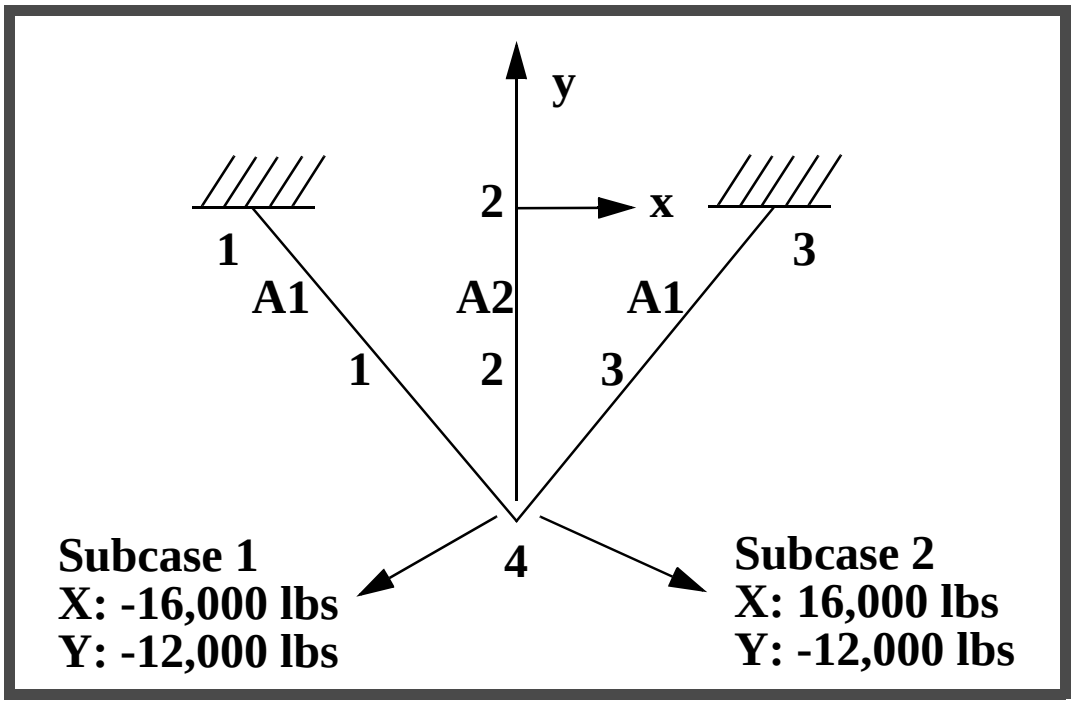
WORKSHOP 1



Objectives:

- Optimize the following three-bar truss problem subject to static loading.

Model Description:

- Objective Function: Weight minimization
- Design Variables: Cross-sectional area A1 and A2
- Constraints:

Stress Allowable:	20 ksi tension
	15 ksi compression
Displacements at grid 4:	X direction $\leq \pm 0.2$ in
	Y direction $\leq \pm 0.2$ in

Figure 1.1 - Geometry and Finite Elements

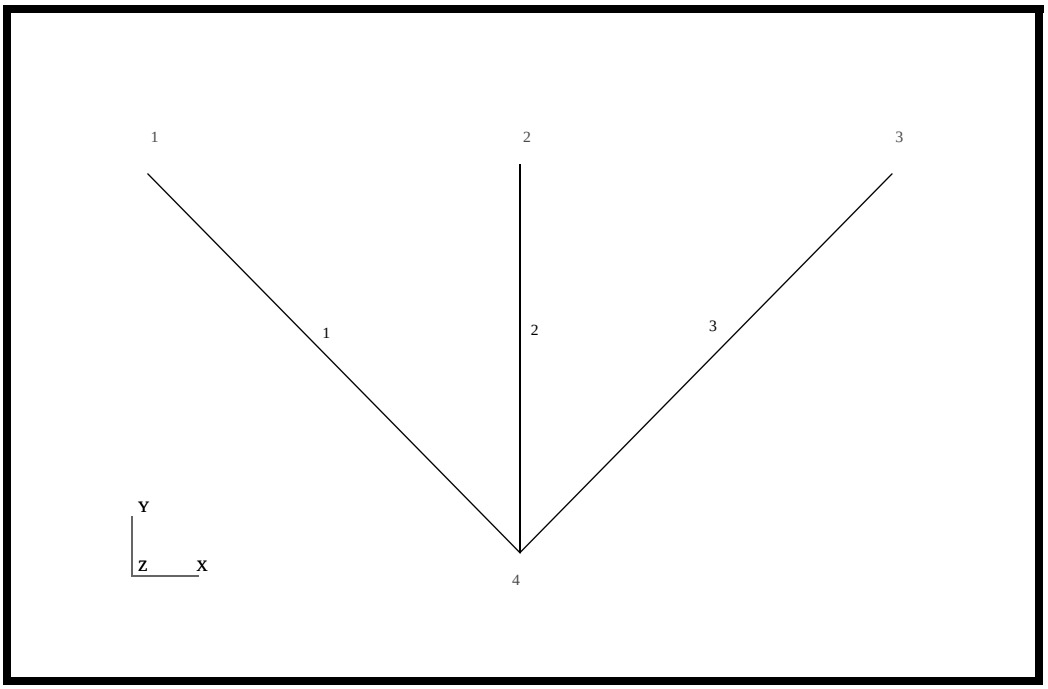


Figure 1.2 - Constraints and Applied Forces (Case 1)

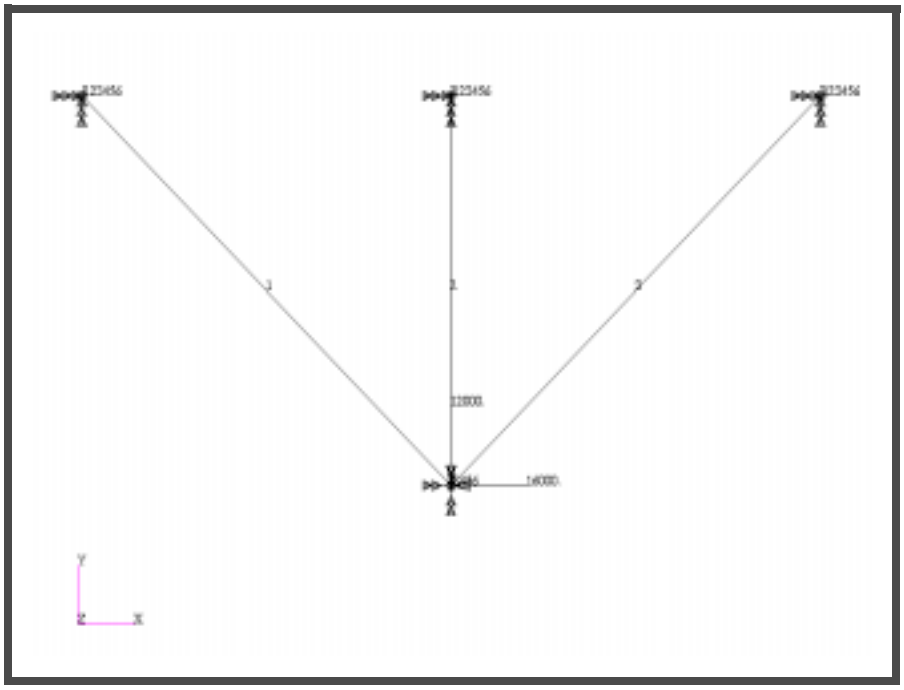


Figure 1.3 - Constraints and Applied Forces (Case 2)

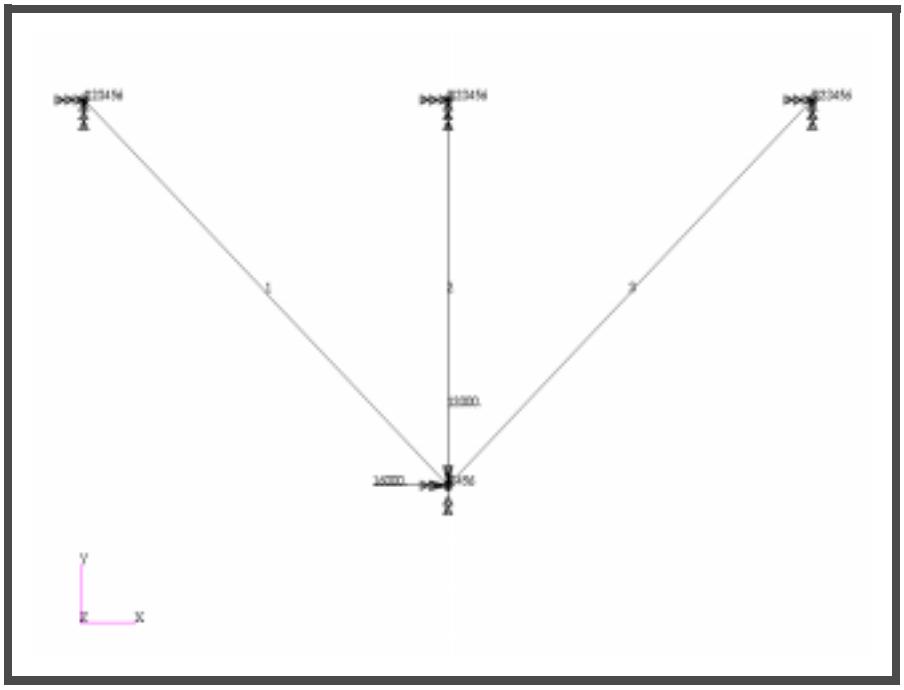


Table 1.1 - Material Properties.

Elastic Modulus	**10e6 psi**
Poisson's Ratio	**0.3**
Density	**0.101**

Exercise Procedure:

1. Users who are not utilitizing MSC.Patran for generating an input file should go to Step 15, otherwise, proceed to step 2.

2. Create a new database called **wkshp1.db**.

File/New...

New Database Name: **wkshp1**

OK

In the **New Model Preferences** form set the following:

Tolerance: ● **Default**

Analysis Code: **MSC/NASTRAN**

Analysis Type: **Structural**

OK

3. Activate the entity labels by selecting the **Show Labels** button on the toolbar.

Show Labels

4. Change to a front view by selecting the **Front View** button on the toolbar.

Front View

Whenever possible click ❑ **Auto Execute** (turn off).

5. Create nodes.

◆ **Finite Elements**

Action: **Create**

Object: **Node**

Method: **Edit**

❑ **Associate with Geometry**

Node Location List: **[-10 0 0]**

Apply

Node Location List: **[0 0 0]**

Apply

Node Location List: **[10 0 0]**

Apply

Node Location List: **[0 -10 0]**

Apply

6. Create bars.

◆ **Finite Elements**

Action: **Create**

Object: **Element**

Method: **Edit**

Shape: **Bar**

Node 1 = **Node 1**

Node 2 = **Node 4**

Apply

Node 1 = **Node 2**

Node 2 = **Node 4**

Apply

Node 1 = **Node 3**

Node 2 = **Node 4**

Apply

7. Next, define a material using the specified modulus of elasticity and allowable stresses.

◆ **Materials**

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name: **alum**

Input Properties...

Elastic Modulus = **10e6**

Poisson Ratio= **0.3**

Density= **0.101**

OK

Apply

8. Next, reference the material that was created in the previous step.

◆ **Properties**

Action: **Create**

Dimensior: **1D**

Type: **Rod**

Property Set Name: **prop_1**

Input Properties...

Material Name: **m:alum**

Area: **1**

OK

Pick **Select Members** and then click on the **Beam element** icon from the small menu window.

Beam element

Select Members: **Elm 1 3**

Add

Apply

Property Set Name: **prop_2**

Input Properties...

Area: **2**

OK

Select Members: **Elm 2**

Add

Apply

9. Create nodal constraints.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **disp_1**

Input Data...

Translations < T1 T2 T3 >: **< 0, 0, 0 >**

Rotations < R1 R2 R3 >: **< 0, 0, 0 >**

OK

Select Application Region...

Geometry Filter: ● **FEM**

Select Nodes: **Node 1:3**

Add

OK

Apply

New Set Name: **disp_2**

Input Data...

Translations < T1 T2 T3 >: **< , , 0 >**

Rotations < R1 R2 R3 >: **< 0, 0, 0 >**

OK

Select Application Region...

Geometry Filter: ● **FEM**

Select Nodes: **Node 4**

Add

OK

Apply

10. Apply forces on X and Y.

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Type: **Nodal**

New Set Name: **force_1**

Input Data...

Force < F1 F2 F3 >: **< -16000, -12000, >**

OK

Select Application Region...

Geometry Filter: ● **FEM**

Select Nodes: **Node 4**

Add

OK

Apply

New Set Name: **force_2**

Input Data...

Force < F1 F2 F3 >: **< 16000, -12000, >**

OK

Select Application Region...

Geometry Filter: ● **FEM**

Select Nodes: **Node 4**

Add

OK

Apply

11. Create Load Cases.

◆ **Load Cases**

Action: **Create**

Load Case Name: **case_1**

Assign/Prioritize Loads/BCs

Select Individual Loads/BCs: **Displ_disp_1**

Displ_disp_2

Force_force_1

OK

Apply

Load Case Name: **case_2**

Assign/Prioritize Loads/BCs

Select Individual Loads/BCs: **Displ_disp_1**

Displ_disp_2

Force_force_2

OK

Apply

12. Using **Tools** for **Model Variables**.

Tools/Model Variables...

Action: **Create**

Object: **Variable**

Method: **Property**

Dimension: **1D**

Type: **Rod**

Select Property Set: **prop_1**

Select Property Name: **Area**

Apply

Select Property Set: **prop_2**

Select Property Name: **Area**

Apply

Close

13. Using **Tools** for **Design Study**.

Tools/Design Study...

Action: **Create**

Object: **Design Study**

Design Study Name: **opt_1**

Design Variables...

	Lower Bound	*Upper Bound*
prop_1_Area:	**0.1**	**100**
prop_2_Area:	**0.1**	**100**

OK

Design Objective...

Existing Objectives: **Total_Weight**

Min/Max: **minimize**

OK

Design Constraints...

Action: **Create**

Solution: **Linear Static**

Response: **Displacement**

Constraint Name: **DISP_1**

Select Node: **Node 4**

Displacement Component: ● **TX**

Lower Bound: **-0.2**

Upper Bound: **0.2**

Apply

Constraint Name: **DISP_2**

Select Node: **Node 4**

Displacement Component: ● **TY**

Lower Bound: **-0.2**

Upper Bound: **0.2**

Apply

Response: **Stress**

Constraint Name: **STRESS_1**

Constraint Region: ● **FEM**

1D

Rod

Select Finite Element: **Element 1:3**

Stress Component: **Axial**

Lower Bound: **-15000**

Upper Bound: **20000**

Apply

Close

Apply

Close

Now you are ready to generate an input file for analysis.

14. Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown here.

◆ **Analysis**

Action: **Optimize**

Object: **Entire Model**

Method: **Analysis Deck**

Job Name: **wkshp1**

Translation Parameters...

Data Output: **OP2 and Print**

MSC/NASTRAN Version: **70.5**

OK

Optimization Parameters...

Maximum Number of Design cycles (DESMAX) = **10**

Print Design Data (P1) every n-th cycle where n = **1**

Print Analysis Results (NASPRT) every n-th cycle where n = **1**

OK

Subcase Select...

Solution Type: **101 LINEAR STATIC**

Subcases Available: **case_1**

case_2

OK

Apply

An MSC.Nastran input file called **wkshp1.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC.Nastran users should proceed to **Step 15**.

Generating an input file for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table. The result should be similar to the output below.

15. MSC.Nastran input file: **wkshp1.dat**

```
SOL 200
TIME 600
CEND
TITLE =
ECHO = NONE
MAXLINES = 999999999
DESOBJ(MIN) =  1
ANALYSIS = STATICS
SUBCASE 1
     SUBTITLE=case_1
     SPC = 2
     LOAD = 2
     DISPLACEMENT(SORT1,REAL)=ALL
     SPCFORCES(SORT1,REAL)=ALL
     STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
     DESSUB =  1
SUBCASE 2
     SUBTITLE=case_2
     SPC = 2
     LOAD = 4
     DISPLACEMENT(SORT1,REAL)=ALL
     SPCFORCES(SORT1,REAL)=ALL
     STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
     DESSUB =  1
BEGIN BULK
PARAM       POST                        -1
PARAM       PATVER                       3.
PARAM       AUTOSPC                     YES
PARAM       COUPMASS                    -1
PARAM       K6ROT                        0.
PARAM       WTMASS                       1.
PARAM,NOCOMPS,-1
PARAM       PRTMAXIM                    YES
PARAM       NASPRT                       0
PROD          1         1         1.
CROD          1         1         1         4
CROD          3         1         3         4
PROD          2         1         2.
CROD          2         2         2         4
MAT1          1       1.+7                .3        .101
GRID          1                  -10.      0.        0.
GRID          2                    0.      0.        0.
GRID          3                   10.      0.        0.
GRID          4                    0.     -10.       0.
SPCADD        2         4         6
LOAD          2        1.        1.         1
LOAD          4        1.        1.         3
SPC1          4    123456        1         2         3
SPC1          6      3456        4
```

```
FORCE          1          4          0          20000.     -.8        -.6        0.
FORCE          3          4          0          20000.      .8        -.6        0.
DESVAR         1          prop_1:1   1.         .1         100.       1.
DESVAR         2          prop_2:2   2.         .1         100.       1.
DVPREL1        1          PROD       1          4                                        +     A
+              A          1          1.
DVPREL1        2          PROD       2          4                                        +     B
+              B          2          1.
DRESP1         1          W          WEIGHT
DRESP1         2          DIS2       DISP                                1          4
DRESP1         3          DIS3       DISP                                2          4
DRESP1         4          STR4       STRESS     ELEM                     2          1         +     C
+              C          2          3
DCONSTR        1          2          -.2        .2
DCONSTR        1          3          -.2        .2
DCONSTR        1          4          -15000.    20000.
DOPTPRM DESMAX 5            P1         0          P2         1          CONV1      .001      +     D
*      D CONV2             1.-20                 CONVDV                .001                           *     E
*      E CONVPR            .01                   DELP                  .2                             *     F
+      F DELX              1.                    DPMIN                 .01        DXMIN      .05
ENDDATA
```

SUBMITTING THE INPUT FILE FOR MSC.Nastran and MSC.Patran USERS:

16. Submit the input file to MSC.Nastran for analysis.

    16a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter: **nastran wkshp1.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    16b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter: **nastran wkshp1 scr=yes**. Monitor the run using the UNIX **ps** command.

17. When the run is completed, edit the **wkshp1.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

Comparison of Results:

18. Compare the results obtained in the **.f06** file with the following:

****

S U M M A R Y   O F   D E S I G N   C Y C L E   H I S T O R Y

****

NUMBER OF FINITE ELEMENT ANALYSES COMPLETED 10
NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS 9

OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY

CYCLE NUMBER	OBJECTIVE FROM APPROXIMATE OPTIMIZATION	OBJECTIVE FROM EXACT ANALYSIS	FRACTIONAL ERROR OF APPROXIMATION	MAXIMUM VALUE OF CONSTRAINT
INITIAL		4.876711E+00		-3.234952E-01
1	3.900626E+00	3.901369E+00	-1.906068E-04	-1.543690E-01
2	3.237960E+00	3.237859E+00	3.122109E-05	1.123340E-03
3	3.051926E+00	3.052044E+00	-3.851201E-05	-2.282910E-03
4	2.919098E+00	2.919285E+00	-6.386610E-05	-3.163770E-03
5	2.831564E+00	2.831548E+00	5.894056E-06	-3.612891E-03
6	2.772826E+00	2.772921E+00	-3.439241E-05	-9.210937E-04
7	2.750082E+00	2.750096E+00	-4.941595E-06	-3.640527E-03
8	2.734132E+00	2.734130E+00	7.848081E-07	-1.868945E-03
9	2.7321978+00	2.731945E+00	1.213062E-05	-1.724023E-03

DESIGN VARIABLE HISTORY

INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	INITIAL	1	2	3	4	5
1	1	PROP_1:1	1.0000E+00	8.0000E-01	6.8089E-01	7.0642E-01	7.3235E-01	7.5955E-01
2	2	PROP_2:2	2.0000E+00	1.6000E+00	1.2800E+00	1.0238E+00	8.1898E-01	6.5519E-01

INTERNAL DV. ID.	EXTERNAL DV. ID.	LABEL	6	7	8	9	10	11
1	1	PROP_1:1	7.8303E-01	8.1287E-01	8.3422E-01	8.4446E-01		
2	2	PROP_2:2	5.3072E-01	4.2373E-01	3.4752E-01	3.1639E-01		

19. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

20. Proceed with the Reverse Translation process, that is, importing the **prob1.op2** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

Action: **Read Output 2**

Object: **Result Entities**

Method: **Translate**

Select Results File...

Selected Results File: **prob1.op2**

OK

Apply

21. Plot **Select Result Graph**.

◆ **XY Plot**

Action: **Post**

Object: **XY Window**

Select Current XY Window: **DesignVariableHistory**

Post/Unpost XY Windows: **DesignVariableHistory**

Apply

Object: **Curve**

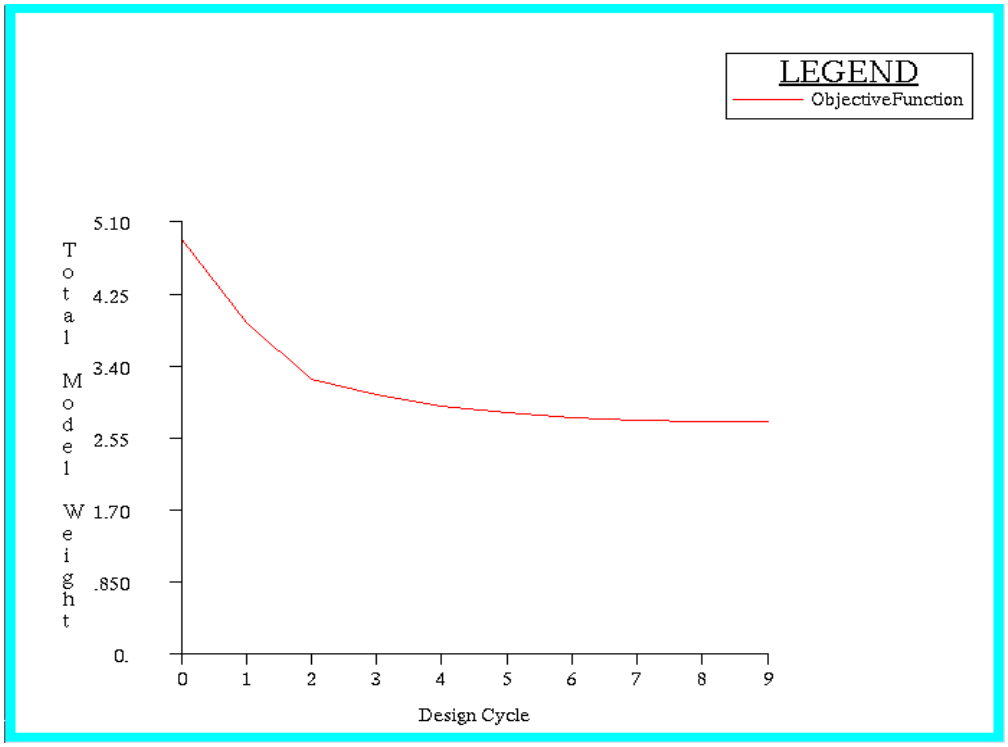
Post/Unpost Curves: **Objective Function**

Apply

Action: **Modify**

Object:	**Axis**
Active Axis:	● **Y**
Title...	
Active Axis:	■ **Display Axis Title**
Axis Title:	**Total Model Weight**
Font Size:	**12**
Apply	
Cancel	

Figure 2.1 - Objective Function



Action: **Post**

Object: **Curve**

Post/Unpost Curves: **PROP_1:1**
PROP_2:2

Apply

Action: **Modify**

Object: **Axis**

Active Axis: ● **Y**

Title...

Active Axis: ■ **Display Axis Title**

Axis Title: **Area**

Font Size: **12**

Apply

Cancel

Action: **Modify**

Object: **Curve**

Curve List: **PROP_1:1**

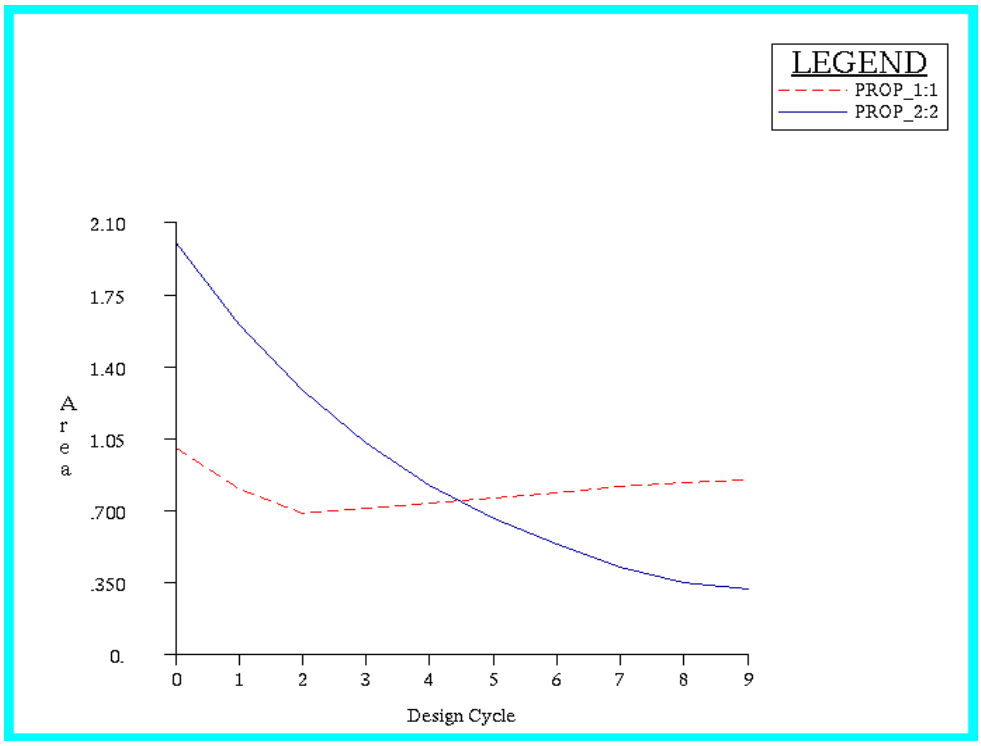
Options...

Line Style: **LongDash**

Apply

Cancel

Figure 2.2 - Design Variable



Action: **Post**

Object: **Curve**

Post/Unpost Curves: **Maximum Constraint**

Apply

Action: **Modify**

Object: **Axis**

Active Axis: ● **Y**

Title...

Active Axis: ■ **Display Axis Title**

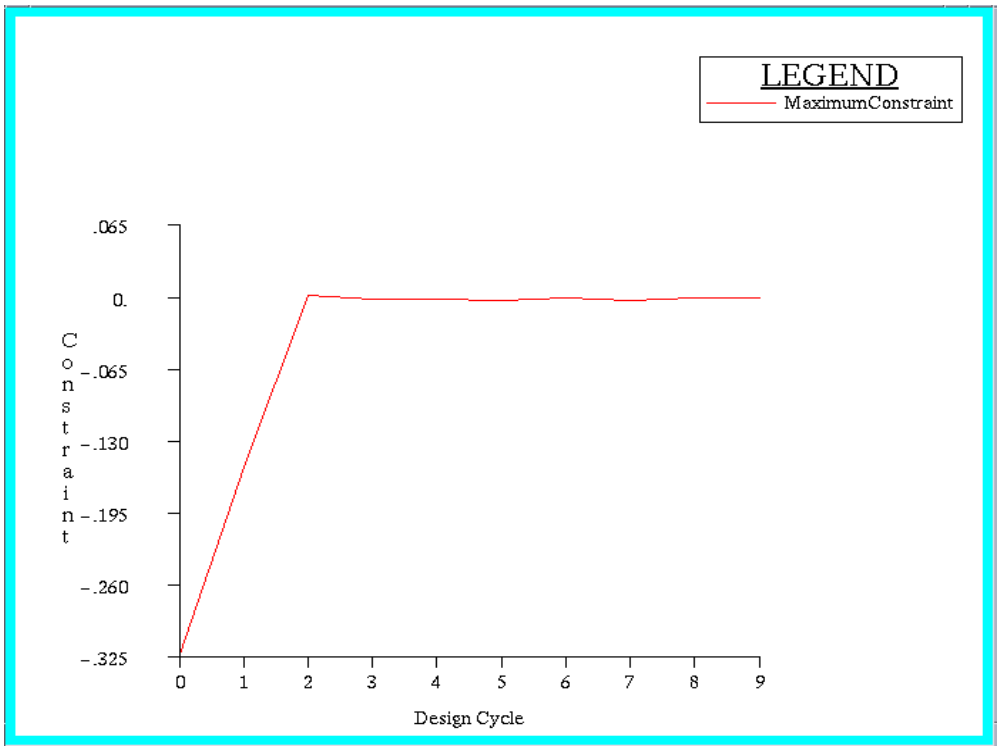
Axis Title: **Constraint**

Font Size: **12**

Apply

Cancel

Figure 2.3 - Maximum Constraint



Quit MSC.Patran when you have completed this exercise.