# MSC/NASTRAN

## Análise Estática de Estruturas

Eliseu Lucena Neto

# 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 lugarcomum, 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.



(with and without mid-side nodes)

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



## 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, podese 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

 $http://www.mscsoftware.com \Rightarrow mechanical solutions \Rightarrow support \Rightarrow applica$  $tion examples \Rightarrow example exercises \Rightarrow msc.nastran for windows$ 

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      | $1.76 \times 10^6$ psi |
|----------------------|------------------------|
| Poisson's Ratio      | 0.3                    |
| Cross-Sectional Area | $5.25 \text{ in}^2$    |

## 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.

| Open Model             | File                |            |   | ? )   | × |
|------------------------|---------------------|------------|---|---|---|
| Look jn: 🔁             | Work1               | <b>–</b> 🖻 |   | <u>i i i i i i i i i i i i i i i i i i i </u> |   |
|                        |                     |            |   |   |   |
|                        |                     |            |   |   |   |
|                        |                     |            |   |   |   |
|                        |                     |            |   |   |   |
|                        |                     |            |   |   |   |
| File <u>n</u> ame:     |                     |            |   | <u>O</u> pen                                  |   |
| Files of <u>type</u> : | Model Files (*.MOD) |            | • | <u>N</u> ew Model                             |   |

Turn off the workplane:

Tools / Workplane (or F2) /  $\Box$  Draw Workplane / Done

View / Regenerate (or Ctrl G).

2. Create a material called  ${\bf mat\_1}.$ 

From the pulldown menu, select  ${\bf Model}$  /  ${\bf Material}.$ 

| N. N         | ntitled       | - MSC.Na          | stran fo      | r Wind             | ows             | Eval         | uation (       | Сору -        | Not for        | Commercial Use |   |
|--------------|---------------|-------------------|---------------|--------------------|-----------------|--------------|----------------|---------------|----------------|----------------|---|
| <u>F</u> ile | <u>T</u> ools | G <u>e</u> ometry | <u>M</u> odel | Me <u>s</u> h      | M <u>o</u> dify | <u>L</u> ist | <u>D</u> elete | <u>G</u> roup | o <u>V</u> iew | <u>H</u> elp   |   |
|              | •             | ⇒ <b>1</b> -      | Coc           | rd Sys             |                 |              | q t            |               | ⊕]≣            |                | + |
|              |               |                   | <u>N</u> oc   | le                 | Ctrl+           | N            |                |               |                |                | _ |
|              | )etault       | t XY View         | <u>E</u> ler  | nent               | Ctrl+           | E            |                |               |                |                |   |
| V1           |               |                   | <u>M</u> at   | erial              |                 |              |                |               |                |                |   |
|              |               |                   | P <u>r</u> oj | perty              |                 |              |                |               |                |                |   |
|              |               |                   | Loa           | d                  |                 | •            |                |               |                |                |   |
|              |               |                   | Cor           | is <u>t</u> raint  |                 | •            |                |               |                |                |   |
|              |               |                   | <u>C</u> or   | itact              |                 | •            |                |               |                |                |   |
|              |               |                   | Opt           | imi <u>z</u> atior | ì               |              |                |               |                |                |   |
|              |               |                   | F <u>u</u> n  | ction              |                 |              |                |               |                |                |   |
|              |               |                   | <u>0</u> ut   | put                |                 | •            |                |               |                |                |   |
|              |               |                   |               |                    |                 |              |                |               |                |                |   |
|              |               |                   |               |                    |                 |              |                |               | •              |                |   |
|              |               |                   |               |                    |                 |              |                |               |                |                |   |
|              |               |                   |               |                    |                 |              |                |               |                |                |   |
|              |               |                   |               |                    |                 |              |                |               |                |                |   |
| Γ.           |               |                   |               |                    |                 |              |                |               |                |                |   |
|              |               |                   |               |                    |                 |              |                |               |                |                |   |
|              |               |                   |               |                    |                 |              |                |               |                |                |   |

| Define Isotropic Material   |  | X   |
|---|--|---|
| ID 1 Little mat_1   | Color 55 Pale                                | tte Layer Type  |
| Stiffness       Youngs Modulus, E       1.76e6       Shear Modulus, G | Limit Stress<br>Tension 0,<br>Compression 0, | Mass De <u>n</u> sity 0,<br>Da <u>m</u> ping, 2C/Co<br>0, |
| Poisson's Ratio, n <u>u</u> 0.3<br>Thermal                            | Shear 0,                                     | Re <u>f</u> erence Temp                                   |
| Expansion Coeff, <u>a</u> 0,<br>Conductivity, <u>k</u> 0.             | Functions >>                                 | Loa <u>d</u> <u>S</u> ave                                 |
| Specific <u>H</u> eat, Cp 0,  | Nonlinear >>                                 | Сору  |
| Heat Generation Factor 0,   | Phase Change >>                              | <u>O</u> K Cancel   |

| 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.

|                      | AIE Element Type    |   |
|----------------------|---------------------|---|
| ID 1 Iill            | e prop_1            | Material 1mat_1                           |
| <u>C</u> olo         | r 110 Palette       | Layer Elem/Property Type                  |
| Property Values      |                     | - Additional Options                      |
| Thicknesses, Tav     | g or T <u>1</u> 0.  | Bend Stiffness, 12I/T**3 0.               |
| blan                 | k or T <u>2</u> 0.  | TS <u>h</u> ear/Mem Thickness,ts/t 0.     |
| blan                 | k or T <u>3</u> 0.  | B <u>e</u> nding 0Plate Material          |
| blan                 | k or T <u>4</u> 0.  | Tra <u>n</u> sverse Shear DPlate Material |
| Nonstructural ma:    | ss/ <u>a</u> rea 0. | Memb-Bend Coupling 0None - Ignore         |
| Stress Recovery ( De | fault=T/2)          |   |
| To                   | p <u>F</u> iber 0.  | Loa <u>d S</u> ave <u>O</u> K             |

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

| 711  | • 1         |
|------|-------------|
| Mate | rnal        |
| muuu | 1 1 1 1 1 1 |
|      |             |

| $\operatorname{mat}$ | 1 |  |  |
|----------------------|---|--|--|
| _                    | _ |  |  |

| ement / Property T  | ype 🗙  |
|---|--|
| Parabol<br>Line Elements<br>Provide<br>Curved Tube<br>Curved Tube<br>Bag<br>Eeam<br>Curved Beam<br>Curved Beam<br>Curved Beam<br>Curved Beam<br>DOE Spring<br>DOE Spring<br>Plot Only<br>Other Elements | ic Elements<br>Plane Elements<br>O Shear Panel<br>O Membrane<br>O Bending Only<br>O Plate<br>O Laminate<br>O Plane Strain<br>O Plot Only<br>Volume Elements<br>O Axisymmetric<br>O Solid   |
| C <u>M</u> ass<br>C Mass Matri <u>x</u><br>Element Mate   | C Rigid<br>C Stiffness Matrix<br>C Slide Line  |
|   | ement / Property T<br>Parabol<br>Line Elements<br>Parabol<br>Curved Tube<br>Bag<br>Curved Tube<br>Bag<br>Curved Beam<br>Curved Beam<br>Curved Beam<br>Curved Beam<br>DOF Spring<br>DOF Spring<br>DOF Spring<br>DOF Spring<br>Mass<br>Mass Matrix<br>Element Mate |

Change the property type from  ${\bf Plate}$  element (default) to  ${\bf Rod}$  element.

Line Elements

Select  $\mathbf{OK}$ .

| Define Property - ROD Element Type | ×                               |
|------------------------------------|---------------------------------|
| ID 1Itle prop_1                    | Material 1mat_1                 |
| Color 110 Palette Layer 1          | Elem/Property Type              |
| Property Values                    |                                 |
| Area, <u>A</u> 5.25                | Coeff. for Torsional Stress 0.  |
| Torsional Constant, J 0.           | Nonstructural mass/length 0.    |
| Perimeter 0.                       | Initial Tension (Cable Only) 0. |
| Land Save Care                     |                                 |
| Loag <u>Save</u> Copy              |                                 |

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).

| Locate - E | ×                               |                                     |                  |
|------------|---------------------------------|-------------------------------------|------------------|
| × 0        | Ϋ́Ο                             | ZO                                  | Pre <u>v</u> iew |
|            |                                 |                                     | <u>0</u> K       |
| ID 1       | <u>C</u> Sys 0Basic Rectangular | <u>Parameters</u> <u>M</u> ethods ^ | Cancel           |



Repeat the process for the other 6 nodes:

| Node | Х          | Υ         | Ζ | Select |
|------|------------|-----------|---|--------|
| 2    | 144        | <b>72</b> | 0 | ОК     |
| 3    | 192        | 0         | 0 | ОК     |
| 4    | <b>288</b> | 144       | 0 | ОК     |
| 5    | 384        | 0         | 0 | ОК     |
| 6    | 432        | 72        | 0 | ОК     |
| 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)

| Define ROD Element - Enter Nodes or Se | elect with Cursor        | ×      |
|--|--------------------------|--------|
| ID 1 Color 124 Palette                 | Layer 1 Property 1prop_1 | Туре   |
| Nodes                                  | <u> </u>                 | Cancel |

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!

|          |                  |                                | 4<br>×                  |        |
|----------|------------------|--------------------------------|-------------------------|--------|
|          |                  | 2                              | × <sup>6</sup>          |        |
|          | ⊕ <sup>1</sup>   | ×3                             | 5<br>×                  | ×      |
| De       | fine ROD Element | - Enter Nodes or Sele          | ect with Cursor         | ×      |
| ĪD       |                  | 124 <u>P</u> alette <u>L</u> a | ayer 1 Property 1prop_1 | ▼ Туре |
| <u>N</u> | odes  1  ;       | 2                              | <u>0</u> K              | Cancel |
| Nodes:   | 1                | 2                              | select <b>OK</b>        |        |

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

| No | odes | Select |
|----|------|--------|
| 2  | 4    | OK     |
| 4  | 6    | ОК     |
| 6  | 7    | ОК     |
| 2  | 3    | ОК     |
| 3  | 4    | ОК     |
| 4  | 5    | ОК     |
| 5  | 6    | OK     |
| 1  | 3    | OK     |
| 3  | 5    | ОК     |
| 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

 $\mathrm{constraint}_1$ 

Select  $\mathbf{OK}.$ 

Now, define the relevant constraint for the model.

4 6 9 10 11 Entity Selection - Enter Node(s) to Select х ● <u>A</u>dd ● <u>R</u>emove ● E<u>x</u>clude +1 Select All R<u>e</u>set Pi<u>c</u>k ^ ID 🛛 by 1 to **Previous** <u>D</u>elete <u>0</u>K <u>M</u>ore Met<u>h</u>od Cancel

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

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



Select OK / Cancel.

7. Create the model loading.

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

| Untitled - MSC.Na:                                   | stran for Windows E  | valuation Copy - No  | t for Commerci            | al Use        |
|--|--|--|---------------------------|---------------|
| le <u>T</u> ools G <u>e</u> ometry                   | <u>M</u> odel Me <u>s</u> h M <u>o</u> dify  | <u>L</u> ist <u>D</u> elete <u>G</u> roup <u>)</u>                 | <u>∕</u> iew <u>H</u> elp |               |
| <u>}</u> + + + 4                                     | Coord Sys  | <b>&amp;_</b>  |                           | <u>∎∎+×</u> ∎ |
| <mark>∦</mark> Default XY Vie <del>w</del><br>1<br>1 | <u>N</u> ode Ctrl+N<br><u>E</u> lement Ctrl+E<br><u>M</u> aterial<br>P <u>r</u> operty |  |                           |               |
|  | Loa <u>d</u>   | ▶ <u>S</u> et  | Ctrl+F2                   |               |
|  | Cons <u>t</u> raint<br><u>C</u> ontact<br>Optimi <u>z</u> ation                        | <ul> <li>▶ Body</li> <li>▶ Nodal</li> <li>Nodal on Face</li> </ul> |                           |               |
|  | Function   | <u>E</u> lemental<br>Nonlinear Fo <u>r</u> ce                      |                           |               |
|  | <u>O</u> utput   | On Point   |                           |               |
|  |  | On C <u>u</u> rve<br>On Su <u>r</u> face<br>E <u>x</u> pand        |                           | 8 4           |
|  | 2 <u>5</u><br>12   | B Nonlinear Analy<br>Dynamic Analys                                | sis<br>is                 | 2 11 Z        |

Model / Load / Set (or Ctrl F2)

| Create or Activate Load Set | ×             |
|-----------------------------|---------------|
| ID 1 Iitle load_1           |               |
|                             | <u>R</u> eset |
|                             | <u>0</u> K    |
|                             | Cancel        |

#### Title

 $load_1$ 

Select  $\mathbf{OK}$ .

Now, define the relevant loading conditions.

#### Model / Load / Nodal

| Entity Selection - Enter Node(s) to Select 🛛 🛛 🗙  |          |                    |                   |                 |
|---|----------|--------------------|-------------------|-----------------|
| ● <u>A</u> dd ● <u>R</u> emove ● E <u>x</u> clude | +2<br>+4 | <u>S</u> elect All | R <u>e</u> set    | Pi <u>c</u> k ^ |
|   | +6       | <u>P</u> revious   | <u>D</u> elete    | <u></u> K       |
| <u>G</u> roup                                     |          | <u>M</u> ore       | Met <u>h</u> od ^ | Cancel          |

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

| Create Loads on Nodes  |  | ×   |
|--|--|---|
| Load Set 1 load_1  |  |   |
| Color 10 Palette   | Layer 1 Coord Sys 0Basic Re  | ctangular 💌   |
| Force<br>Moment<br>Displacement<br>Enforced Rotation<br>Velocity<br>Rotational Velocity<br>Acceleration<br>Rotational Acceleration   | Direction     Components     Vector     Along Curve     Normal to Plane     Specify  | Method<br>ⓒ Co <u>n</u> stant<br>ⓒ Variable<br><u>A</u> dvanced |
| Temperature<br>Heat Flux<br>Heat Generation<br>Static Fluid Pressure<br>Total Fluid Pressure<br>General Scalar<br>Steam Quality<br>Relative Humidity<br>Fluid Height Condition | Load         Value         Function           FX         ✓         -1300         0None           FY         ✓         -1500         0.           FZ         ✓         0.         0.           Phase         0.         0None         0None | etion Dependence  |
|  | <u></u> K  | Cancel  |

#### 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

Vector Length

Options

Color / Component Mode

Select  $\mathbf{OK}$ .

Load Vectors

Scale by Magnitude

Load-Force

Entity, Components

8. Submit the model for analysis.

#### File / Analyze

| NASTRAN Analysis Cor              | ntrol         |          |   | ×          |
|-----------------------------------|---------------|----------|---|------------|
| Analysis Conditions               |               |          | Additional Info                               |            |
| Analysis <u>T</u> ype             | 1Static       | •        | Number of Time Steps                          | 10         |
| ☑ Loads                           | 1load_1       | •        | Initial Time Increment                        | 1          |
| Constraints                       | 1constraint_1 | •        | Output Step Interval                          | 1          |
| ☐ I <u>n</u> itial Conditions     |               | ~        | 🔽 Run Analysis                                |            |
|                                   | ,             |          | T Iterative Solver                            | Restarts   |
| Output Requests                   |               |          | Estimated Disk Space :                        | 5 MBytes   |
| O <u>u</u> tput Types             | 0Standard     | <b>•</b> | <u>A</u> dvanced                              | <u>0</u> K |
| For <u>G</u> roup                 | 0Entire Model | •        |   | Cancel     |
| nalysis Type<br>ads<br>onstraints |               | Sta      | tic<br>load_1<br>constraint_1<br>Run Analysis |            |

| Χ. |
|----|
|    |

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

| File Save As           |                     |   |   |                 | ? ×                  | 1 |
|------------------------|---------------------|---|---|-----------------|----------------------|---|
| Look jn:               | 🔄 Work1             | • | £ | <del>ri</del> k | 9-8-<br>5-6-<br>8-6- |   |
|                        |                     |   |   |                 |                      |   |
|                        |                     |   |   |                 |                      |   |
|                        |                     |   |   |                 |                      |   |
|                        |                     |   |   |                 |                      |   |
| L                      |                     |   |   |                 |                      |   |
| File <u>n</u> ame:     | work_1              |   |   |                 | <u>S</u> ave         |   |
| Files of <u>type</u> : | Model Files (*.MOD) |   | • |                 | Cancel               |   |

Be sure to set the desirable working directory.

File Name

Select Save.

| 0      | Fatal Error(s)                               |
|--------|--|
| 0<br>8 | Warning Message(s)<br>Information Message(s) |
| Show D | etails                                       |

work 1

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

| Entity Selection - Select Output Set(s) to List       |                    |                |                 |
|---|--------------------|----------------|-----------------|
| <u>Add</u> O <u>R</u> emove O E <u>x</u> clude +1,1,1 | <u>S</u> elect All | R <u>e</u> set | Pi <u>c</u> k ^ |
|   | Previous           | <u>D</u> elete | <u>0</u> K      |
| Group   | <u>M</u> ore       | Method ^       | Cancel          |

Select All / OK

| List Unformatted Output   |  | × |
|---|--|---|
| <ul> <li>✓ List Set Data</li> <li>✓ List Vector Data</li> <li>✓ List Vector Statistics</li> <li>✓ List Output Data</li> <li>✓ List Summary</li> </ul> | ✓ All Vectors, or         Data Format         ● 1 Column         ○ 3 Column         ○K |   |

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 Query   | ×   |
|--|---|
| Output to List<br>Output Set 1MSC/NASTRAN Case<br>Category 0Any Output | 1     ✓     Image: Entity       1     ✓     ✓       Image: Model C Elem     More       Image: Ima |
| Output Set   | MSC / NASTRAN Case 1  |
| Category   | Any Output  |
| Entity   | Node  |

Select **OK**.

ID

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

 $\mathbf{7}$ 

What is the displacement at grid (node) 7?



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)

| View Quick Optic                               | ons  |   | ×  |
|--|--|---|--|
| View 1 D                                       | )efault XY View  |   |  |
| Geometry                                       | Mesh   | Loads   |  |
|  | 🔽 Coordinate System                                    | Force   | 🔽 Heat Generation  |
| 🔽 Point  | 🔽 Node   | Moment  | 🔽 Heat Flux  |
| Curve  | Perm Constraint  | 🔽 Thermal   | Convection   |
| Surface  | Element  | Distributed Load  | Radiation  |
| Volume   | Others<br>Constraint<br>Constraint Equation<br>Contact | <ul> <li>Pressure</li> <li>Acceleration</li> <li>Velocity</li> <li>Nonlinear Force</li> <li>Displacement</li> </ul> | <ul> <li>Fluid Tracking</li> <li>Unknown Condition</li> <li>Slip Condition</li> <li>Fan Curve</li> <li>Periodic Condition</li> </ul> |
| <u>A</u> ll Entities Or<br><u>G</u> eometry On | n <u>All E</u> ntities Off<br>Geometry Off             | Entity <u>C</u> olors<br><u>R</u> eset View   | Vie <u>w</u> Colors  |
| Load/Constraint                                | On Analysis Entities Off<br>On Load/Constraint Off     | Load Vjew<br><u>S</u> ave View  | Done   |

 $\Box$  Force /  $\Box$  Constraint / Done / OK

Plot the deformation of the truss.

#### View/ Select (or F5)



#### Deformed Style

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       |