

Appendix B

Finite Element Analysis Using NEi Nastran

B.1 INTRODUCTION

NEi Nastran is engineering analysis and simulation software developed by Noran Engineering, Inc. NEi Nastran is a general purpose finite element analysis tool with an integrated graphical user interface and model Editor which is used to analyze linear and nonlinear stress, dynamics, and heat transfer characteristics of structures and mechanical components. A complimentary copy of NEi Nastran student edition is included with this book, which is composed of two modules: NEi Fusion and NEi Nastran. NEi Fusion combines a finite element modeler, pre- and post-processing capabilities, and Nastran solvers to create an analysis package for small and medium size problems. It is good for engineers who need simulation for product development, virtual testing, design validation, and quality assurance. Parts and assemblies can be analyzed for a spectrum of structural and thermal loadings. In this chapter, we will present two tutorials using integrated NEi Fusion/NEi Nastran software and several example problems in the text using NEi Nastran alone.

B.2 GETTING START

To access NEi Fusion program on Windows systems, choose **Start > All Programs > NEi Fusion > NEi Fusion v1.2** (Note that the version number v1.2 may be different for other versions). The startup screen will only show **New** and **Open** icons in the standard toolbar. In order to see main windows, choose **File > New** in the main menu. Choose **Part** and click **OK** in the New Document window. This command will open NEi Fusion window as shown in Figure B.1. The NEi Fusion window is composed of different areas. The graphic area shows geometries and finite elements as well as results of simulation. The main menu contains primary NEi Fusion functions. The standard toolbars collects commands that are frequently used. The Tab Menu on the left of the screen provides model related functions. In order to finish NEi Fusion, choose **File > Exit** command. NEi Fusion has its full documents as well as tutorials in help menu.

B.3 PLATE WITH A HOLE ANALYSIS

A. Introduction

This example problem illustrates the use of NEi Fusion/NEi Nastran for a simple static analysis. You will learn how to build a solid model of a plate with a hole using NEi Fusion

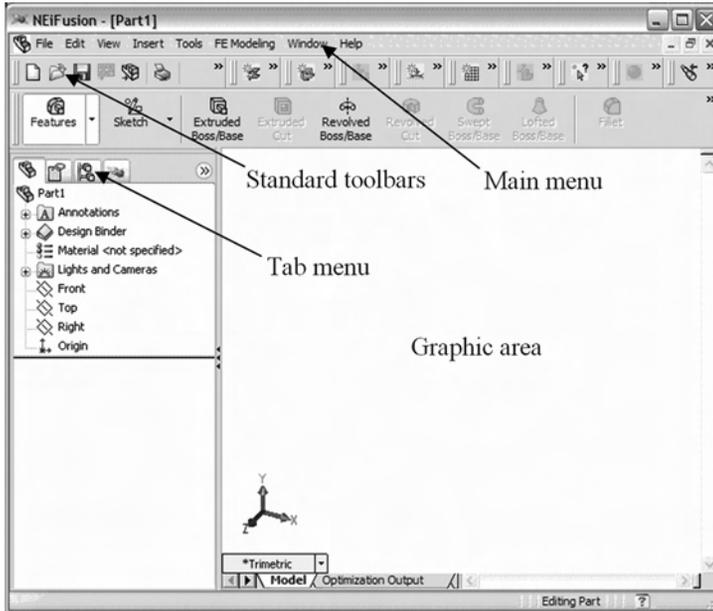


Figure B.1 NEi Fusion main windows

pre- and post-processor, perform the analysis with NEi Nastran, and examine the results with NEi Fusion.

Figure B.2 shows a plate with a hole. The plate is fixed on the left edge and a uniform pressure is applied on the right edge. The magnitude of pressure is chosen such that the total applied force is 300 lb. The purpose of finite element analysis is to estimate the stress concentration along the hole. Even if the model is three-dimensional solid, we will use plane stress elements because stress along the thickness direction is negligible.

B. Creating the Part

1. Create a new part by selecting **File**> **New** in the main menu. Choose **Part** and click **OK** in the New Document window.
2. Click **Tools**> **Options...** to open System Options window. Click **Document Properties** tab and select **Units** on the left-hand side tree. Change the units to **IPS (inch, pound, second)** in Unit system. Click **OK** to close Document Properties window.

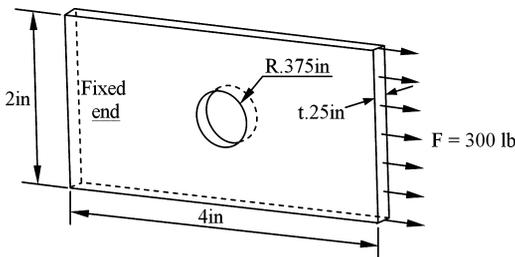
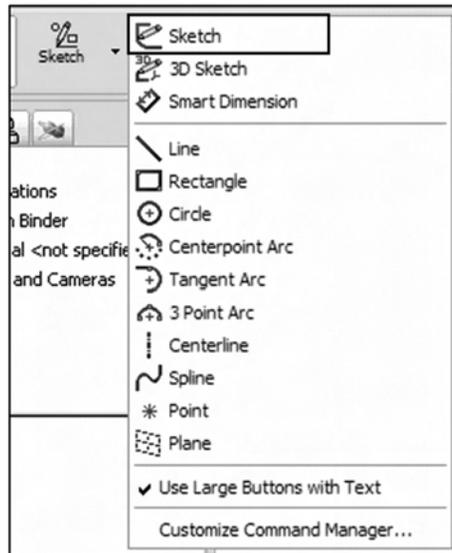
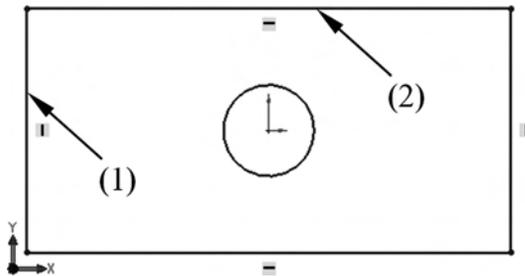


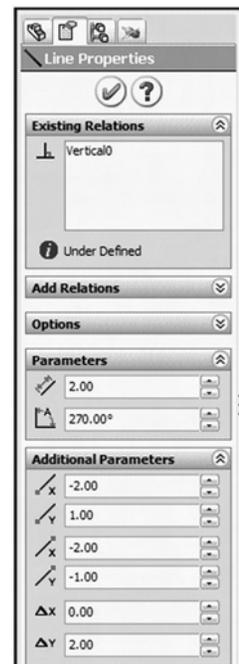
Figure B.2 Plate with a hole model



- In the standard toolbar, select pull-down button of **Sketch** and choose Sketch in the pull-down menu. Select **Front** plane in the graphic window as a sketch plane. Note that the Front plane is x-y plane in the global coordinate system.
- Draw a rectangle using rectangle icon in the toolbar. It is unnecessary to be a precise dimension yet. Draw a circle inside the rectangle using circle icon in the toolbar.



- The vertical and horizontal line symbols on the rectangle represent that these lines are either vertical or horizontal. Choose the left vertical line (1) of the rectangle. On the Tab menu area, Line Properties window will appear. Change the starting coordinates to $x = -2$ and $y = 1$ and length = 2. Choose the top horizontal line (2) of the rectangle. Change the starting coordinates to $x = -2$ and $y = 1$ and length = 4. Now the rectangle is properly located.
- Choose the circle and make its center locations (0, 0) and radius 0.375.
- Click **Exit Sketch** icon in the standard toolbar to leave the sketch mode.
- In the standard toolbar, select pull-down button of **Feature** and choose **Extrude Boss/Base** command.
- In the Extrude window, change thickness = 0.25 in and click **OK** button to generate a solid.



10. Save the model by selecting **File > Save** menu. Provide “PlateHole.SLDPRT” as the name of the part.



C. Pre-Processing the Model

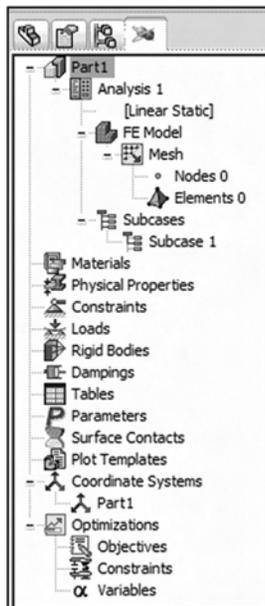
Using the solid model that we created in the previous section, we will apply (1) material properties, (2) physical properties, (3) boundary conditions, (4) external forces, and (5) mesh the geometry. These steps are called pre-processing and will be done with the NEi-Fusion program.

Switching to NEi Fusion

1. The NEi Fusion Modeler tab view looks as shown below. Choose the NEi Fusion tab at the top of the tree view to switch to the NEi Fusion environment.



2. The NEi Fusion tree view looks as shown in the figure below.



Select Materials

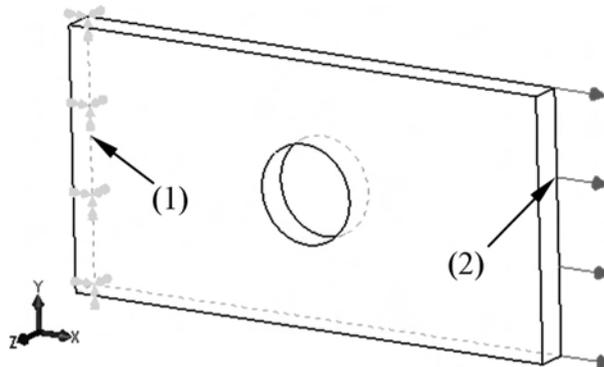
1. Right click on **Materials** in the entity list, and select **New** to create a new material.
2. In the Material window, type in **Material1** in Name section.
3. Change the Young's modulus $E = 2.8E7$ and Poisson's ratio $\nu = 0.3$. Leave shear modulus G empty as it will be calculated using previous two constants.
4. Click **OK** (checkmark icon) to leave material section.

Apply the Constraints

1. We will create the mesh on the back face of the plate (This is the plane whose z-coordinates are zero). Change the view to **Hidden Line** using the icon in the standard toolbar.



2. Right-click on **Constraints** in the Entity List and select **New**.
3. In Constraints window, type in **Constraint1** in Name section.
4. Select the left edge of the back face (1) as shown in the figure below. Click the **checkmark** icon to add the constraint to the finite element model.



Apply the Load

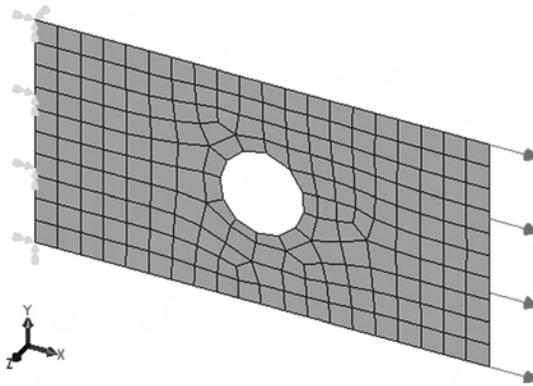
1. The load is a 300 lbf force applied at the right edge of the plate in x-direction. Right-click on **Loads** in the Entity List and select **New**.
2. In Load window, type in **Load1** in Name section.
3. Select the right edge of the back face (2) as shown in the figure. Type in 300 in **Fx** section. Click the **checkmark** icon to add the load to the finite element model.

Note that the constraint and load are defined before generating finite elements. The nodal constraints and forces will be calculated when the model is meshed using finite elements. If the loads and boundary conditions are defined in the solid model, it is unnecessary to change them when different element sizes are used.

Automatic Mesh Generation

1. NEi Fusion can create mesh and update the boundary conditions (BC) automatically with feature changes in the model.

2. Right-click on **Mesh** in the entity list, and select **Edit** to define mesh parameters.
3. In Mesh window, type in 0.2 in **Element Size (in)** section. NEi Fusion will try to make elements whose edges are close to 0.2 in.
4. Unselect the check mark in **Solid Elements** and select **Shell Elements** because we are going to make plane elements.
5. Click rectangular box under **Quadrilateral** section, and select the back face of the part in the graphic area. The rectangular box will now list **Face <1>**. This means the back face will be meshed using quadrilateral elements. You may need to rotate the part in order to select the back face. In the graphic window, hold the middle mouse button and move the mouse around. You will see dynamic rotation of the part.
6. Click the **checkmark** icon to add the mesh to the finite element model.
7. At the top of the entity list, right-click on Part1 and select **Hide/Display Body**. This will turn off the solid model and only show finite elements.



Define Element Properties

1. Right-click on **Physical Properties** in the entity list and select **New**.
2. In the **Physical Property** form, type in **PlaneStress1** for Name section; select **Shell Elements** under Geometry Type; and type in 0.25 for Thickness.
3. Click the **checkmark** icon to add the physical property to the finite element model.

Setup the Analysis

1. In the entity list, right-click on **Analysis 1** and select **Edit**.
2. In Analysis window, type in **First try** in the Name section and **Plate with a hole** in the Title Section.
3. Click the **checkmark** to leave the analysis setup.

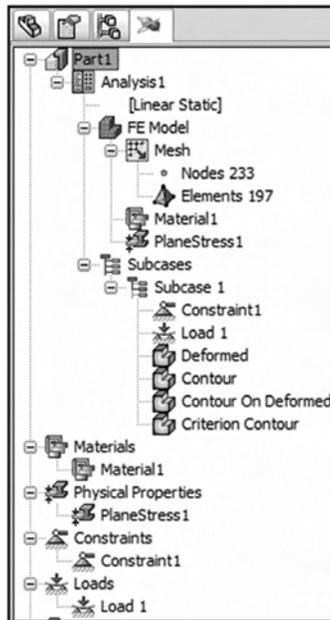
D. Running the Analysis

1. Right-click on the analysis name, **First try** and select **Solve in NASTRAN**.
2. This option allows the model to be run in a hidden mode; the default name of the analysis input file would be the same as the saved part file name, **PlateHole.NAS**.

In the hidden mode, the NEi Nastran engine runs the model and displays the progress in the NEi Nastran Output window inside NEi Fusion Modeler. Depending on the system, it may take a few minutes to run. Results are automatically loaded into NEi Fusion Modeler after the model is solved.

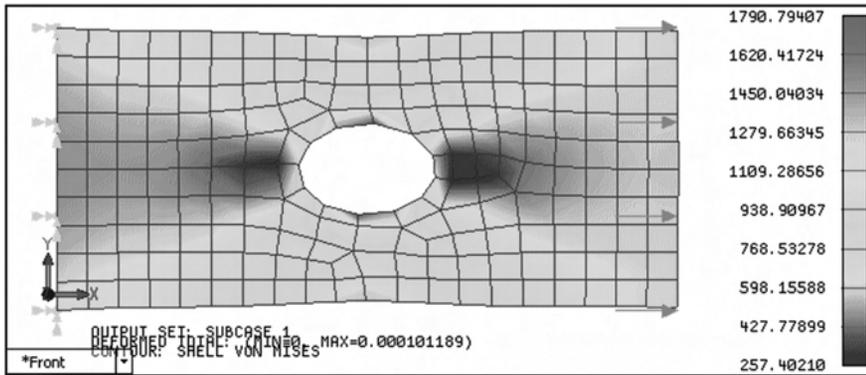
E. Post-Processing the Results

1. If you are in the NEi Nastran Output window, click on the **Model** tab at the bottom of your screen to view the Model from the calculations.



After the solution is complete, NEi Fusion places four different contour plots under **Subcase 1** entity. The tree view looks as shown in the figure after the results are loaded.

2. Right-click on the **Contour on Deformed** plot under **Subcase 1** and select **Edit**.
3. Select **SHELL VON MISES** in the **Result Data** of **Contour Options**. This will plot the von Mises stress contour plot.
4. Type in 10 in Levels section.
5. Unselect Min/Max checkmark.
6. Click the checkmark icon at the top of the Plot window.
7. Right-click on the **Contour on Deformed** plot under **Subcase 1** and select **Display**. The von Mises stress contour plot is displayed on the deformed geometry, as shown in the figure. Compared to the nominal stress of 600 psi, the concentrated stress is about 1790 psi, which is consistent to the traditional stress concentration factor of 3.



8. In the main menu, select **File > Save** to save the model file.
9. In the main menu, select **File > Exit** to leave the NEi Fusion program. This will end the tutorial.

B.4 STATIC ANALYSIS OF BEAMS¹

A. Introduction

In this tutorial, we will learn how to perform a linear static analysis of a trailer model composed of beams, as shown in Figure B.3. Through this analysis, the user will learn how to use beam and bar elements in NEi Fusion/NEi Nastran. The units used in this analysis are IPS (inches, pounds, and seconds). The effects of gravity are considered in this model.

B. Opening the Model

1. From the main menu select **File > Open** and open **Tutorial 7.SLDPRT** file located in the “**C:\Program Files\NEi Fusion\Application\Tutorial**” folder.

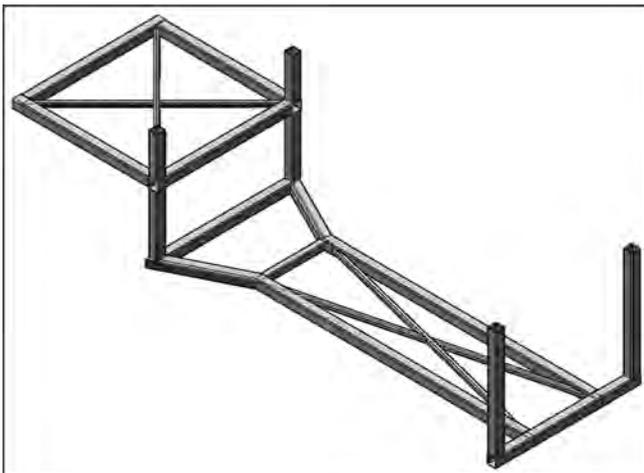


Figure B.3 A trailer model composed of beams

¹ Courtesy of Noran Engineering, Inc.

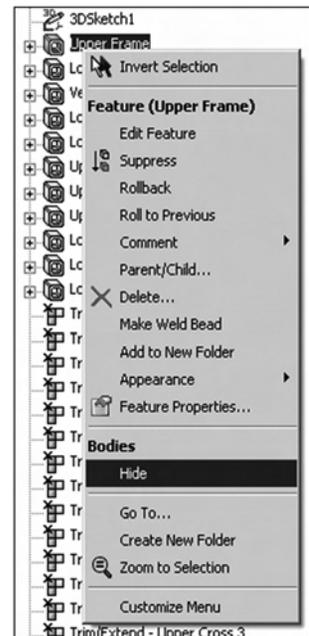
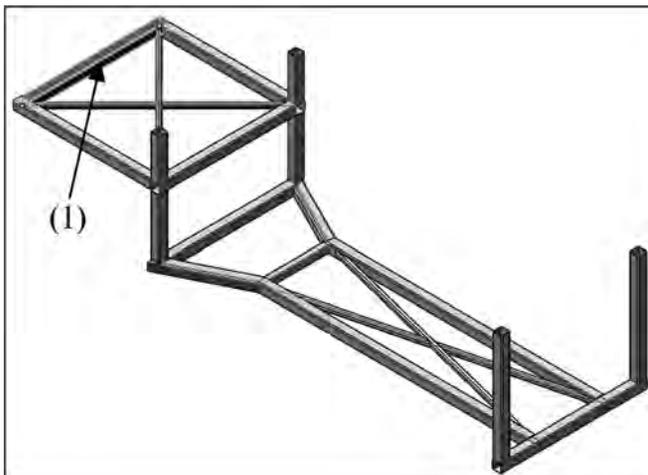
2. Click **File > Save As** and save the part as **“Tutorial 7 Trailer Frame.SLDPRT”** in the working folder. This way the original part can be reused.
3. Switch to the NEi Fusion by clicking  tab in the Tab menu.

C. Define Materials

1. Create a new material by loading AISI 304 Steel from the material library.
2. Right-click on **Materials** and choose **New**, then click on **Load**.
3. Browse to the location of the material library file, **“C:\Program Files\NEi Fusion\Application”** and open the **“solidworks materials.sldmat”** file.
4. The material tree is populated with the available materials. Under the **Steel** category, select **AISI 304**.
5. Delete the **G** value from the Structural properties field under the tree.
6. Click the **checkmark** icon to load the material.

D. Define Element Properties

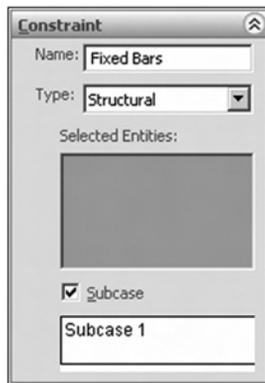
1. In the tree view, right-click on **Physical Properties** and choose **New**.
2. Select Line Elements under Geometry Type.
3. The trailer frame is composed of structural members created using the NEi Fusion Weldments Tools. By creating a line element property, you will be able to create one-dimensional elements that have the physical properties based on the Structural Member cross section and the material property. Select a structural member (1) on the model and click on the NEXT  arrow to create another structural member physical property.



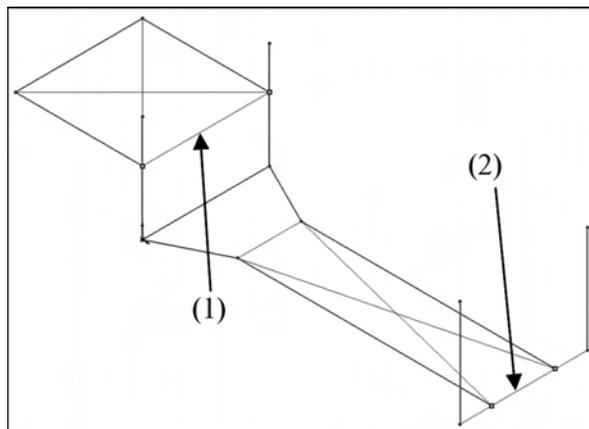
4. Repeat step 3 until all of the structural members have been selected. Click the **checkmark** instead of the **NEXT** arrow to create the final physical property. In total there should be 25 physical properties.

E. Define Constraints

1. To make it easier to define the constraints and mesh the model, you will hide the structural members. Switch to the Feature Manager Design Tree by clicking  in the Tab menu.
2. In order to hide structural members, it will be convenient to add the **Hide** feature to the **Structural Member** right-click menu. To do this right click on a structural member “Upper Frame” (it can be distinguished by the  icon beside them) and click on double arrows at the bottom to expand the full menu. Click on **Customize Menu**.
3. Check the **Hide** option. Now the Hide feature will be shown to the **Structural Member** right-click menu.
4. Right-click on a **Structural Member** in the tree and select **Hide**. Repeat for other structural members until all are hidden. Now all structural members are displayed as lines.



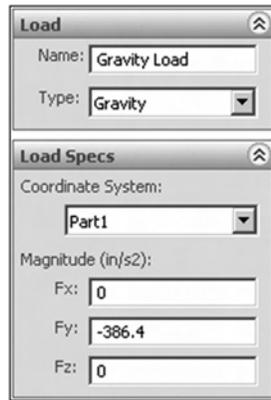
5. Switch to the NEi Fusion  tab
6. In the tree view, right-click on **Constraints** and choose **New**.
7. Change the **Name** to **Fixed Bars**.
8. Select the lines (1) and (2) to fix as shown in the figure below.



9. Click the **checkmark** to create the fixed constraint.

F. Define the Loads

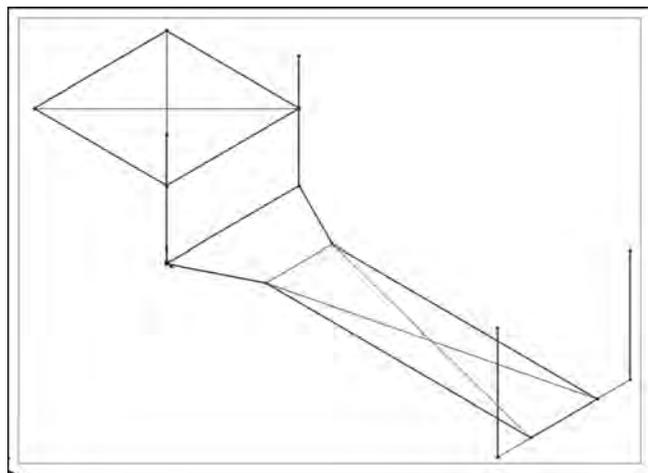
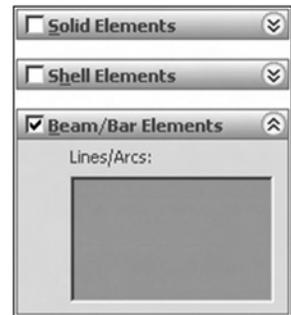
1. Right-click on the **Loads** and choose **New**. Change the **Name** to **Gravity Load**.



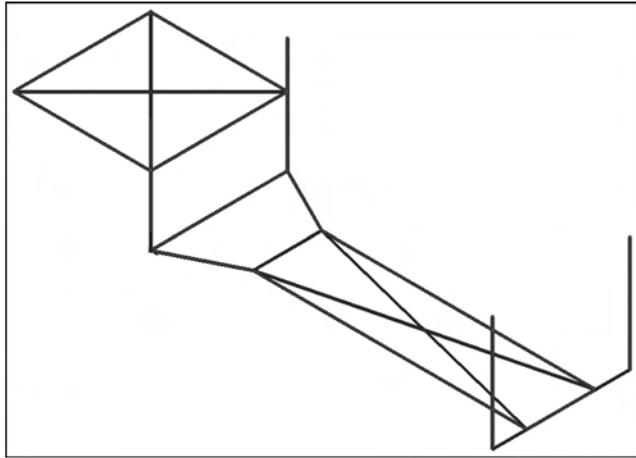
2. Under **Load**, select **Gravity** for **Type**. Type in **-386.4** in the **Fy** field under **Magnitude**.
3. Click the checkmark to create the load.

G. Mesh the Model

1. Right-click on **Mesh** and choose **Edit**. Type **4** for **Element Size**.
2. Uncheck the **Solid Elements** checkbox and check the **Beam/Bar Elements** checkbox. Click on the **Lines/Arcs** box in the **Beam/Bar Elements** form so that it is active.
3. Select all the sketch lines by clicking on the graphics area and drag a box over the entire model. This will select all the lines within the box to create the **Beam/Bar Elements**.

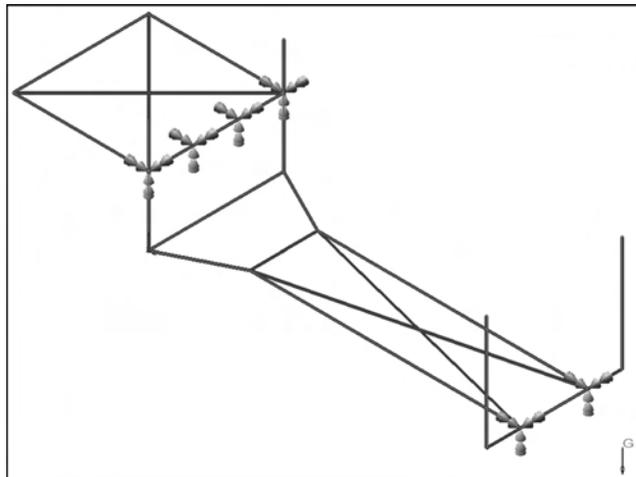


4. Click the checkmark to generate the mesh.
5. The mesh should look like the one shown below.



H. Add Entities to Model

1. Drag and drop constraint **Fixed Bars**, and load **Gravity Load**, one by one onto **Subcases**.
2. Now the model looks like as shown in the image below.



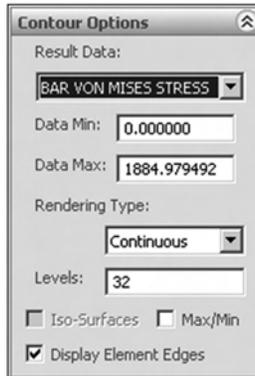
NOTE: By selecting Subcase 1 when defining your loads and constraints, this section could have been skipped.

I. Run the Analysis

1. Right click on **Analysis 1** and select **Solve in Nastran**.

J. Post-Process Results

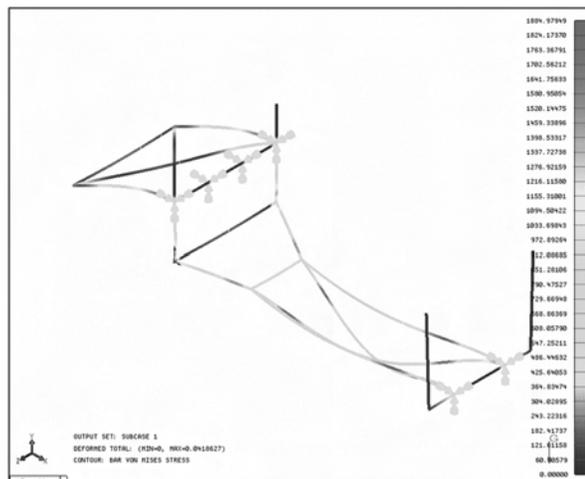
1. The results are automatically loaded into NEi Fusion.



2. Right-click on **Contour On Deformed** under Subcase 1 and choose **Edit**.
3. Select BAR VON MISES STRESS for the Results Data under Contour Options. Deselect the Max/Min checkbox.



4. Make sure **Deformed** and **Contour** are selected under **Name**. Click on **Display**. Note the results.
5. Hide the sketches by switching to the Feature Manager Design Tree , and hiding the sketches similar to how you hid the structural members. The results should look like the image below.



This concludes the tutorial. The major topics that were covered in this tutorial were as follows:

- Running an analysis of a bar/beam structure
- Create line element physical properties based on existing structural elements created with NEi Fusion Weldments Tools
- Using gravity loads

B.5 EXAMPLES IN THE TEXT

NEi Nastran Unser Interface

The finite element modeling and analysis can also be performed using NEi Nastran Editor. The NEi Nastran Editor is a utility that allows the user to manage analysis files, monitor analysis progress, queue multiple analysis jobs, change NEi Nastran settings and options, and access the NEi Nastran help file. NEiNastran Editor can be launched from **Start> All Programs> NEi Nastran> NEi Nastran Editor** in Windows system.

In the large **Editor** View of the NEi Nastran Editor (see Fig. B.4) you should see the **Errors/Warnings** View. In this view you will see a listing of any errors and warnings generated during the analysis. You can see more information by selecting from the several tabs at the bottom of the Editor View.

Click the **Nastran** tab and you will see the Model Input File. This is the file that NEi Fusion prepares when you choose **Solve in Nastran** command. It contains all of the pre-processed model data needed by the NEi Nastran solver to perform the analysis.

The **Analysis** View shows the status information provided by the solver during the analysis.

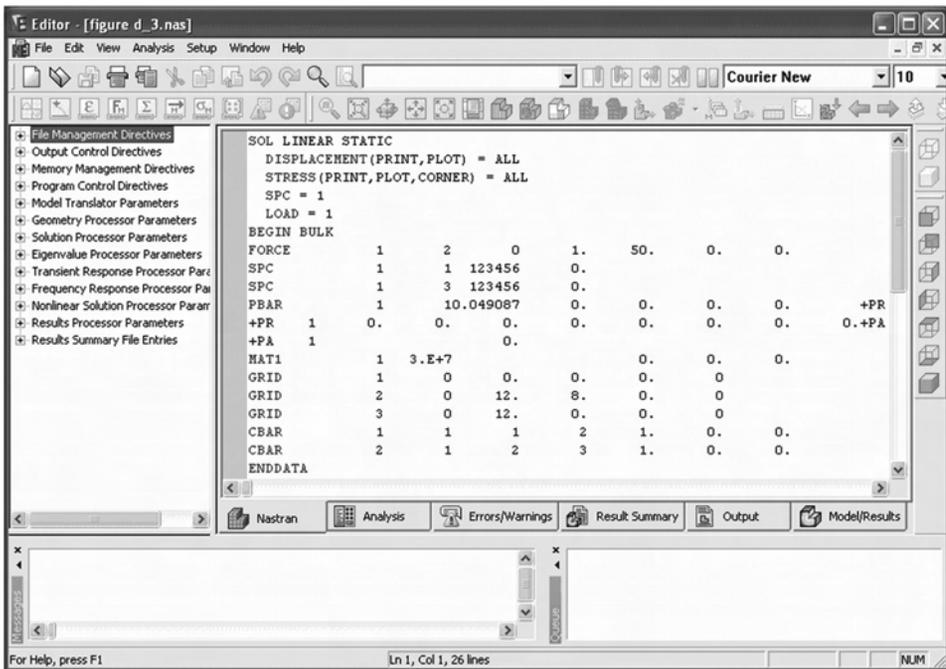


Figure B.4 NEi Nastran Editor window

Click the **Result Summary** tab and the Result Summary File appears in the view. The Results Summary File is a report of any runtime errors or warnings, the model's properties, the number of elements, number of degrees of freedom, maximum aspect ratios of the elements, total mass, epsilon, strain energy, extreme values of various stresses, and other data. The report's contents can be interrogated or printed from the Editor.

The **Output** View has the listing of the output file that is written by NEi Nastran when the analysis is complete. This file contains all of the analysis results in an ASCII format.

To the left of the **Editor** View is the **Options** View. Here the Model Initialization File settings can be viewed or modified. These settings allow you to configure NEi Nastran.

The window on the bottom is the **Messages** Window. The messages displayed here come from the NEi Nastran Editor and solver.

NEi Nastran Model Input File Format

The first step in performing analysis using NEi Nastran is to generate the Model Input File, which defines the geometry, material properties, boundary conditions, and applied loads. The Model Input File is an 80 column ASCII text file and can be created using NEi Nastran Editor or any text editors. The Model Input file also specifies analysis procedures as well as outputs. The Model Input file is divided into two distinct sections: the Case Control Section and the Bulk Data Section. Input in the Case Control Section is referred to as a command and in the Bulk Data Section as an entry. The Case Control section must appear first, followed by Bulk Data section.

The Case Control Section begins with the first command and ends with the command, BEGIN BULK. It defines the subcase structure for the problem, makes selections from the Bulk Data Section, defines the output coordinate system for element and grid point results, and makes output requests for the Model Results Output File.

The Bulk Data section begins with the entry following BEGIN BULK and ends with the entry ENDDATA. It contains all of the details of the structural model and the conditions for the solution. BEGIN BULK and ENDDATA must be present even though no new bulk data is being introduced into the problem or all of the bulk data is coming from an alternate source, such as user-generated input. The format of the BEGIN BULK entry is either a free field or a structured field. Data are separated by comma in free field, while each data is written within eight columns in structured field.

Plane Truss Example 1

In this section, the two-bar truss in Example 2.4 is solved using NEi Nastran Editor. The problem we are analyzing is shown in Figure 2.11 in Chapter 2 and the corresponding NEi Nastran Model Input File is shown in Table B.1. The Case Control section has five commands. The first command "SOL LINEAR STATIC" specifies that the linear static analysis will be employed. Next two commands request for outputs. In this case, displacements of all nodes and stresses of all elements will be included. Also these data will be available in the output file as well as database.

The next "SPC=1" command directs NEi Nastran to apply constraints defined by the SPC entry with an identification number (ID) of 1 in the Bulk Data Section. The "LOAD=1" command directs NEi Nastran to apply loading defined by the FORCE entry with an ID of 1 in the Bulk Data Section.

The start of the Bulk Data Section is denoted by the BEGIN BULK delimiter and the end, the ENDDATA delimiter. Both are delimiters are required. The model's geometry is defined via the GRID entry, which corresponds to node. Each grid point coordinate is defined in the default basic coordinate system. The format for GRID entry is:

Table B.1 NASTRAN Commands for Plane Truss Example 1

```

SOL LINEAR STATIC
DISPLACEMENT(PRINT, PLOT) = ALL
STRESS(PRINT, PLOT, CORNER) = ALL
SPC = 1
LOAD = 1
BEGIN BULK
GRID          1          0          0.  0.  0.  0
GRID          2          0         12.  8.  0.  0
GRID          3          0         12.  0.  0.  0
CBAR          1          1          1  2  1.  0.  0.
CBAR          2          1          2  3  1.  0.  0.
PBAR          1          10.04908  7  0.  0.  0.  0.
MAT1          1          3.E+7          0.  0.  0.
FORCE         1          2          0  1.  50.  0.  0.
SPC           1          1          123456  0.
SPC           1          3          123456  0.
ENDDATA
    
```

GRID	ID	CP	X	Y	Z	CD			
------	----	----	---	---	---	----	--	--	--

ID is non-overlapping node number, CP = 0 for the default basic coordinate system, and (X, Y, Z) is the coordinates of the node, and CD is the displacement coordinate system.

Element connectivity is defined via the CBAR entries. The format of CBAR entry is:

CBAR	EID	PID	GA	GB	X1	X2	X3		
------	-----	-----	----	----	----	----	----	--	--

EID is the element identification number, PID is property identification number of PBAR, GA and GB are two grid points, and (X1, X2, X3) are vector that defines the local x-y plane in the displacement coordinate system.

The properties of BAR element are defined using PBAR entry. The format of PBAR entry is:

PBAR	PID	MID	A	I1	I2	J	NSM		
------	-----	-----	---	----	----	---	-----	--	--

PID is the property identification number, MID is the material identification number, A is the cross-sectional area, I1 and I2 are area moment of inertia, J is the torsional constant, and NSM is nonstructural mass per unit length.

The material property of the BAR element is defined using MAT1 entry, which defines the material properties of linear, temperature-independent, isotropic materials. The format of MAT1 entry is:

MAT1	MID	E	G	NU	RHO	A	TREF	GE	
------	-----	---	---	----	-----	---	------	----	--

MID is the material identification number, E the Young’s modulus, G the shear modulus, NU Poisson’s ratio, RHO the density, A the thermal expansion coefficient, TREF the reference temperature, and GE structural damping coefficient.

The applied load can be specified using FORCE entry. The format of FORCE entry is:

FORCE	SID	G	CID	F	N1	N2	N3		
-------	-----	---	-----	---	----	----	----	--	--

SID is the load set identification number, G the grid point that the force is applied, CID the coordinate system, F the load scale factor, (N1, N2, N3) is the vector of load in CID coordinate system.

The displacement boundary conditions are applied using two SPC entries. The format of SPC entry is:

SPC	SID	G1	C1	D1	G2	C2	D2		
-----	-----	----	----	----	----	----	----	--	--

SID is the identification number of single point constraint set, G1 and G2 are Grid points, C1 and C2 are component numbers of global coordinates, and D1 and D2 are enforced displacements. For example, when all six DOFs are fixed, C1 and C2 have the value of 123456.

Once, the Model Input File is prepared, finite element analysis can be performed by selecting **Analysis > Run** command or simply clicking **F5** key. Analysis progress will be shown in Analysis window. Once the analysis is finished, **Errors/Warnings** window will show any errors or warnings that occur during analysis.

The Output window shows analysis results. There is a quite amount of information in the output. By scrolling down the output window, we can find the nodal displacement and element stress. Since Nodes 1 and 3 are fixed, their displacements will be zero. Node 2 is the only free node, whose displacement is

GRID	T1	T2	T3	R1	R2	R3
2	8.28E-4	-1.81E-4	0.0	0.0	0.0	0.0

Note that T1, T2, and T3 are displacements in X, Y, and Z-coordinates, respectively, and R1, R2, and R3 are rotations.

The axial stresses of two elements can also be found in the Output window.

Element	E1	E2
Axial Stress	1.22E+3	-6.79E+2

Plane Truss Example 2

In this section, a nine-bar truss in Figure B.5 is solved using NEi Nastran Editor. The corresponding NEi Nastran Model Input File is shown in Table B.2. The Case Control section is identical to that of the two-bar truss example.

In the Bulk Data section, the model has six GRID entries, nine CBAR entries. But, their definitions are similar to that of the previous example. We used Young's modulus of $E = 2.9 \times 10^7$ psi. Since 1,200 lb force is applied in the negative y-axis at Node 3, the FORCE entry has -1200 for N2 column. Also, 400 lb force is applied in N1 column of Node 6. Since Node 1 is fixed, all six DOFs are constrained, while Node 4 is allowed to move in the x-coordinate direction.

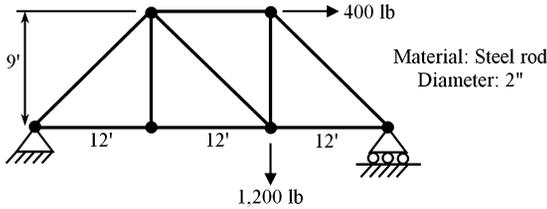


Figure B.5 Nine-bar truss model

After finishing finite element analysis, the displacements and stresses of the model can be found in the Output window. However, NEi Nastran Editor provides graphical post-processing capability.

Select Model/Results window. In the **Options** window (left to the main screen), Results> NEiNastran>Vector Results> Total Translation. This command will produce a contour plot of displacement. In the Toolbar, click “Display Contour” icon and “Display Deformed View” icon. The main window now shows the deformed geometry of the model, as shown in Figure B.6.

Portal Frame Analysis

Consider the plan frame in Figure B.7. All frames have the same material properties. The horizontal frame has different cross-sectional geometry from two vertical frames. All frames are made of the same material with Young’s modulus $E = 210 \text{ GPa}$. Two vertical frames have area $A_1 = A_2 = 4.53 \times 10^{-3} \text{ m}^2$ and moment of inertia

Table B.2 NASTRAN Commands for Plane Truss Example 2

SOL LINEAR STATIC						
DISPLACEMENT(PRINT, PLOT) = ALL						
STRESS(PRINT, PLOT, CORNER) = ALL						
SPC = 1						
LOAD = 1						
BEGIN BULK						
GRID	1	0	0.	0.	0.	0
GRID	2	0	144.	0.	0.	0
GRID	3	0	288.	0.	0.	0
GRID	4	0	432.	0.	0.	0
GRID	5	0	144.	108.	0.	0
GRID	6	0	288.	108.	0.	0
CBAR	1	1	1	2	0.	1. 0.
CBAR	2	1	2	3	0.	1. 0.
CBAR	3	1	3	4	0.	1. 0.
CBAR	4	1	1	5	0.	1. 0.
CBAR	5	1	2	5	1.	0. 0.
CBAR	6	1	3	5	0.	1. 0.
CBAR	7	1	5	6	0.	1. 0.
CBAR	8	1	3	6	1.	0. 0.
CBAR	9	1	4	6	0.	1. 0.
PBAR	1	1	3.14159	0.	0.	0. 0.
MAT1	1	2.9E+7				
FORCE	1	3	0	1.	0.	-1200. 0.
FORCE	1	6	0	1.	400.	0. 0.
SPC	1	1	123456	0.		
SPC	1	4	23456	0.		
ENDDATA						

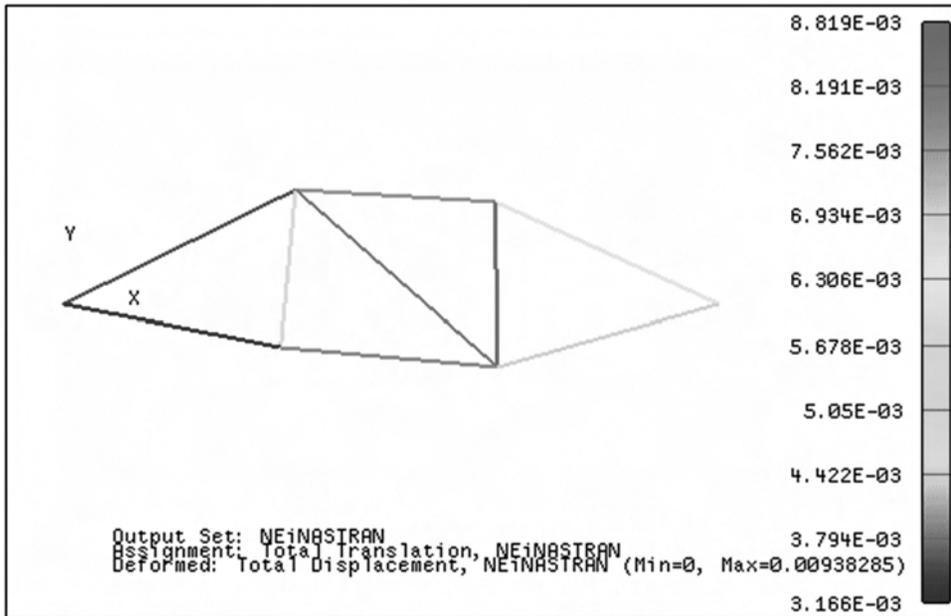


Figure B.6 Displacement contour plot of nine-bar truss

$I_1 = I_2 = 2.51 \times 10^{-5} \text{ m}^4$. The horizontal frame has area $A_3 = 1.43 \times 10^{-2} \text{ m}^2$ and moment of inertia $I_3 = 3.309 \times 10^{-4} \text{ m}^4$. Uniformly distributed load $p = 75 \text{ kN/m}$ is applied at the horizontal frame, and a concentrated force $F = 1 \text{ kN}$ is applied in the horizontal direction at Node 2.

The NEi Nastran commands for the portal frame example are shown in Table B.3. Note that FORCE command is added in the CASE Control section, which will print element forces (bending moment, shear force, and axial force). In the Bulk Data section, the same CBAR elements are used for the frame elements. In fact, CBAR can be used for truss, beam and frame elements. In PBAR entry, not only for the cross-sectional area, but also the moment of inertia column is provided because the element will behave like the combined uniaxial bar and beam element. In addition to the FORCE entry, PLOAD1 entry is provided, which defines distributed loads to CBAR elements at user chosen points along the axis. The format of PLOAD1 entry is:

PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	
--------	-----	-----	------	-------	----	----	----	----	--

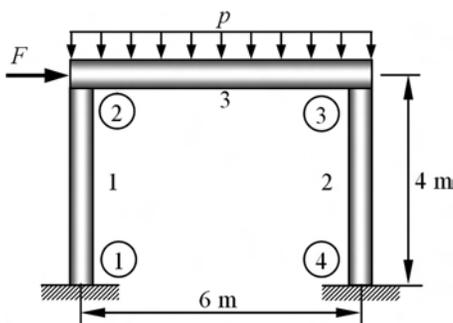


Figure B.7 Plane frame structure under distributed and concentrated load

Table B.3 NASTRAN Commands for Portal Frame

```

SOL LINEAR STATIC
DISPLACEMENT(PRINT, PLOT) = ALL
STRESS(PRINT, PLOT, CORNER) = ALL
FORCE(PRINT, PLOT, CORNER) = ALL
SPC = 1
LOAD = 1
BEGIN BULK
GRID      1      0      0.      0.      0.      0
GRID      2      0      0.      4.      0.      0
GRID      3      0      6.      4.      0.      0
GRID      4      0      6.      0.      0.      0
CBAR      1      1      1      2      1.      0.  0.
CBAR      2      1      3      4      1.      0.  0.
CBAR      3      2      2      3      0.      1.  0.
PBAR      1      1      0.00453  2.51E-5  0.      0.  0.
PBAR      2      1      0.0143  3.309E-4  0.      0.  0.
MAT1      1      2.1E+11
FORCE     1      2      0      1.  1000.      0.  0.
PLOAD1   1      3      FYE      FR      0. -75000.  1. -75000.
SPC       1      1      123456      0.
SPC       1      4      123456      0.
ENDDATA
    
```

SID and EID are load set and element identification numbers, respectively; TYPE is the type of distributed load (FX, FY, FZ, FXE, FYE, FZE, MX, MY, MZ, MXE, MYE, MZE); SCALE is the scale factor; X1 and X2 are locations starting from the first node; and P1 and P2 are load factors at location X1 and X2, respectively. If SCALE = FR (fractional), the Xi values are ratios of the distance along the axis to the total length, and (if X2 ≠ X1) Pi are load intensities per unit length of the element.

The analysis procedure is identical to the previous examples. In the Output window, we can find the bending moment, shear force, and axial force of each element, as shown in Table B.4.

Table B.4 Bending Moment, Shear Force, and Axial Force of the Portal Frame

Element	Node	Bending moment	Shear force	Axial force
1	1	-1.980678E+04	-1.512745E+04	-2.246731E+05
	2	4.070301E+04	-1.512745E+04	-2.246731E+05
2	3	-4.266460E+04	-1.612745E+04	-2.253269E+05
	4	2.184518E+04	-1.612745E+04	-2.253269E+05
3	2	-4.070301E+04	-2.246731E+05	-1.612745E+04
	3	-4.266460E+04	-2.246731E+05	-1.612745E+04