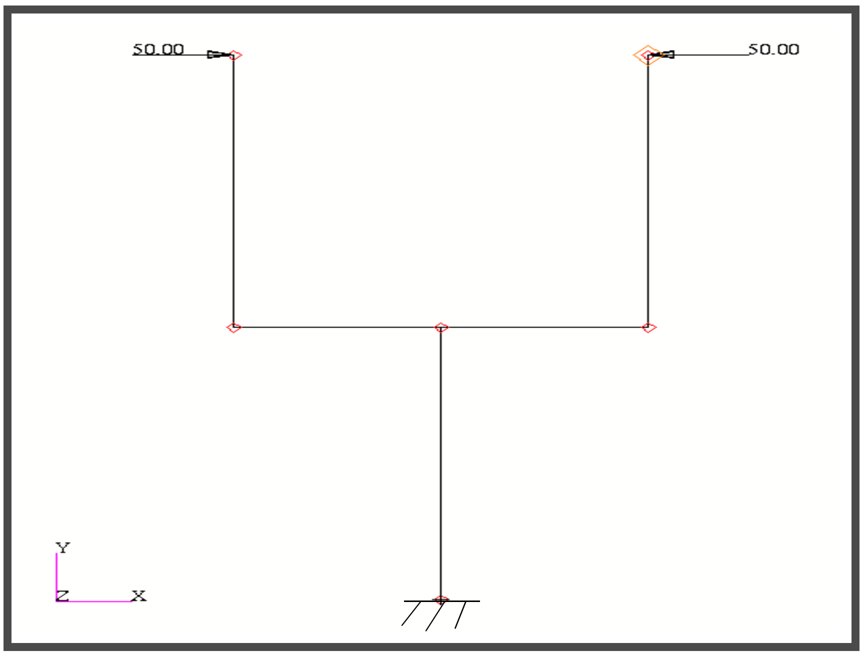
WORKSHOP 6a

Multipoint Constraints Relative Motion



Objectives

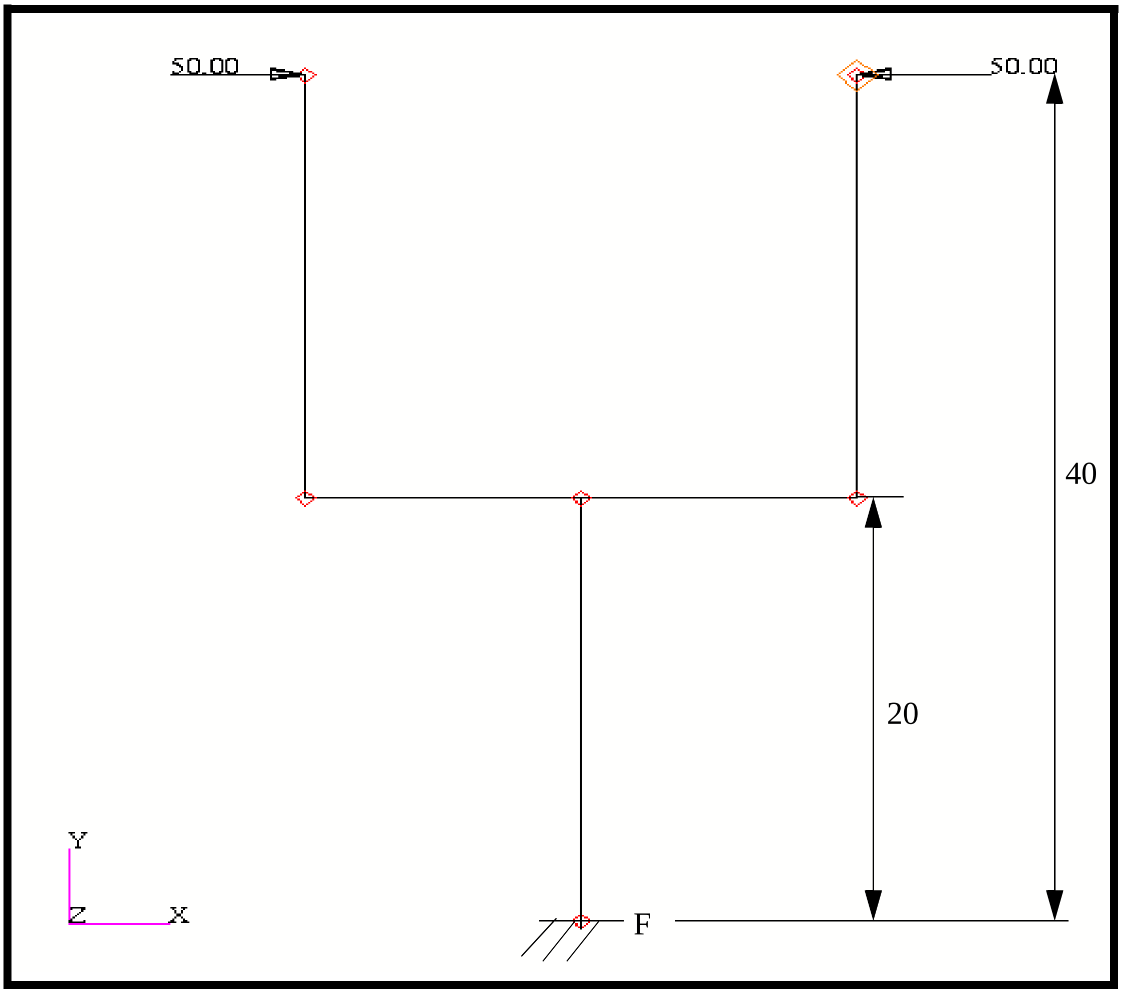
- Define time-varying excitation.
- Create a MSC.Nastran dynamic math model.
- Submit the file for analysis in MSC.Nastran.
- Compute nodal displacements for desired time domain.

Model Description:

Using the direct method, determine the transient response of the structure, under time-varying excitation. This example structure shall be excited by a 50 lb force applied at the left. Additionally, a negative 50lbs force is applied at the right also varying at 250Hz. Both time dependent dynamic loads are applied for the duration of 0.008 seconds only. Use structural damping of g=0.06 and convert this damping to equivalent viscous damping at 250Hz. Carry the analysis for 0.04 seconds.

Below is a finite element representation of the structure. It also contains the loads and boundary conditions.

Figure 6a.1-Loads and Boundary Conditions



Suggested Exercise Steps

- Define the time-varying tip loads (FORCE, LSEQ and TLOAD2).
- Specify integration time steps (TSTEP).
- Prepare the model for a direct transient analysis (SOL 109).
- Specify the structural damping and convert this damping to equivalent viscous damping.
  - PARAM, G, 0.06
  - PARAM, W3, 1571.0
- Request response in terms of nodal displacement at grid points 5 and 6.
- Generate an input file and submit it to the MSC.Nastran solver for direct transient analysis.
- Review the results, specifically the nodal displacements and xy-plot output.

Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0 and begin to create a new model.

   Double click on the icon labeled **MSC.Nastran for Windows V4.0**.

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

   Open Model File: **New Model**

2. Create the material for the model.

   Model/Material...

   ID: **1**

   Title: **mat_1**

   Youngs Modulus, E: **29E6**

   Poisson Ratio: **0.3**

   Mass Density: **0.286**

   OK

   Cancel

3. Create the elements properties for the model.

   Model/Property...

   Element/Property Type...

   Line Elements: ● **Beam**

   OK

   ID: **1**

   Title: **Prop_1**

   Material: **1..mat_1**

   Area: **0.1**

   Moment of Inertia, I1 or Izz: **1**

   I2 or Iyy: **1**

   Torsional Constant, J: **2**

OK

Cancel

4. Turn off the workplane and turn on the Node ID labels.

   First, turn off the workplane.

   Tools/Workplane...

   (uncheck box) ☐ **Draw Workplane**

   Done

   View/Regenerate <Ctrl+G>

   Turn on the surface ID labels.

   View/Options...

   Category: ● **Labels, Entities, and Color**

   Options **Node**

   Label Mode: **1..ID**

   Apply

   OK

5. Create six nodes.

   Model/Node...

   Locate -- Enter Coordinates or select with cursor dialog window:

   ID: **1**

Base:	*X:*	*Y:*	*Z:*
	0	**0**	**0**

   OK

   OK

Repeat the above operation to create the other 5 nodes.

ID	X	Y	Z	
2	-10	20	0	**OK**
3	0	20	0	OK
4	10	20	0	OK
5	-10	40	0	OK
6	10	40	0	OK

Cancel

<Ctrl + A>

6. Create the elements.

Model/Element...

ID: **1**

Property: **1..Prop_1**

Nodes: **1** **3**

Under Orientation, click Vector. You will specify the orientation of the beam element (this will apply for all other beam elements you will create).

Orientation: **Vector**

Vector Locate - Define Element Orientation Vector dialog window:

	X:	Y:	Z:
Base:	**0**	**0**	**0**
Tip:	**0**	**0**	**1**

OK

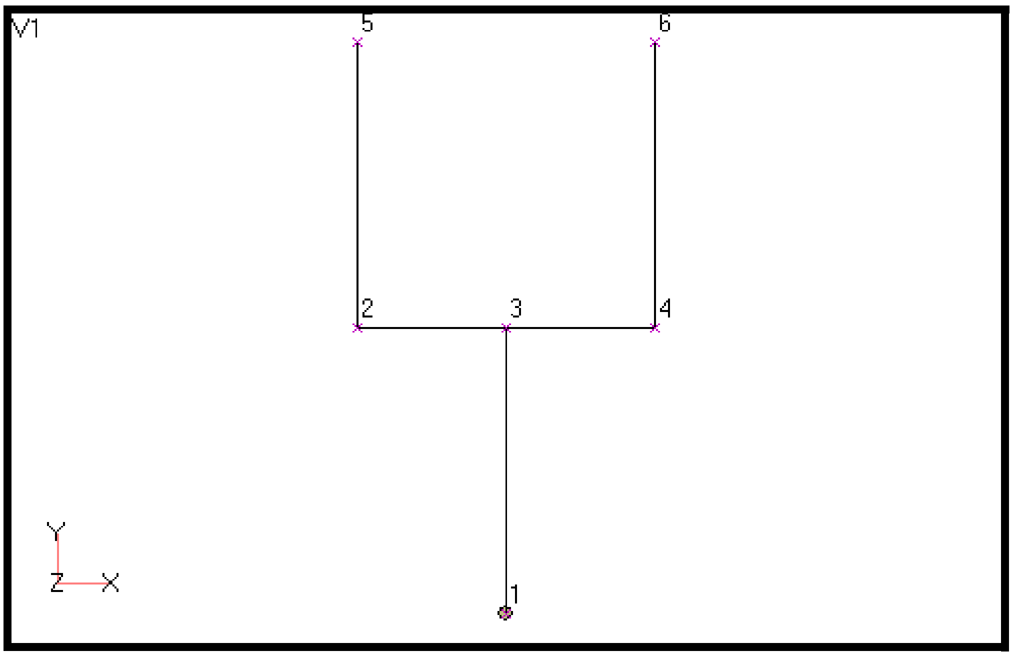
Create the other elements by nodes. Use the table below for reference. You need not re-specify the orientation vector when creating the remaining elements.

ID	Nodes		
2	2	3	**OK**
3	3	4	**OK**
4	2	5	**OK**
5	4	6	**OK**

Cancel

The model should appear as follows:

Figure 6a.2



7. Create a time-dependent function for the transient response for the nodal force.

Model/Function...

ID: **1**

Title: **time_depedent_force**

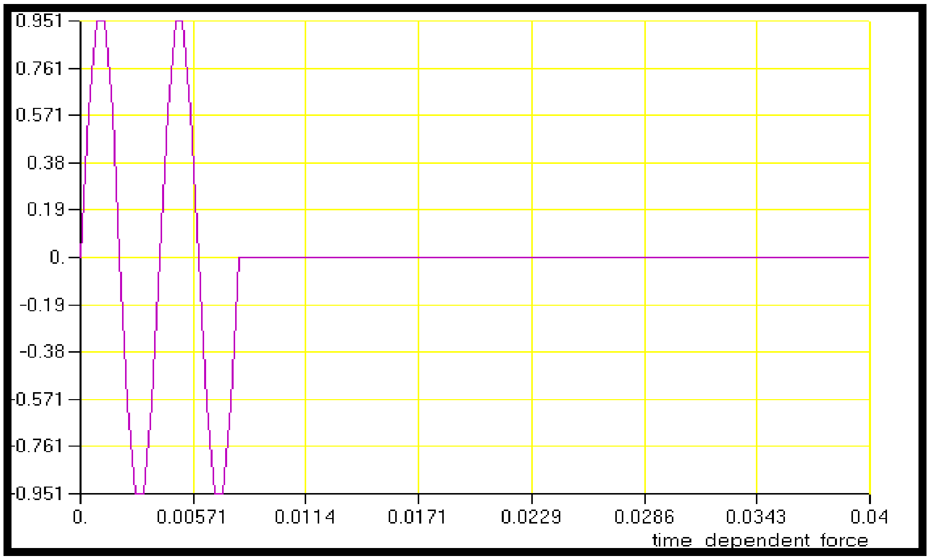
Type:	**1..vs. Time**
Data Entry:	● **Equation**
Delta X:	**0.0004**
X Variable:	**t**
X:	**0.0**
To X:	**0.008**
Y:	**sin(250*360*!t)**
More	
Data Entry:	● **Single Value**
X:	**0.008**
Y:	**0.0**
More	
X:	**0.04**
Y:	**0.0**
OK	
Cancel	

8. Plot the input time function.

View/Select...	
XY Style:	● **XY of Function**
Model Data...	
function	**1..time_dependent_force**
OK	
OK	

The plot should resemble the following figure.

Figure 6a.3



Turn off the plot.

View/Select...

● **Draw Model**

OK

9. Define a dynamic load case.

Model/Load/Set...

ID:	**1**
Title:	**force**

OK

Model/Load/Nodal...

<Select Node 5>

OK

On the Create Loads on Surfaces form, <highlight force>

under Select Force

FX	☒	**50**
FY	☒	**0**
FZ	☒	**0**

Function Dependence **1..time_dependent_force**

OK

<Select Node 6>

OK

On the Create Loads on Surfaces form, <highlight force>

under Select Force

FX	☒	**-50**
FY	☒	**0**
FZ	☒	**0**

Function Dependence **1..time_dependent_force**

OK

Cancel

10. Create the constraint.

Model/Constraint/Set...

ID: **1**

Title: **constraint**

OK

Model/Constraint/Nodal...

<select Node 1>

OK

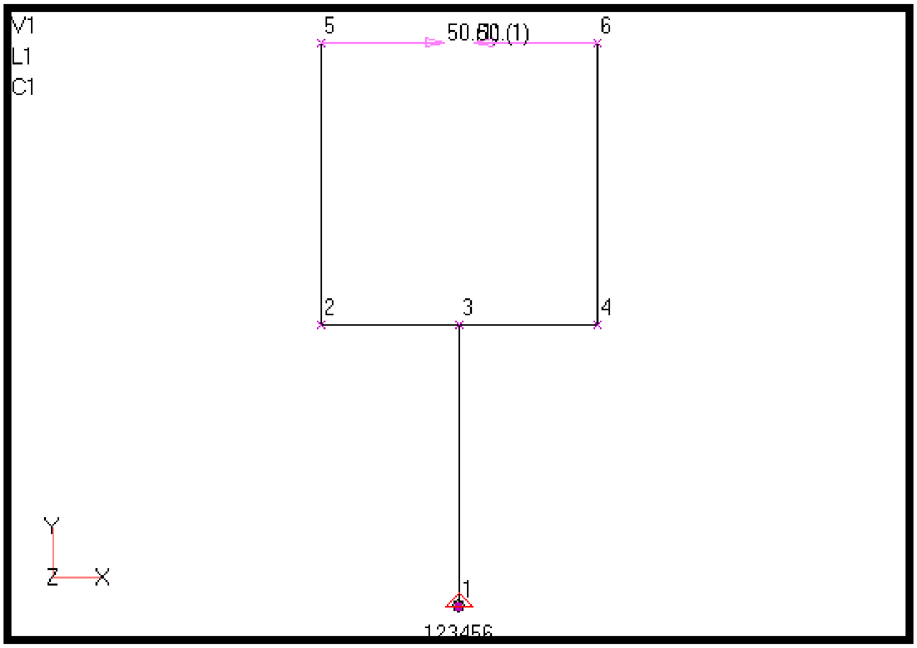
Fixed

OK

Cancel

Your display should resemble the following:

Figure 6a.4



11. Set the dynamic analysis load set options.

Model/Load/Dynamic Analysis...

Solution Method: ● **Direct Transient**

Under Equivalent Viscous Damping,

Overall Structural Damping Coeff (G): **0.06**

Under Equivalent Viscous Damping Conversion,

Frequency for System Damping (W3-Hz): **250**

Under Transient Time Step Intervals,

Number of Steps: **100**

Time per Step: **0.0004**

Advanced...

Mass Formulation: ● **Coupled**

OK

OK

12. Write the input deck and run the analysis.

File/Export/Analysis Model...

Analysis Format/Type: **3..Transient Dynamic/Time History**

OK

Filename: **mpc1**

Write

Additional Info: ☒ **Run Analysis**

Advanced...

Solution Type: ● **Direct**

OK

OK

OK

Under PARAM,

☒ **WTMASS** **0.00259**

OK

When asked “OK to Save Model Now?”

Yes

When the MSC.Nastran manager is through running, MSC.Nastran 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 successfully, we will not bother with the details this time.

Continue

13. Open the mpc1.dat using Notepad. Your file should look similar to the one on the following page.

```
INIT MASTER(S)
ID D:\Scrat,MSC/N
SOL SEDTRAN
TIME 10000
CEND
  ECHO = NONE
  DISPLACEMENT(SORT1,PLOT) = ALL
  OLOAD(SORT1,PLOT) = ALL
  SPCFORCE(SORT1,PLOT) = ALL
  SPC  = 1
  DLOAD = 1
  LOADSET = 1
  TSTEP = 1
BEGIN BULK
$
***********************************************************
*****************
$   Written by : MSC/NASTRAN for Windows
$   Version    : 6.00
$   Translator : MSC/NASTRAN
$   From Model : D:\Scratch\ws6b.MOD
$   Date       : Mon Nov 22 20:45:27 1999
$   Output To  : D:\scratch\mpc1
$
***********************************************************
*****************
$
PARAM,POST,-1
PARAM,OGEOM,NO
PARAM,AUTOSPC,YES
PARAM,MAXRATIO,1.E+8
PARAM,GRDPNT,0
PARAM,WTMASS,0.00259
CORD2C         1       0      0.      0.      0.      0.      0.      1.+MSC/NC1
+MSC/NC1      1.      0.      1.
CORD2S         2       0      0.      0.      0.      0.      0.      1.+MSC/NC2
+MSC/NC2      1.      0.      1.
$ MSC/NASTRAN for Windows Load Set 1 : force
PARAM,COUPMASS,1
PARAM,G,0.06
PARAM,W3,1570.8
PARAM,CURVPLOT,1
LSEQ           1     101     101
$ MSC/NASTRAN for Windows Function 1 : time_dependent_force
TABLED2        1      0.                                                +
+             0.      0.   4.E-4 0.58779   8.E-4 0.95106  0.0012 0.95106+
+        0.0016 0.58779   0.002      0.  0.0024-0.58779  0.0028-0.95106+
+      0.0032-0.95106 0.0036-0.58779  0.0041.53E-15 0.0044 0.58779+
+       0.0048 0.95106 0.0052 0.95106 0.0056 0.58779  0.006-1.4E-15+
+               0.0064-0.58779    0.0068-0.95106    0.0072-0.95106    0.0076-
0.58779+
```

```
+         0.008     0.    0.04     0.ENDT
TLOAD1      101     101                  1
FORCE       101       5       0      1.    50.      0.      0.
LSEQ          1     102     102
TLOAD1      102     102                  1
FORCE       102       6       0      1.   -50.      0.      0.
DLOAD         1      1.      1.     101     1.     102
TSTEP         1     100   4.E-4
$ MSC/NASTRAN for Windows Constraint Set 1 : constraint
SPC           1       1  123456      0.
$ MSC/NASTRAN for Windows Property 1 : Prop_1
PBEAM         1       1     0.1      1.      1.      0.      2.      0.+PR    1
+PR    1     0.      0.      0.      0.      0.      0.      0.      0.+PA    1
+PA    1   YESA      1.                                              +PC    1
+PC    1     0.      0.
$ MSC/NASTRAN for Windows Material 1 : mat_1
MAT1          1  2.9E+7             0.3   0.286      0.      0.
GRID          1       0      0.      0.      0.       0
GRID          2       0    -10.     20.      0.       0
GRID          3       0      0.     20.      0.       0
GRID          4       0     10.     20.      0.       0
GRID          5       0    -10.     40.      0.       0
GRID          6       0     10.     40.      0.       0
CBEAM         1       1       2       3      0.      0.      1.
CBEAM         2       1       3       4      0.      0.      1.
CBEAM         3       1       1       3      0.      0.      1.
CBEAM         4       1       2       5      0.      0.      1.
CBEAM         5       1       4       6      0.      0.      1.
ENDDATA 6336c227
```

14. Plot the x-component of the displacement response at Node 5 and Node 6.

View/Select...

XY Style: ● **XY vs. Set**

XY Data...

First, plot the displacement response of Node 5 as Curve 1.

Curve: ● **1**

Output Set: **1..Case 1 Time 0**

Output Vector: **2..T1 Translation**

Output Location: **Node** **5**

Then, plot the displacement response of Node 6 as Curve 2.

Curve: ● **2**

Output Set: **1..Case 1 Time 0**

Output Vector: **2..T1 Translation**

Output Location: **Node** **6**

OK

OK

Your plot should look similar to the following:

Figure 6a.5