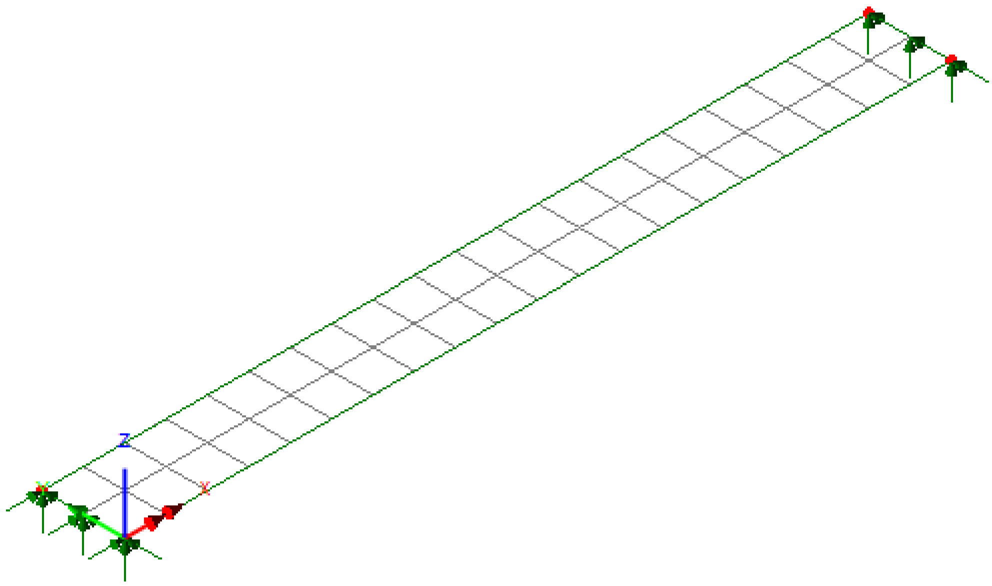
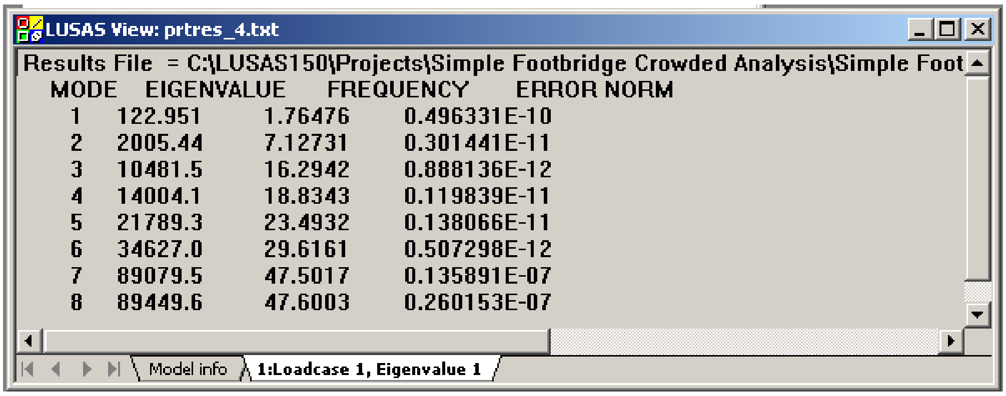
Performing Steady-State Pedestrian Crowd Loading Analyses for NA to BS EN 1991-2:2003 in LUSAS

NA.2.44.5 describes the procedure that needs to be followed to assess the maximum vertical accelerations that result from pedestrians in crowded conditions using a vertical pulsating distributed load approach.

In order to demonstrate the procedures for setting this analysis up in LUSAS a very simple footbridge will be used which consists simply of a steel deck of 300mm thickness with a width of 2m and a span of 20m meshed with linear thick shell elements and with simple supports as shown in the following figure. Name the file "Simple Footbridge Crowded Analysis".



If we perform an eigenvalue analysis over the frequency range of 0 to 50Hz of this structure we find the following frequency and mass participation behaviour (using **Utilities>Print Results Wizard**):


LUSAS View: prtres_4.txt

Results File = C:\LUSAS150\Projects\Simple Footbridge Crowded Analysis\Simple Foot

MODE	EIGENVALUE	FREQUENCY	ERROR NORM
1	122.951	1.76476	0.496331E-10
2	2005.44	7.12731	0.301441E-11
3	10481.5	16.2942	0.888136E-12
4	14004.1	18.8343	0.119839E-11
5	21789.3	23.4932	0.138066E-11
6	34627.0	29.6161	0.507298E-12
7	89079.5	47.5017	0.135891E-07
8	89449.6	47.6003	0.260153E-07

Model info | 1:Loadcase 1, Eigenvalue 1

LUSAS View: prtres_5.txt

Results File = C:\LUSAS150\Projects\Simple Footbridge Crowded Analysis\Simple Foot

MODE	MASS PF X	MASS PF Y	MASS PF Z
1	0.293643E-34	0.262373E-31	0.810416
2	0.793687E-33	0.417140E-32	0.179334E-25
3	0.562312E-31	0.162901E-30	0.898654E-01
4	0.445703E-30	0.739803	0.330213E-30
5	0.953276E-31	0.237720E-28	0.686979E-26
6	0.771465E-30	0.752224E-29	0.245445E-27
7	0.135722E-20	0.217054E-20	0.283471E-19
8	0.104452E-19	0.166861E-19	0.321236E-01

From these we can see that there are two key modes of the structure that have approximately 80% of the structure mass participating in them for the mode 1 vertical (Z) and mode 4 lateral (Y) directions. These will therefore be used to define the frequency range for the Rayleigh damping parameters.

Using the Microsoft Excel spreadsheet downloadable from the User Area on the LUSAS Support Website (link) and assuming 3% damping we can calculate the Rayleigh damping parameters as indicated below.

Rayleigh Damping Coefficents

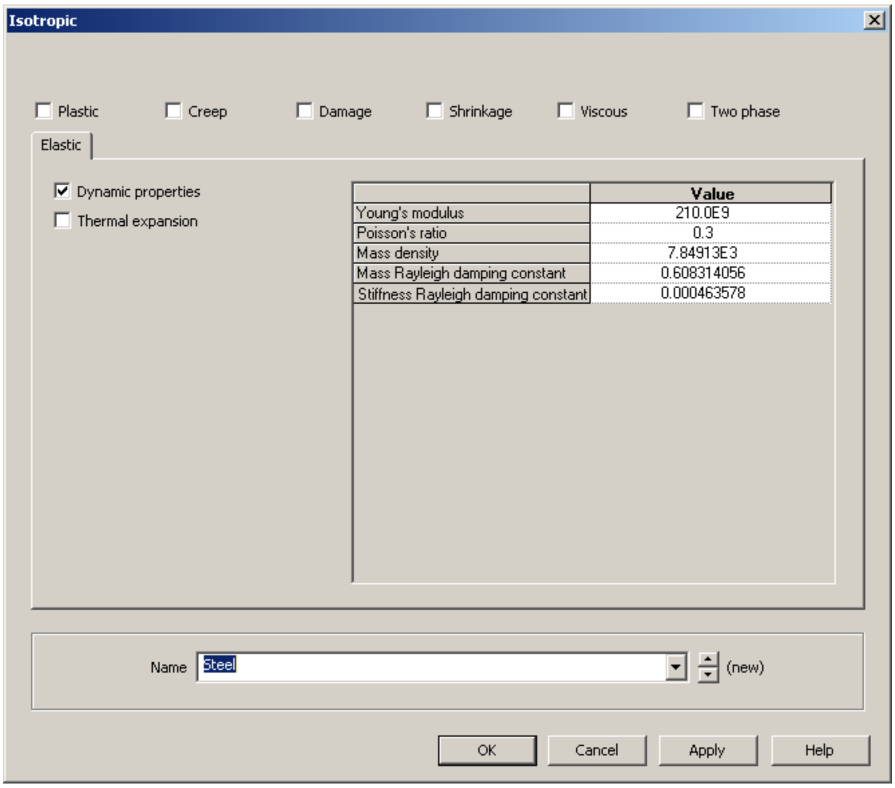
Usage:
a) Input the start and end frequencies of interest
b) Input the damping ratios required at these frequencies
c) Extract the Rayleigh damping coefficients and specify in the material properties of the material

	Frequency (Hz)*	Damping Ratio*	Damping (%)	Circular Frequency (/s)	Eigenvalue
1st	1.7648	0.03	3	11.09	122.96
2nd	18.8343	0.03	3	118.34	14,004.21

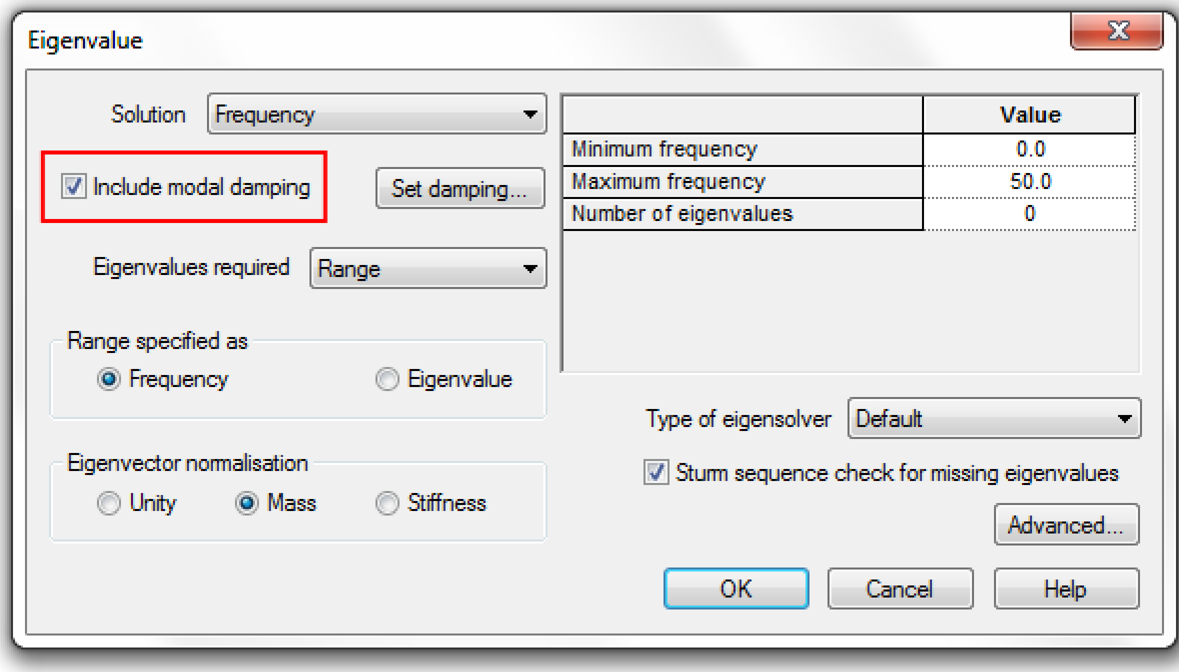
Rayleigh Damping Parameters:

Alpha:	0.608314056
Beta:	0.000463578

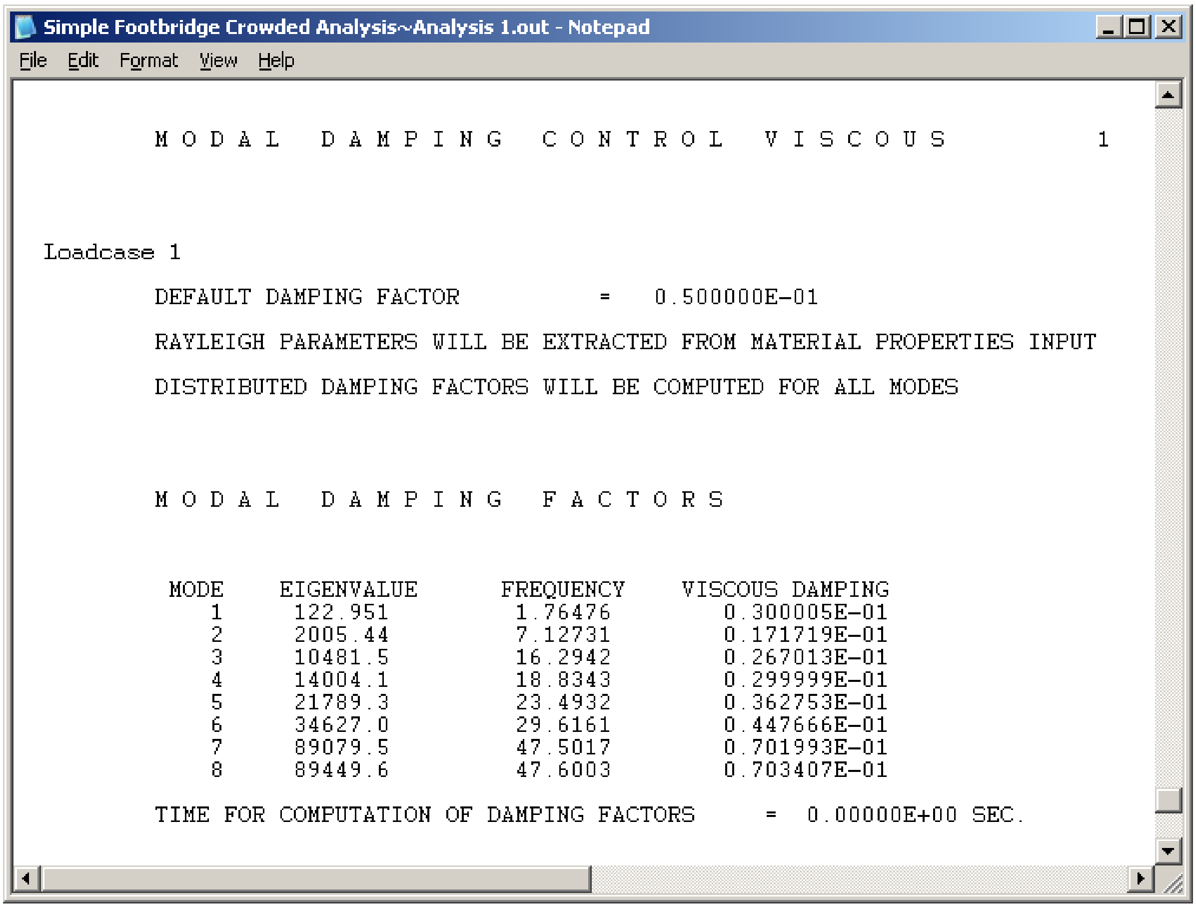
These can be added to the material properties of the model by right-clicking the material attribute and selecting 'edit attribute', before ticking the 'Dynamic properties' box in the elastic material properties tab. The Alpha and Beta Rayleigh damping parameters correspond to the Mass and Stiffness damping constants respectively.



To check that we have achieved the damping levels that we require we can reanalyse the eigenvalue analysis using this modified material property and enable the **Include modal damping** as indicated in the figure below (we are using the default settings to calculate all modes so we do not need to click on the **Set damping...** button).



If we now look at the analysis output file `*.out` we will see underneath the frequency and mass participation information a section called **Modal Damping Factors** which will tell us the distributed damping factors attained using the material Rayleigh damping parameters. Looking at modes 1 and 4 which were the limits of our range for tuning the damping parameters we see that we have successfully obtained 3% or (0.3E-1) at these target frequencies with the remaining modal damping values following the Rayleigh proportional damping behaviour (refer to the LUSAS Theory Manual for more information on Rayleigh damping).



M O D A L   D A M P I N G   C O N T R O L   V I S C O U S   1

Loadcase 1

DEFAULT DAMPING FACTOR = 0.500000E-01

RAYLEIGH PARAMETERS WILL BE EXTRACTED FROM MATERIAL PROPERTIES INPUT

DISTRIBUTED DAMPING FACTORS WILL BE COMPUTED FOR ALL MODES

M O D A L   D A M P I N G   F A C T O R S

MODE	EIGENVALUE	FREQUENCY	VISCOUS DAMPING
1	122.951	1.76476	0.300005E-01
2	2005.44	7.12731	0.171719E-01
3	10481.5	16.2942	0.267013E-01
4	14004.1	18.8343	0.299999E-01
5	21789.3	23.4932	0.362753E-01
6	34627.0	29.6161	0.447666E-01
7	89079.5	47.5017	0.701993E-01
8	89449.6	47.6003	0.703407E-01

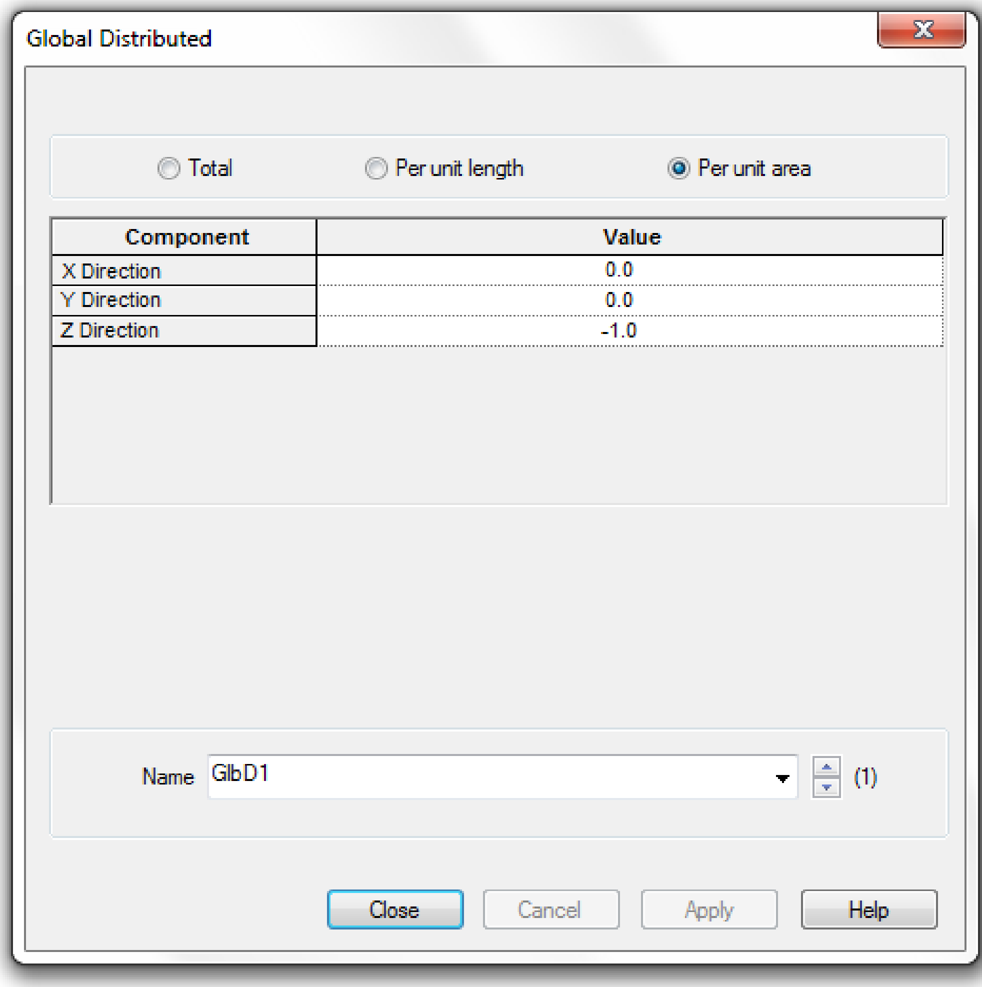
TIME FOR COMPUTATION OF DAMPING FACTORS = 0.00000E+00 SEC.

We now have our damping requirements for the structure.

The remaining tasks for the creation of the model are the definition and assignment of the pedestrian crowd loading and the analysis load curve and controls to carry out the solution. From the equation given in NA.2.44.5 we need to determine the parameters that define the amplitude of the vertical pulsating distributed load which are dependent on the frequency of the structure and other factors.

If we assume that we will need to look at more than one forcing frequency as, in accordance with NA.2.44.3, we may need to take account of modes other than the fundamental mode in order to calculate the maximum responses, we will set up the loading of the model as a unit distributed load that can be assigned to the necessary locations and conduct all of the scaling in the load curve. For this simple structure we can therefore define a single unit global distributed load to apply to the deck of the bridge as indicated in the following figure.

NOTE: In clause NA.2.44.5 it states that in order to obtain the most unfavourable effect this loading should be applied over all relevant areas of the footbridge deck with the direction of the force varied to match the direction of the vertical displacements of the mode for which responses are being calculated. This means that for the second vertical mode which will typically have a full sinusoid, half of the structure is going up while the other half is going down. To analyse this mode of vibration half of the length of the structure would need to be assigned the distributed loading with a positive load factor and the other half a negative load factor. If features are not present in the model that allow direct assignment of these loads to discrete features the discrete patch loads in LUSAS may be used to achieve this.



For the analysis the pedestrians are walking which provides us with the reference load F_0 of 280N acting over the area of the bridge which is $40m^2$. The remaining quantities that we need to determine are the pedestrian combined factor, k(fv), the unsynchronized reduction factor, γ, the total number of pedestrians distributed over the span, N, and the factor to reduce the effective number of pedestrians to the span contributing to the mode of interest, λ.

Based on the natural frequency being analysed of 1.76476Hz and the walking pedestrians, Figure NA.8 gives us a k(fv) value of 1.0. The unsynchronized reduction factor is dependent upon both the damping of the structure and the effective span for the mode shape corresponding to our vertical natural frequency. The effective span will be taken as the total span length of 20m as this is considered conservative by the code of practice but can also be calculated in accordance with Figure NA.7. Based on this and a logarithmic decrement δ of 0.18858 (equivalent to a damping ratio of 3% and calculated from $\delta = 2\pi\xi \big/ \sqrt{1-\xi^2}$), Figure NA.9 gives a γ value of 0.22.

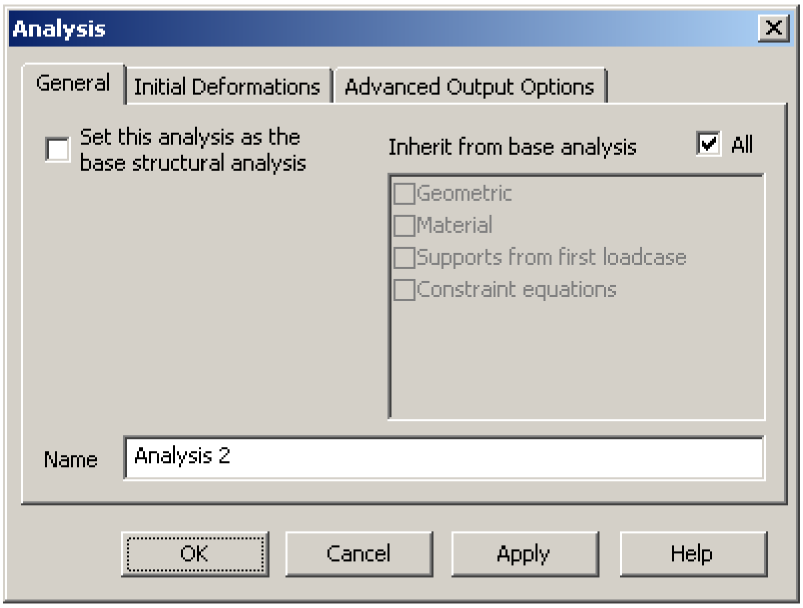
If we assume that the footbridge is for an urban route, Table NA.7 gives us a crowd density of walking pedestrians, ρ, of 0.8 persons/m^2. The total number of pedestrians distributed over the span is therefore ρA giving us a value for N of 32. The final parameter we need is the reduction factor for the effective number of pedestrians, λ, which, since we are conservatively assuming that the effective span is equal to the total span, is equal to 0.634.

We now have sufficient information to define our load curve and assign the loading to it. From the equation in NA.2.44.5 the amplitude and frequencies of our sine curve are as follows:

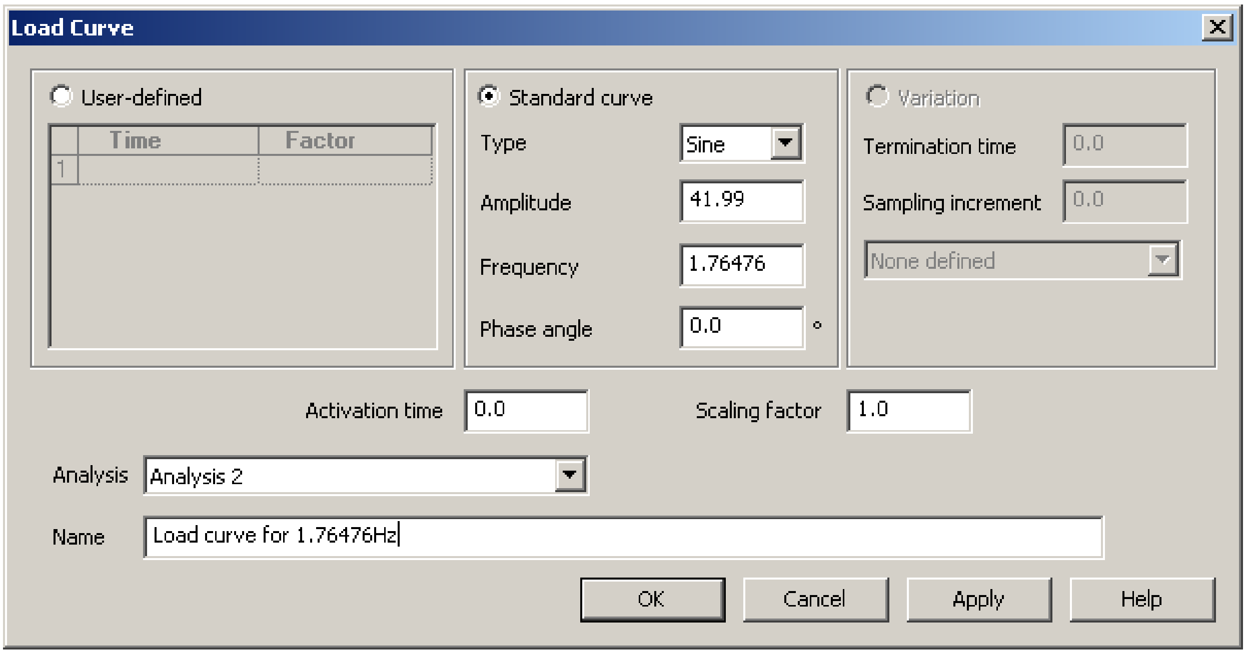
$$\begin{aligned}\text{Amplitude} &= 1.8\left(\frac{F_0}{A}\right).k\left(f_v\right).\sqrt{\gamma.N/\lambda} \\ &= 1.8\times\left(\frac{280}{40}\right)\times 1.0\times\sqrt{0.22\times 32/0.634} \\ &= 41.99\ \text{N/m}^2\end{aligned}$$

$$\text{Frequency} = f_v = 1.76476\text{Hz}$$

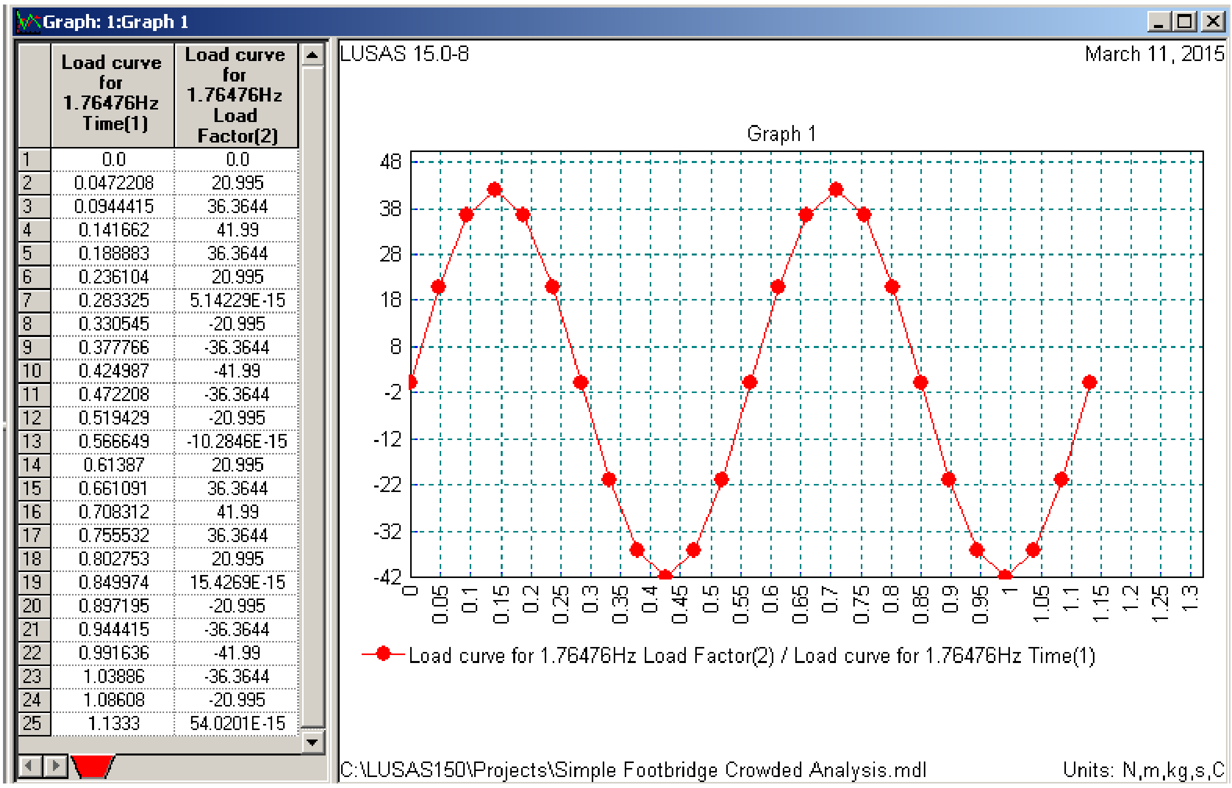
We can create a second analysis in the same model via **Analyses>General Structural Analysis** and inherit attribute assignments from “**Analysis 1**”:



This second analysis will be a transient dynamic analysis, named “**Analysis 2**”. The amplitude and frequency can then be entered directly into the sine load curve (**Analyses>Load Curve**) for use with “**Analysis 2**” as indicated below:

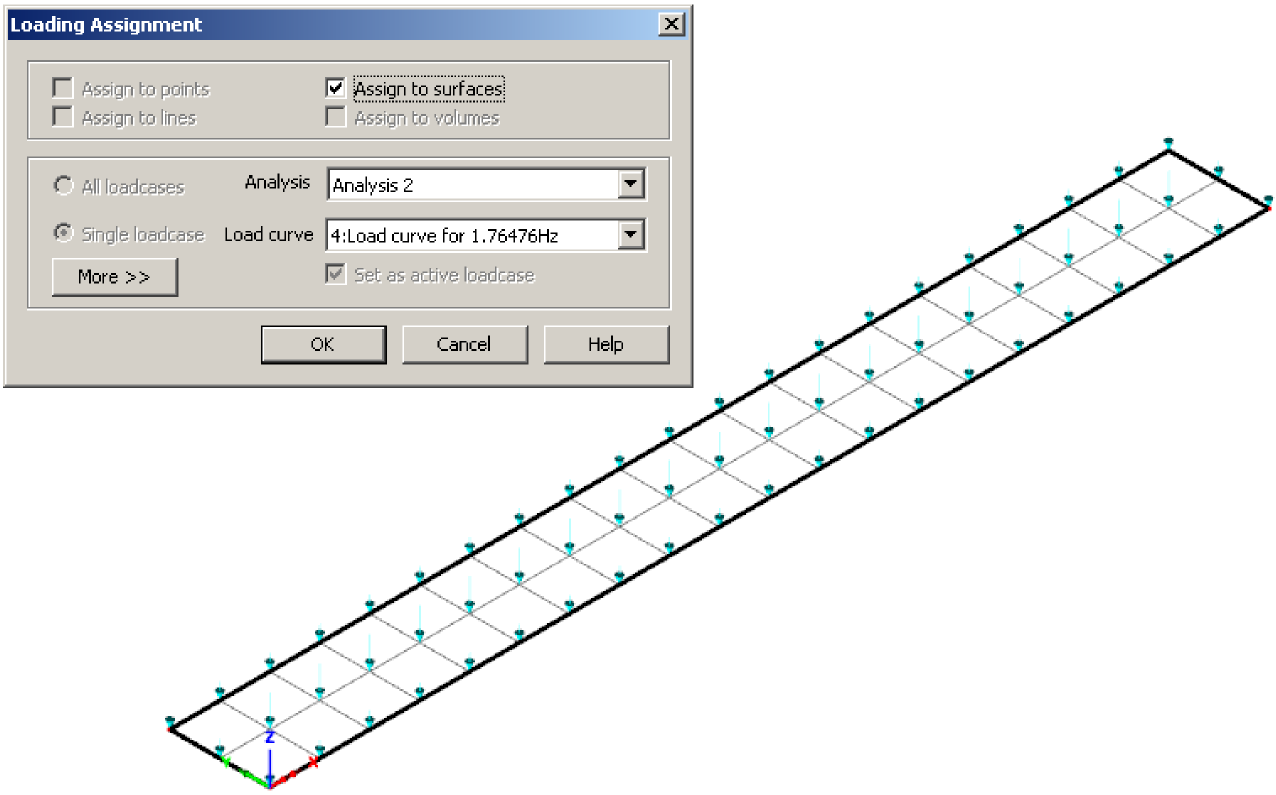


We can use the graph wizard (**Utilities>Graph Wizard**) to check that our load curve is correct as indicated below. From this we see that the amplitude is correct and one cycle takes approximately 0.567 seconds and so the frequency is 1.76476 Hz as required.



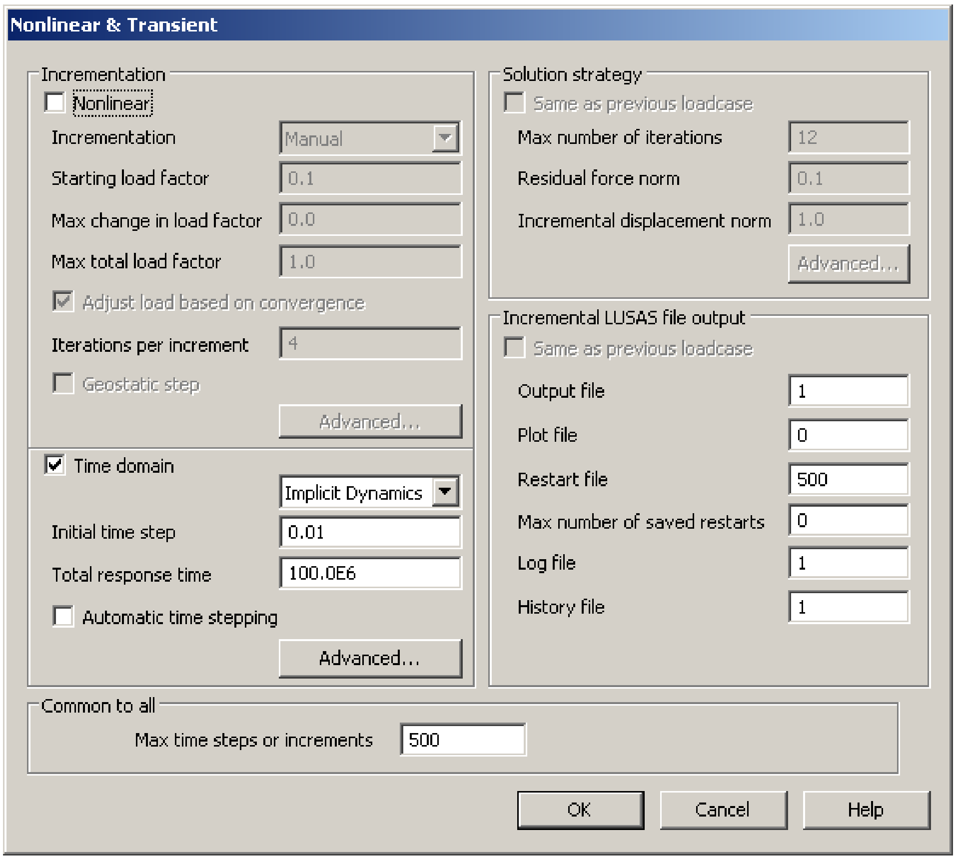
	Load curve for 1.76476Hz Time(1)	Load curve for 1.76476Hz Load Factor(2)
1	0.0	0.0
2	0.0472208	20.995
3	0.0944415	36.3644
4	0.141662	41.99
5	0.188883	36.3644
6	0.236104	20.995
7	0.283325	5.14229E-15
8	0.330545	-20.995
9	0.377766	-36.3644
10	0.424987	-41.99
11	0.472208	-36.3644
12	0.519429	-20.995
13	0.566649	-10.2846E-15
14	0.61387	20.995
15	0.661091	36.3644
16	0.708312	41.99
17	0.755532	36.3644
18	0.802753	20.995
19	0.849974	15.4269E-15
20	0.897195	-20.995
21	0.944415	-36.3644
22	0.991636	-41.99
23	1.03886	-36.3644
24	1.08608	-20.995
25	1.1333	54.0201E-15

We can now assign our unit distributed to the surface of the footbridge deck for this load curve:

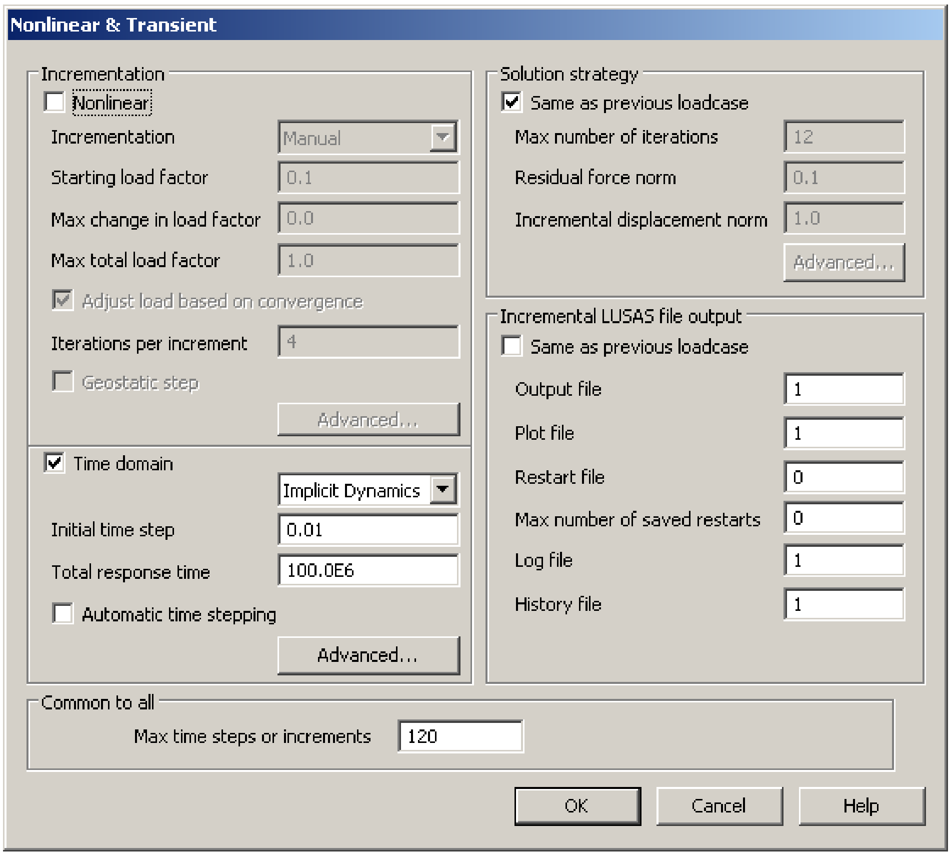


We now need to define the analysis controls for "**Analysis 2**" that will control the solution of the model. For this analysis we need to achieve the steady-state harmonic response of the structure to the vertical pulsating distributed load which will take an amount of time to achieve. If we know how long this will be we can simply define a dynamic control to cover this length of time where no output from the analysis is written to the LUSAS results file followed by a dynamic control that solves, at minimum, one complete cycle of the sinusoidal loading. This will, however, not usually be the case so while we will initially create a model that does this we will convert the analysis into a restart analysis that allows us to extend the solution incrementally until we do achieve the steady-state response.

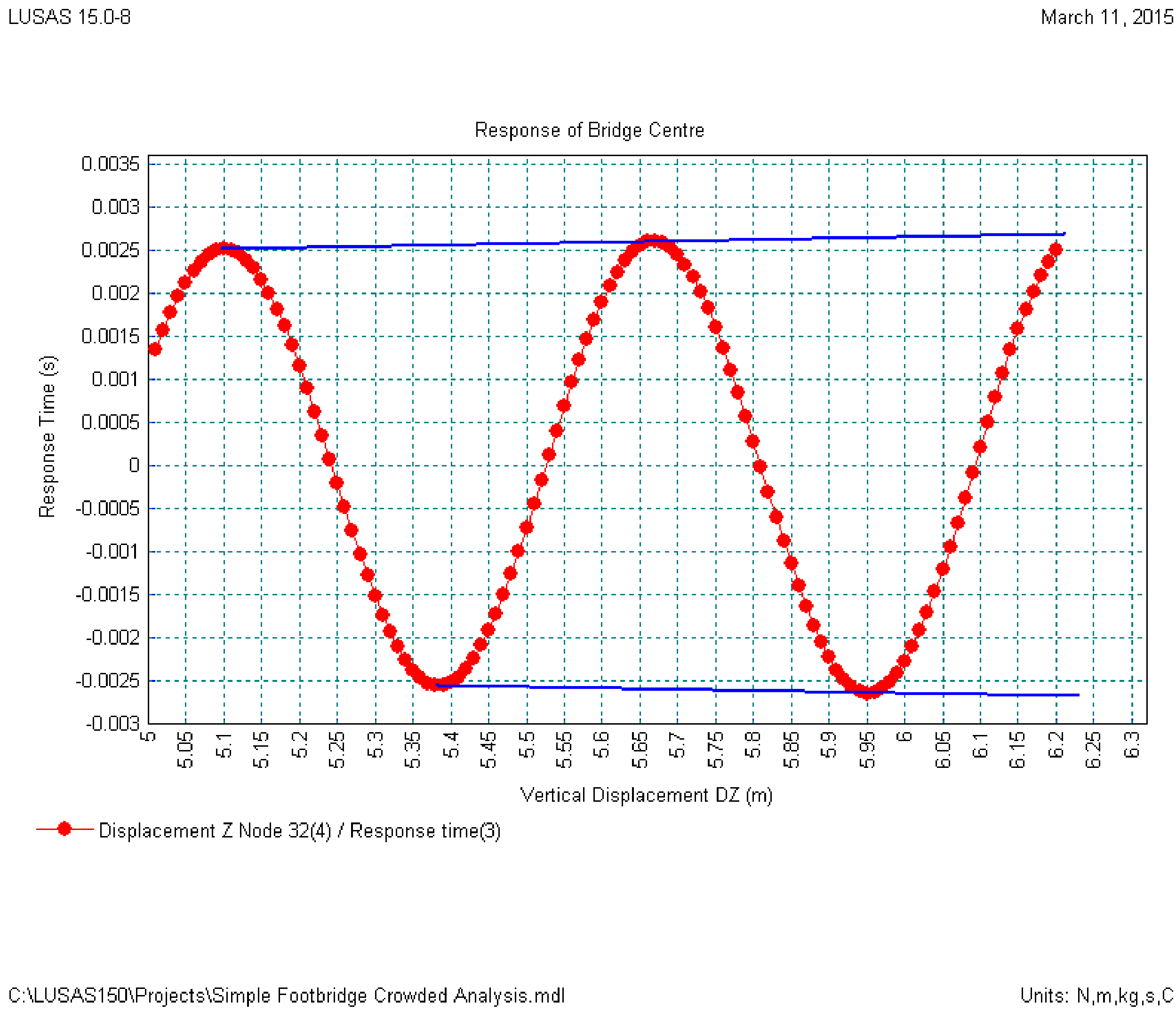
As part of a full analysis we should ensure that the time step used is sufficient to capture the dynamic behaviour of the structure. For this example we will simply use a time step of 0.01sec in the analysis. We can therefore add the following dynamic control to `Load case 2` in "**Analysis 2**" which will solve the first 5 seconds of the dynamic response of the structure to the pulsating load, making sure that the 'plot file' value is set to zero so that the results for this period are not presented, and the 'restart file' value is set to 500 to create a restart file after the 500^{th} step so that the analysis can be continued from this point if necessary.



Assuming at this stage that this will be sufficient to achieve the steady-state response of the structure we can then add a second load case to “Analysis 2” and assign this a new dynamic control to perform at least one cycle of the sine curve and output all of the results to the plot file (one cycle takes approximately 0.57 seconds so we will output 120 time steps to get approximately two complete cycles) as shown below, making sure we enable a different incremental output control in the dialog with the ‘plot file’ value set to 1 this time so that the results are presented.



If we save this model, solve it and look at the response of the central node from 5 seconds and draw straight lines between the approximate peaks of the curve as indicated below we see that the vertical displacement of the bridge centre is still increasing so we have not achieved the steady-state solution yet for this structure.



We could obviously edit our model and change the number of time steps in the first dynamic control from the 500 entered to a larger number and try again but this can be an inefficient way of attaining the steady-state solution for larger models which can take a significant length of time to solve the dynamic behaviour. A better option is to use the restart file generated after the 500th load step to continue the solution from this point.

In order to continue from the restart file, two new LUSAS datafiles are required, one to perform a further 500 increments without printing the results, and one to print the 120 load steps representing two cycles. For the first Datafile, create a new text file in the sub directory `/Associated Model Data/ Simple Footbridge Crowded Analysis` where the model is currently save and rename it:

```
Simple Footbridge Crowded Analysis~Analysis 2_Add5Seconds.dat
```

Now open the original datafile generated when the model was run in Modeller (`Simple Footbridge Crowded Analysis~Analysis 2.dat`) and scroll to the bottom. The set of commands under the first `DYNAMIC CONTROL` heading need to be copied into the newly-created datafile. Also add to the new datafile the line `RESTART READ` at the top and the lines `RESTART WRITE` and `END` at the bottom, so that it looks like this:

Simple Footbridge Crowded Analysis~Analysis 2_Add5Seconds.dat - Notepad

```
RESTART READ
DYNAMIC CONTROL
C dt [tsfac dtincf inctyp dtmin dtmax nperm]
  INCREMENTATION      1.00000000000000e-002     9.00000000000000e-001     D     D     D     1.00000000000000e+008     D
C alpha [beta gamma]
  CONSTANTS      0.00000000000000e+000     2.50000000000000e-001     5.00000000000000e-001
C incout [incplt incrst nrvtsv inclog inchis]
  OUTPUT     1     0     500     0     1     1
C maxinc [ttime dtterm]
  TERMINATION     500     1.00000000000000e+008     0.00000000000000e+000
RESTART WRITE
END
```

To create a second datafile to solve and output the two cycles of the pulsating load create another new text file and rename it:

`Simple Footbridge Crowded Analysis~Analysis 2_SteadyState.dat`

In this datafile enter the first line as `RESTART READ` then copy and paste from the original datafile the commands from the start of the second dynamic control to the end of the file, giving a file that looks like the one below:

Simple Footbridge Crowded Analysis~Analysis 2_SteadyState.dat - Notepad

```
RESTART READ
DYNAMIC CONTROL
C dt [tsfac dtincf inctyp dtmin dtmax nperm]
  INCREMENTATION      1.00000000000000e-002     9.00000000000000e-001     D     D     D     1.00000000000000e+008     D
C incout [incplt incrst nrvtsv inclog inchis]
  OUTPUT     1     1     0     0     1     1
C maxinc [ttime dtterm]
  TERMINATION     120     1.00000000000000e+008     0.00000000000000e+000
END
```

In order for the RESTART READ commands to work, the restart files must have the same file name as the datafile within which the command is written, so copy the restart file generated by the first datafile and rename the copy:

`Simple Footbridge Crowded Analysis~Analysis 2_Add5Seconds.rst`

This could alternatively be done in a batch file (explained below) using the following line of code (NOTE: All one line)

```
COPY " Simple Footbridge Crowded Analysis~Analysis 2.rst" " Simple
Footbridge Crowded Analysis~Analysis 2_Add5Seconds.rst"
```

We now need to create an MS-DOS batch file to run the two newly-created datafiles. To create the batch file, create a new text document and rename it `Add5Seconds.bat` and save it in the same location as the Datafiles. Right-click the batch file and click 'edit' and enter the commands shown below before saving it (NOTE: Again, each command is one line):

```
CALL "C:\LUSAS150\Programs\LUSAS_S.BAT" "Simple Footbridge Crowded
Analysis~Analysis 2_Add5Seconds.dat"

COPY "Simple Footbridge Crowded Analysis~Analysis 2_Add5Seconds.rst" "Simple
Footbridge Crowded Analysis~Analysis 2_SteadyState.rst"

CALL "C:\LUSAS150\Programs\LUSAS_S.BAT" "Simple Footbridge Crowded Analysis~Analysis
2_SteadyState.dat"
```

The batch file can now be run by double-clicking its file icon.

The new results file can be viewed in LUSAS Modeller by opening the original model file then opening the new results file `Simple Footbridge Crowded Analysis_SteadyState.mys` on top.

To open the results on top of the model:

1. Open the model file `Simple Footbridge Crowded Analysis.mdl` in LUSAS Modeller

2. Go to **File>Manage Results Files . . .**

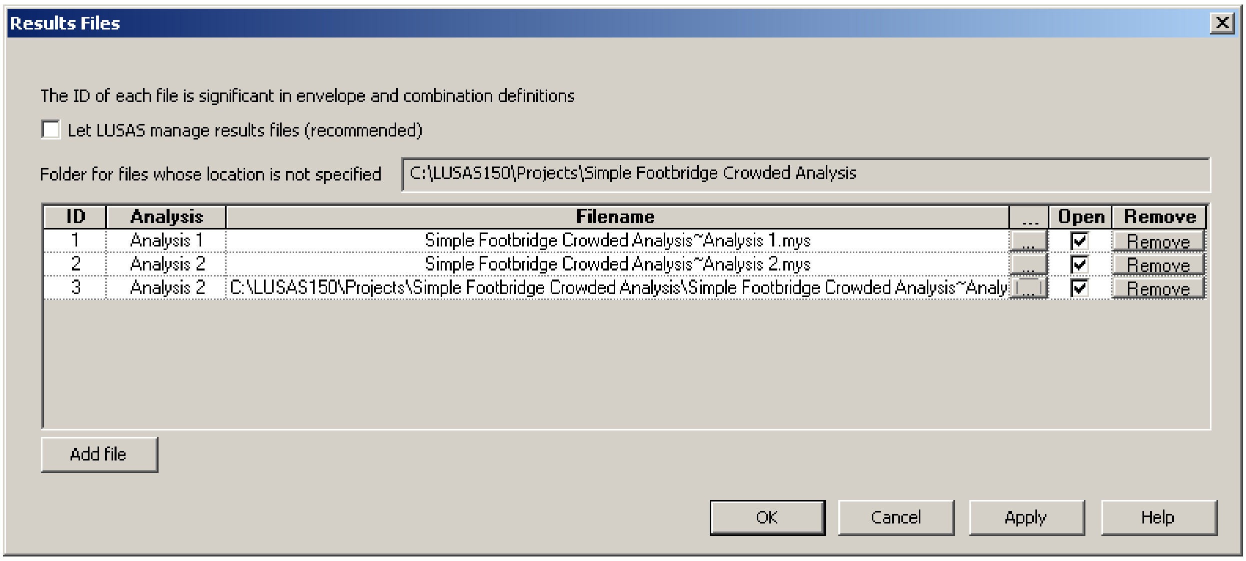
1. Un-tick the option "**Let LUSAS manage results files (recommended)**"

3. Click "**Add File**"

4. Change the **Analysis** for the new results file to open to "**Analysis 2**"

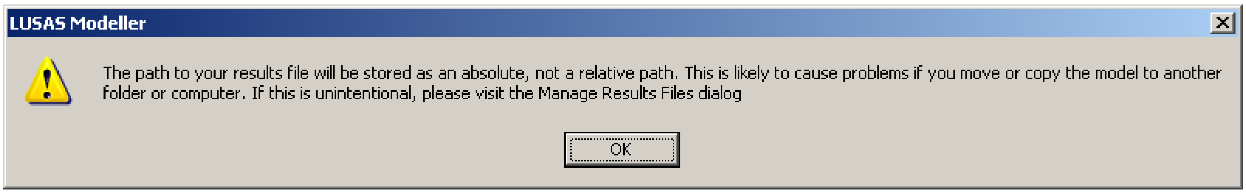
5. Click the "**…**" button to browse for the results file `Simple Footbridge Crowded Analysis~Analysis 2_SteadyState.mys` and click "**OK**"

6. Tick the **Open** checkbox to open the results now and then click "**OK**"



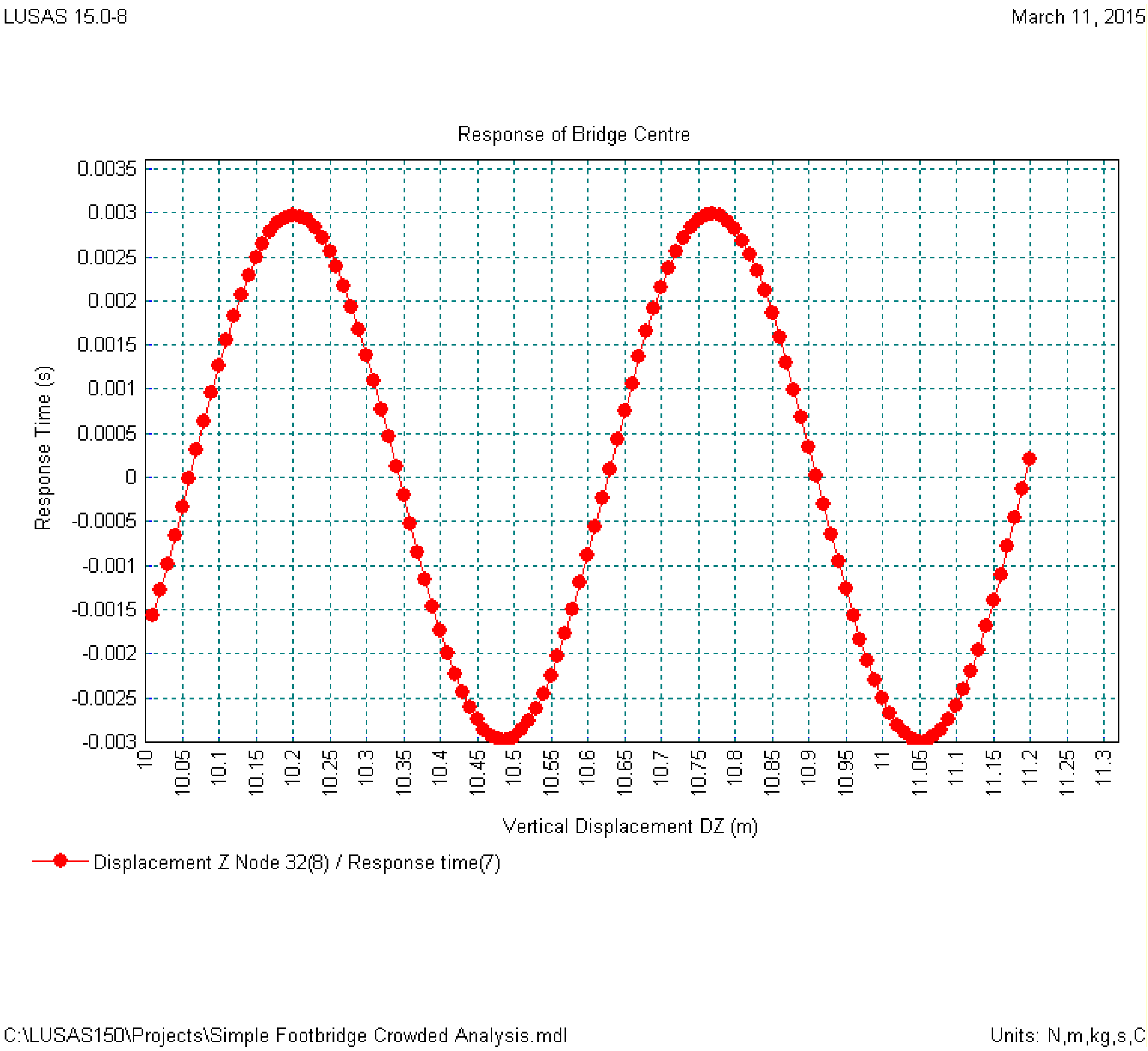
Note

The following message will appear:



The locations of the results files are now manually set using the full path to the results file. If the model is to be moved to a different location on the PC or network, then the paths to the results files will need to be edited as well. If the model is later changed and solved again, then it may be best to tick the **"Let LUSAS manage results files (recommended)"** option again before solving, and repeat the process above if and when there are any further results files subsequently created outside Modeller with different file names or stored in a different location to those generated by Modeller in the normal way.

By graphing the vertical central displacement we see that we appear to have achieved our steady-state solution for this structure.



If we had not achieved the steady-state response then to add a further 5 seconds we would only need to rerun the batch file as many times as necessary to achieve the steady-state condition.

The analysis is now ready for post-processing in LUSAS Modeller as normal.