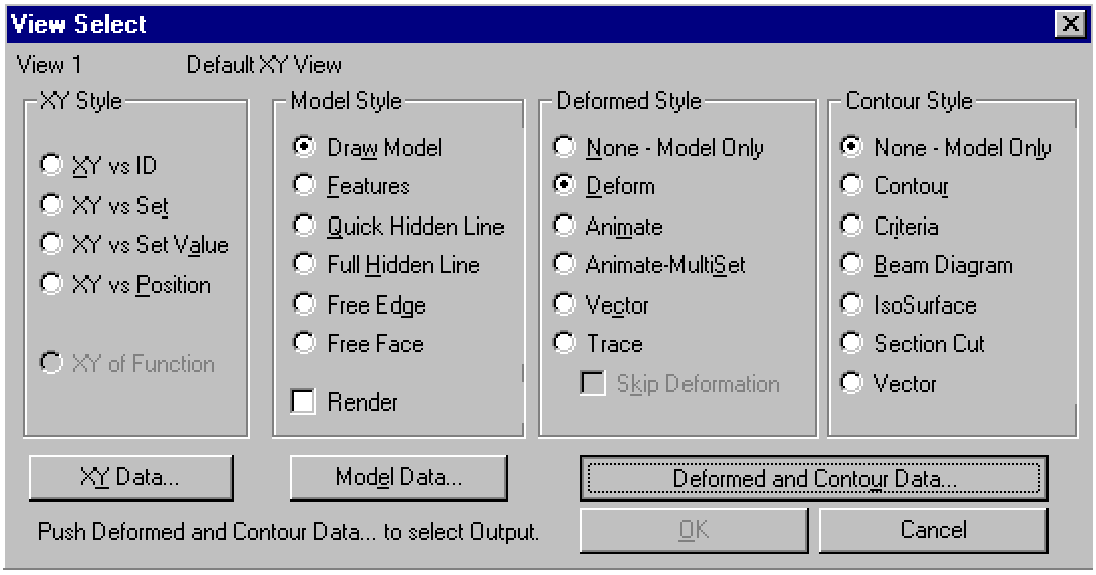
View / Options / Quick Options (or **Ctrl Q**)



☐ **Force** / ☐ **Constraint** / **Done** / **OK**

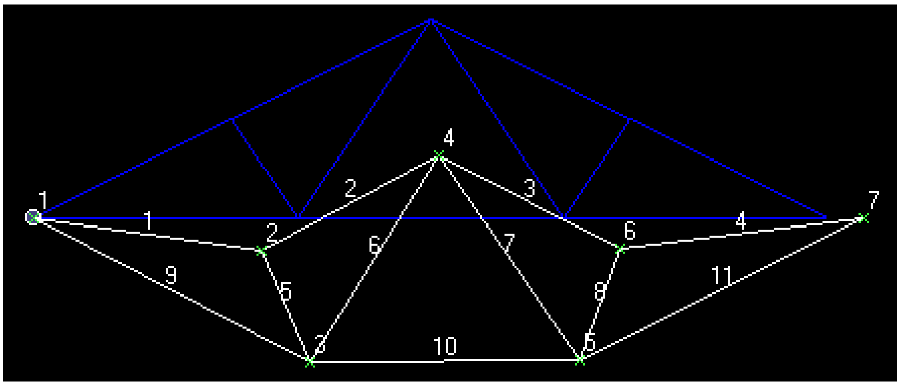
Plot the deformation of the truss.

View/ Select (or **F5**)



Deformed Style $\boxed{\textbf{Deform}}$

Select **Deformed and Contour Data / OK / OK**.



This concludes the exercise.

File / Save

File / Exit.

Answer

node 7			node 1			element 7
disp. X	**disp. Y**	**disp. Z**	**force X**	**force Y**	**force Z**	**axial stress**
0.12779	0	0	3900	2900	0	369.14