Initial exploration of transmissibility by FEA of blades.

These are notes I made while trying various ideas for blade transmissibility. They are not very well structured, they do not reach any particular conclusions, and they leave some questions unanswered. I am publishing them simply as a reference for future work.

1. MODAL ANALYSIS OF "REFERENCE" BLADE.

See directory test4.

The natural frequency of this blade was measured and found to be (from memory) 55 Hz. In order to be sure I am doing the right things I made a simple FE model to see if I could get the same frequency. The dimensions were measured for me by Mike Plissi:

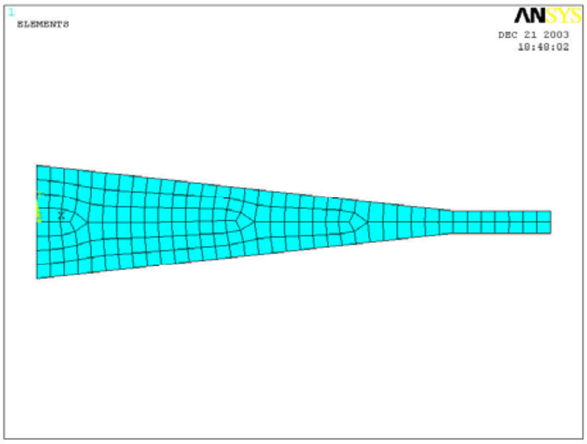
Root width: 82mm
Length: 370mm
Plain portion at end 70mm long by 16mm wide
Thickness 2 mm

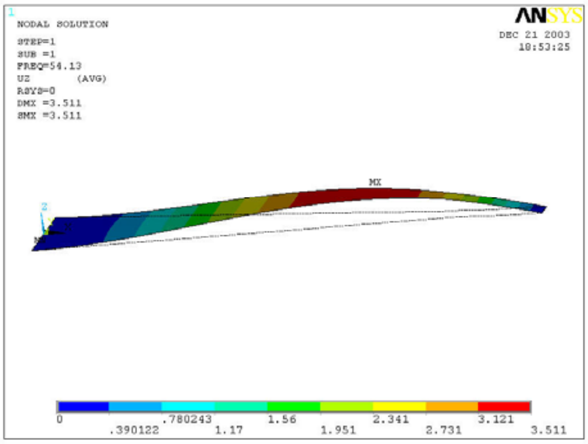
See appendix 1 for macro.
blength=0.37
taperl=0.30
rootwidth=0.082
hroot=rootwidth/2
tipwidth=0.016
htip=tipwidth/2
bthick=0.002
maryoung=1.76e11
marpoiss=0.3
mardens=7800
dampratio=1e-4

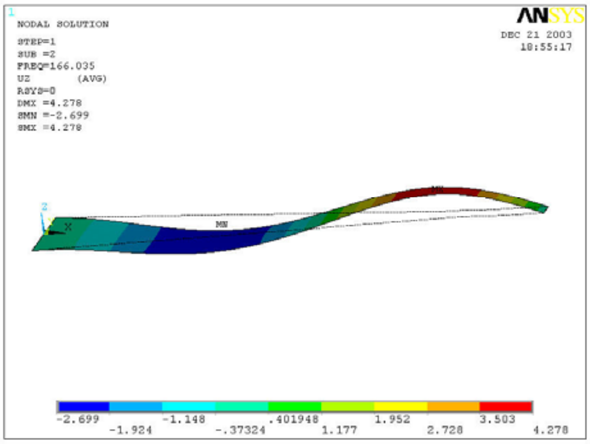
Results were

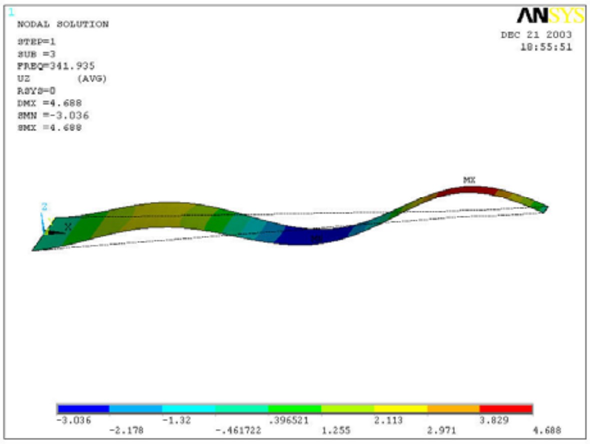
```
  *****  INDEX OF DATA SETS ON RESULTS FILE  *****

   SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
     1  54.130            1         1         1
     2  166.03            1         2         1
     3  341.94            1         3         1
     4  372.61            1         4         1
```

The 54 Hz is very close to 55 – I would not normally expect such a good match and it is an encouraging sign!
See mode shapes next page:









2. MODAL ANALYSIS OF BLADE FROM CONCEPTUAL DESIGN DOC.

See macro in appendix 2. This macro builds a simple model of the blade including an 11kg point mass at the tip.

Blade with variables as follows:
blength=0.48
rootwidth=0.096
hroot=rootwidth/2
tipwidth=0.013
htip=tipwidth/2
bthick=0.0045
maryoung=1.76e11
marpoiss=0.3
mardens=7800
dampratio=1e-4

2.1 Modal analysis – tip fixed in "Z"

(mass of 11 kg was removed, and tip was fixed at the centre in Z for this test)

Results were

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	71.284	1	1	1
2	219.33	1	2	1
3	401.55	1	3	1
4	406.80	1	4	1
5	452.82	1	5	1
6	771.20	1	6	1

```
     7  947.74             1         7         1
```

According to the blade formulae, and extrapolating from the FE results for the reference blade, the first mode should be
F=54.13*(.37/.48)^2*(4.5/2)=72.1Hz, which is very close.

2.2 Modal analysis – tip loaded with a mass of 11kg

(the 11kg mass was in place, no constraint at tip. As per macro in appendix 2)

frequency results were

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE	
1	2.4270	1	1	1	1st bounce
2	31.789	1	2	1	1st lateral
3	71.477	1	3	1	2nd bounce
4	219.49	1	4	1	3rd bounce
5	378.58	1	5	1	1st elongation
6	406.80	1	6	1	1st twist
7	452.96	1	7	1	4th bounce
8	771.32	1	8	1	5th bounce
9	793.91	1	9	1	2nd lateral
10	947.74	1	10	1	2nd twist

The first frequency is the 11kg mass in a bounce mode. (predicted at 2.5 Hz see below). The second is a lateral mode of the 11kg mass. The third is the same as above (nearly). I have labeled all the modes from examining the mode shapes.

3. HARMONIC ANALYSIS OF CONCEPTUAL DESIGN BLADE

For the harmonic analysis I moved the blade root in Z by a unit displacement. This is done at a variety of frequencies (the command pair HARFRQ and NSUBST specify the range and the number of frequencies in the range respectively).

3.1 Harmonic – no damping

Harmonic analysis 0-100 Hz looked sensible. (Very similar to plot in section 3.2 below.) Double peak at end of blade, single peak in middle of blade at around 70Hz. Narrowed in 60-80 Hz 20 steps, max mag was (?) 100.
Narrowed freq range to

```
HARFRQ,71.3,71.45
NSUBST,20,
```

Max magnitude now 2897 at end.
Narrowed to

```
HARFRQ,71.38,71.4
NSUBST,20,
```

Some shape now evident (more obvious with a logarithmic y axis).

```
HARFRQ,71.389,71.391
NSUBST,20,
```

Closer, but still no rounded peak. Saved in 1b.txt but frequencies are only to 3 places. Try again!

```
HARFRQ,71.3898,71.390
NSUBST,20,
```

Peak has now reached 3*10^6 and there are warning messages about pivot terms at one (the peak?) frequency.

```
   *** NODAL LOAD CALCULATION TIMES
  TYPE  NUMBER   ENAME      TOTAL CP  AVE CP

     1     195  SHELL93       0.000   0.000000
     2       1  MASS21        0.000   0.000000
 *** LOAD STEP     1   SUBSTEP     1  COMPLETED.  FREQUENCY=   71.3898
 *** LOAD STEP     1   SUBSTEP     2  COMPLETED.  FREQUENCY=   71.3898
 *** LOAD STEP     1   SUBSTEP     3  COMPLETED.  FREQUENCY=   71.3898
 *** LOAD STEP     1   SUBSTEP     4  COMPLETED.  FREQUENCY=   71.3898
 *** LOAD STEP     1   SUBSTEP     5  COMPLETED.  FREQUENCY=   71.3898
 *** LOAD STEP     1   SUBSTEP     6  COMPLETED.  FREQUENCY=   71.3899
 *** LOAD STEP     1   SUBSTEP     7  COMPLETED.  FREQUENCY=   71.3899
 *** LOAD STEP     1   SUBSTEP     8  COMPLETED.  FREQUENCY=   71.3899
 *** LOAD STEP     1   SUBSTEP     9  COMPLETED.  FREQUENCY=   71.3899
 *** LOAD STEP     1   SUBSTEP    10  COMPLETED.  FREQUENCY=   71.3899
 Small pivot at node 92 ROTY.  Check for unconstrained model.

 *** WARNING ***                        CP=     70.301   TIME= 10:17:52
 There are 1 small equation solver pivot terms.  Check for an
```

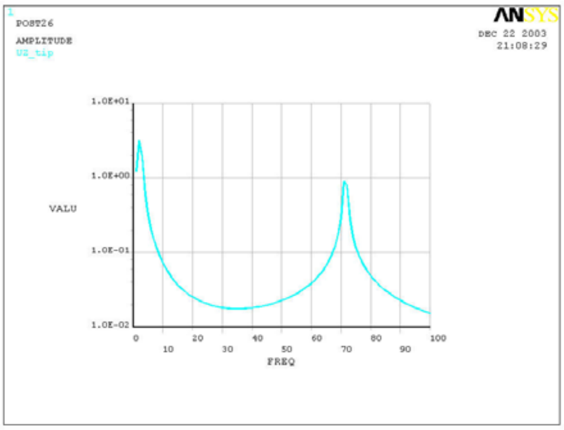
```
insufficiently constrained model.
*** LOAD STEP     1   SUBSTEP    11  COMPLETED.  FREQUENCY=   71.3899
Small pivot at node 92 ROTY.  Check for unconstrained model.

*** WARNING ***                         CP=      71.042   TIME= 10:17:53
There are 1 small equation solver pivot terms.  Check for an
insufficiently constrained model.
*** LOAD STEP     1   SUBSTEP    12  COMPLETED.  FREQUENCY=   71.3899
Small pivot at node 92 ROTY.  Check for unconstrained model.

*** WARNING ***                         CP=      71.703   TIME= 10:17:53
There are 1 small equation solver pivot terms.  Check for an
insufficiently constrained model.
*** LOAD STEP     1   SUBSTEP    13  COMPLETED.  FREQUENCY=   71.3899
*** LOAD STEP     1   SUBSTEP    14  COMPLETED.  FREQUENCY=   71.3899
*** LOAD STEP     1   SUBSTEP    15  COMPLETED.  FREQUENCY=   71.3899
*** LOAD STEP     1   SUBSTEP    16  COMPLETED.  FREQUENCY=   71.3900
*** LOAD STEP     1   SUBSTEP    17  COMPLETED.  FREQUENCY=   71.3900
*** LOAD STEP     1   SUBSTEP    18  COMPLETED.  FREQUENCY=   71.3900
*** LOAD STEP     1   SUBSTEP    19  COMPLETED.  FREQUENCY=   71.3900
*** LOAD STEP     1   SUBSTEP    20  COMPLETED.  FREQUENCY=   71.3900

*** NOTE ***                            CP=      76.370   TIME= 10:17:58
Solution is done!
```

3.2 Harmonic – with damping

Introduced damping with a damping ratio of 1e-4.

Response at tip in range 0-100Hz

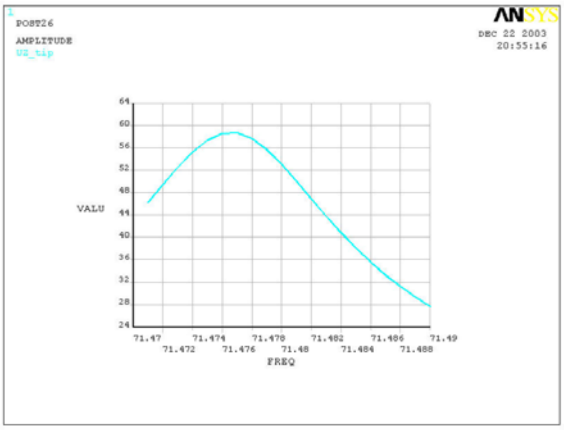
```
HARFRQ,0,100
NSUBST,20,
```



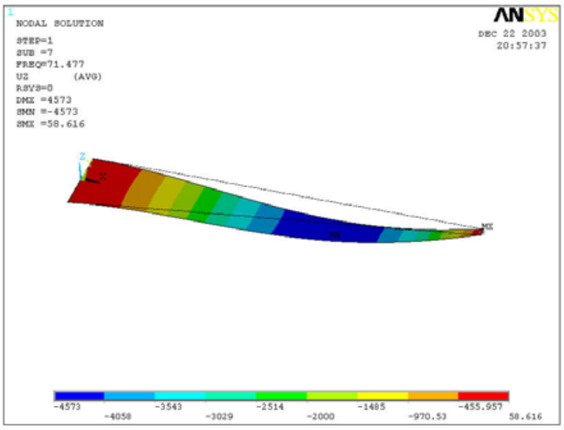
But what is the max displacement at the tip at the ~70Hz resonance?

After few tries for the right frequency limits, got a nicely shaped peak with max amplitude of ~58 at the mass at f~71.477 Hz.

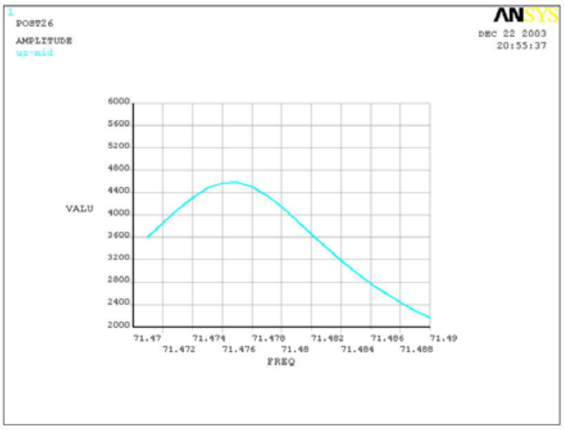
```
HARFRQ,71.38,71.4
NSUBST,20,
```



Blade shape in post1. All makes sense. Tip moves +58.6, middle of blade moves –4573 at node 421.



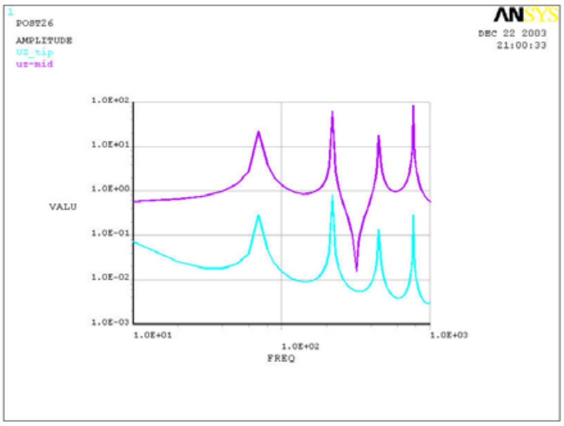
Back to post26, displacement at node 421 is a shaped peak at ~4600.



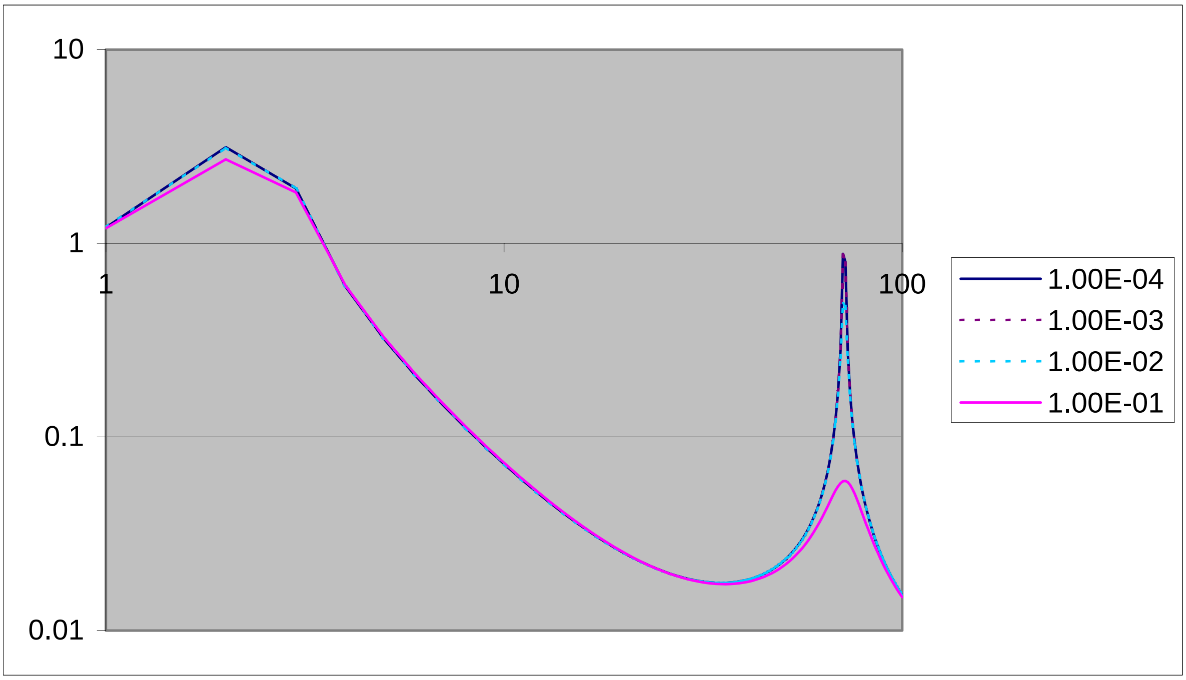
Now look beyond 100 Hz (NB first result is at 10 Hz so peak at ~2.5Hz will be missing)

```
HARFRQ,0,1000
NSUBST,100,
```

In this plot I include the motion of the mid-point and the tip:

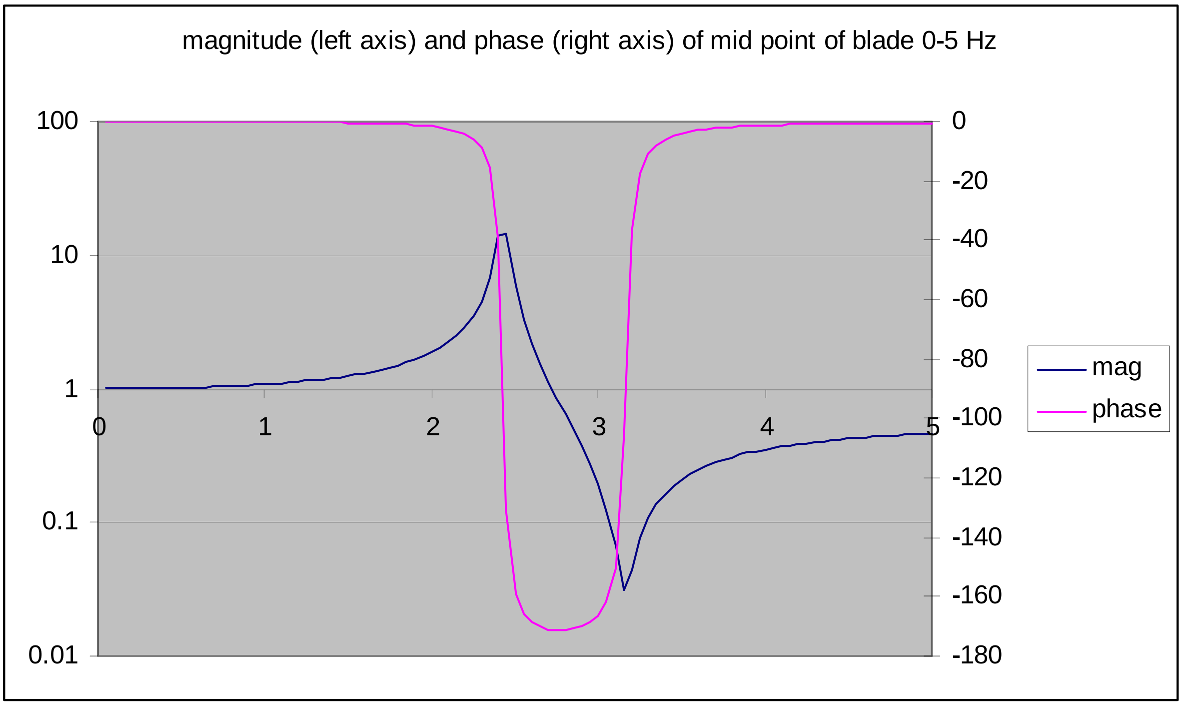


For completeness here is the 1-100 Hz plot repeated for damping ratios of 1e-1,1e-2,1e-3, 1e-4.



Note that only .1 and .01 are distinguishable from the others (and 0.01 only marginally so)

And something else I saw whilst preparing the above – the response of the mid-point of the blade has a null just above the first bounce mode:



Is this expected?

4. EFFECT OF DIFFERENT BLADE GEOMETRY

4.1 Simple change to reference blade

What would be effect on the internal modes the "reference" blade (section 1 above), rather than having a plain (non-tapered) portion at the end, were simply trapezoidal? This can be easily done. The width at the point x=0.3 is increased to (16+(82-16)*70/370) or 28.4mm

```
K,1 ,0,-hroot,0,
K,2 ,taperl,-0.0284/2,0,
K,3 ,blength,-htip,0,
K,4,blength,htip,0,
k,5,taperl,0.0284/2,0
K,6 ,0,hroot,0,
   SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
     1  53.135            1          1         1
     2  164.13            1          2         1
     3  325.04            1          3         1
     4  339.36            1          4         1
```

Not very different from the reference result. And, for completeness, going even further:

```
K,1 ,0,-hroot,0,
K,2 ,taperl,-0.0284,0,
K,3 ,blength,-htip,0,
K,4,blength,htip,0,
k,5,taperl,0.0284,0
K,6 ,0,hroot,0,

   SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
     1  50.980            1          1         1
     2  161.26            1          2         1
     3  233.57            1          3         1
     4  335.70            1          4         1
```

It would seem that the additional mass in the centre is offset by additional stiffness to a large extend, although the reduced width ("waisting") does give a small increase in internal mode frequency.

4.2 Shortening the blade and widening the blade root – theory

An implication of the blade formulae is that we can select the blade length and still achieve the required uncoupled mode and stress levels provide we select width and thickness appropriately. Earlier work (ref) also observed the counter-intuitive effect that a shorter blade, designed in this way for width and thickness, would be wider at the root than a longer one. It also observed that for optimal blade designs – ones having the highest possible internal modes – all such blades would form a family having the same internal modes. However, this last observation was tempered with the proviso that the shape factor remain the same between the different blades. If we, for example, keep a constant width at the tip then varying the base will lead to a change in shape factor and so the internal modes won't be quite the same. To illustrate, I have made various designs of blade in a spreadsheet:

youngs modulus	1.76E+11	1.76E+11	1.76E+11	1.76E+11	1.76E+11
total mass	60	60	60	60	60
local mass	11	11	11	11	11
		Const tip width		Const shape factor	
length	0.480	**0.400**	**0.350**	**0.400**	**0.350**
thickness	4.50E-03	**3.34E-03**	**2.64E-03**	**3.12E-03**	**2.39E-03**
root width	0.096	**0.145**	**0.203**	**0.166**	**0.249**
tip width	0.013	0.013	0.013	**0.023**	**0.034**
beta	0.135	**0.090**	**0.064**	0.135	0.135
alpha	1.265	**1.357**	**1.401**	1.265	1.265
max stress	872.000	872.094	871.949	871.969	871.923
uncoupled f	2.517	2.514	2.514	2.515	2.513
internal mode - from FE	71.200				
int mode extrapolated		76.185	78.684	71.115	70.994

The numbers in red were found by using an optimizer to match the max stress and uncoupled frequency to those in the first column. The values in bold are the changes between columns. In the last two columns, where the tip width was varied to keep the shape factor constant, the predicted internal mode is (nearly) constant, whereas in the third and fourth columns, in which the tip width was maintained, the shorter blades have slightly higher internal modes.

It is a simple matter to test find the lowest mode of each using the macro, the results were:

		Const tip width		Const shape factor	
length	0.480	**0.400**	**0.350**	**0.400**	**0.350**
thickness	4.50E-03	**3.34E-03**	**2.64E-03**	**3.12E-03**	**2.39E-03**
root width	0.096	**0.145**	**0.203**	**0.166**	**0.249**
tip width	0.013	0.013	0.013	**0.023**	**0.034**
beta	0.135	**0.090**	**0.064**	0.135	0.135
alpha	1.265	**1.357**	**1.401**	1.265	1.265
internal mode - from FE	71.200				
int mode extrapolated		76.185	78.684	71.115	70.994
Int mode from FE		76.7	78.9	71.5	71.1
(check) 1st bounce mode	2.42	2.47	2.48	2.43	2.42

From all of which I conclude that

(1) the equation for predicting internal modes based simply on length and thickness seems to work well even when the shape factor is changed.

(2) Making the blade shorter and wider slightly increases the internal modes (but these were rather extreme changes in length and the effect on internal modes was modest).

5. PRESTRESS

I can see no reason why the above analysis will not apply when the blade is stressed. But there clearly are prestress effects and it should be possible with ANSYS to model the blade in a prestressed state and then calculate the internal modes. See macro in appendix 3. (directory test7).

First step – apply a load upwards and then "freeze" the displaced shape with UPCOORD command. Tested by removing the load and solving. Stayed bent with zero stress. Max Z dimension is 0.18704

Reapplied 600N load downwards, nonlinear solution, blade ended up bent down with max displacement 0.2m (NB this is cf bent shape, not cf straight!)

Removed 600N downward force and instead set tip displacement to 0.187. Reaction force was 507N and shape was near straight.

Removed constraint, applied 507N downward force. Blade ends up straight. (max UZ=.1867) and with realistic stresses. Why do we need only 507N to bend it back when it took 600N to bend it in the first place? Because the 507N has a longer lever arm to act on in the steady (loaded) state than did the 600N.

See macro in appendix 3.

All seems under control, so revert to enforced displacement at tip and do a harmonic analysis (see also ANSYS documentation structural guide section 3.2). See macro in appendix 4.

Results from output file were

```
1      31.75955334249
 2      73.25653650001
 3      221.0199066545
 4      381.6920521776
 5      402.0528839514
 6      455.7037349231
 7      773.7125527728
 8      794.0644057474
 9      943.0960317930
```

But I could not get the mode shapes in POST1. Not sure why – I followed the instructions in the documentation. I guess the 31.7 Hz mode is the tip lateral and the 73 Hz mode is the first internal "bounce" mode. (Compare with 2.2 above.). I would say that the prestress has had little or no effect.

6. EFFECT OF WIRE CLAMP

All of the above does not address the question of what will be the effect of the mass of a wire clamp fixed to the end of the blade. For this we need to decouple the 11 kg mass and fix it to the blade with a wire. If we simply model the wire without any tension in it, then it will have the correct longitudinal stiffness but it will be very floppy laterally and so there will be lots of spurious low-frequency modes. There are several ways of getting around this:

- Constrain the wire to move only in Z so that the sideways modes are suppressed. This would constrain the blade tip and the 11kg mass so there would be no lateral or blade stretch modes.

- Use a spring to simulate the wire, with the mass at the end of it. In this case the mass will have to be constrained in x and y.

- Prestress the wire so that it has the correct internal modes. It would still be necessary to constrain the mass because the pendulum behaviour is not captured by the (linear) harmonic analysis.

Of these, the method with the wire seems the simplest.

6.1 Blade with wire clamp but no wire and no test mass

See directory test8. A wire clamp 1cm by 1cm by 2cm would be generous in size and have a mass of .1*.1*.2*8=.016kg (16g=2cc*8g/cc).

First analyse with no mass at the tip, then with a mass:

	no mass	16g mass
1	24.925	23.335
2	114.15	104.54
3	287.69	263.27

4	401.55	367.53
5	406.8	406.8
6	544.32	501.81
7	885.37	823.12
8	947.74	947.74

So the mass has, as expected, a small effect on the early modes. (But quite significant on higher ones.) Note from 2.2 that modes 5 and 8 are twist modes so we would not expect any effect from the point mass.

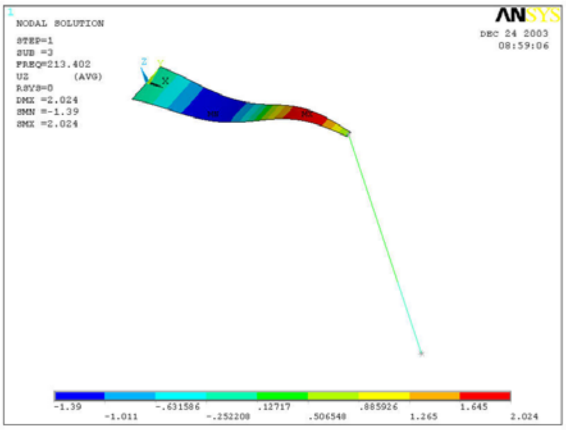
6.2 Blade with clamp, wire, and mass.

For now, ignore the fact that the wire is not vertical. See macro, appendix 5.

```
blength=0.48
bthick=0.00450
rootwidth=0.096
tipwidth=0.013
hroot=rootwidth/2
htip=tipwidth/2
maryoung=1.76e11
marpoiss=0.3
mardens=7800
wireyoung=2e11
wirepoiss=0,3
wiredens=7800
dampratio=1e-4
tipmass=.016
wiredia=7e-4*2
wirelen=0.54
testmass=11
```

Results:

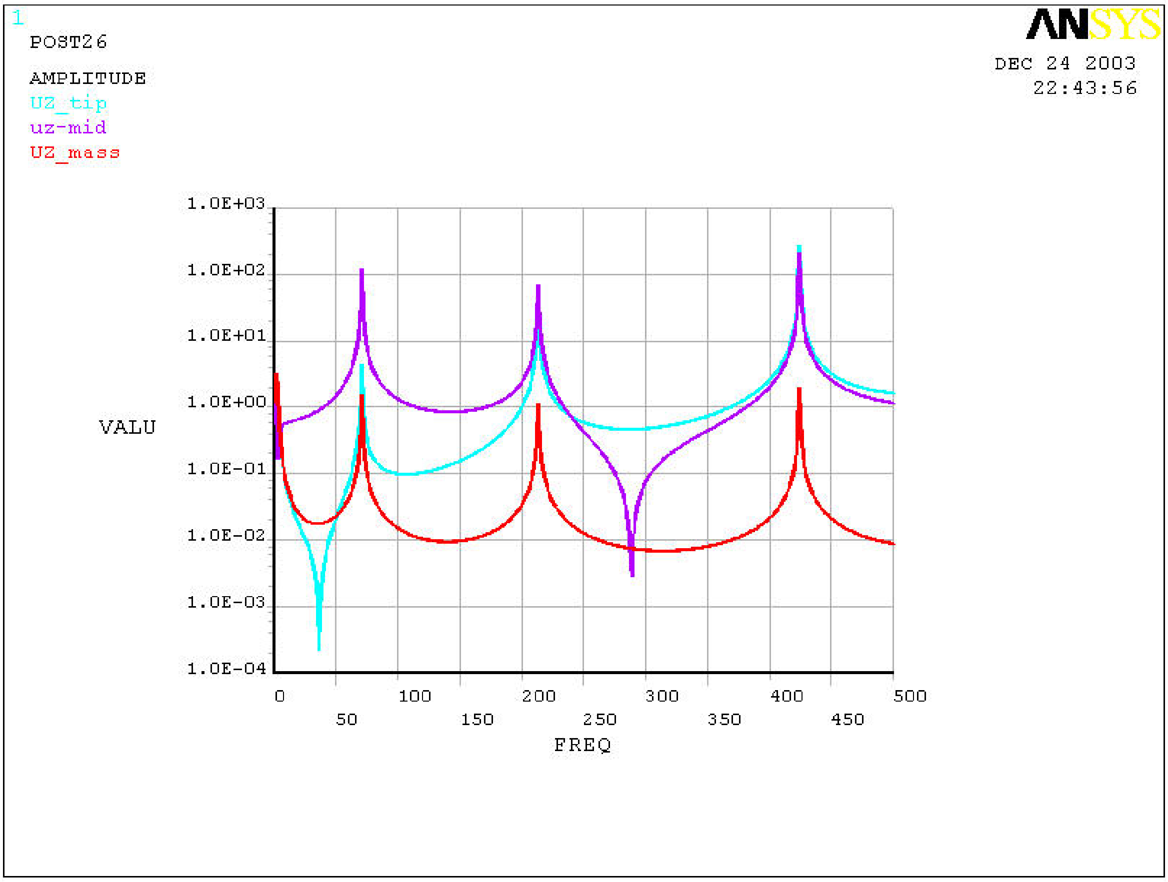
Previous result (section 2.2)		This result	
2.427	1st bounce	2.4192	
31.789	1st lateral	suppressed	
71.477	2nd bounce	70.73	
219.49	3rd bounce	213.4	
378.58	1st elongation	suppressed	Not expected – the 11 kg mass is no longer at the blade tip
406.8	1st twist	406.8	unaffected
452.96	4th bounce	424.26	Now has tip movement
771.32	5th bounce	651.72	Now has tip movement
793.91	2nd lateral	792.64	Not suppressed – no lateral movement at blade tip
	6th bounce	886.73	Now has tip movement
947.74	2nd twist	947.74	unaffected



6.3 Transmissibility 0 – 500 Hz

The transmissibility in the range 0-500Hz at the blade midpoint (UZ_mid, topmost trace), the blade tip where the wire clamp is (UZ_tip, middle trace) and at the test mass (UZ_mass, lowest trace).

```
dampratio=1e-4

HARFRQ,0,1000
NSUBST,1000,
```



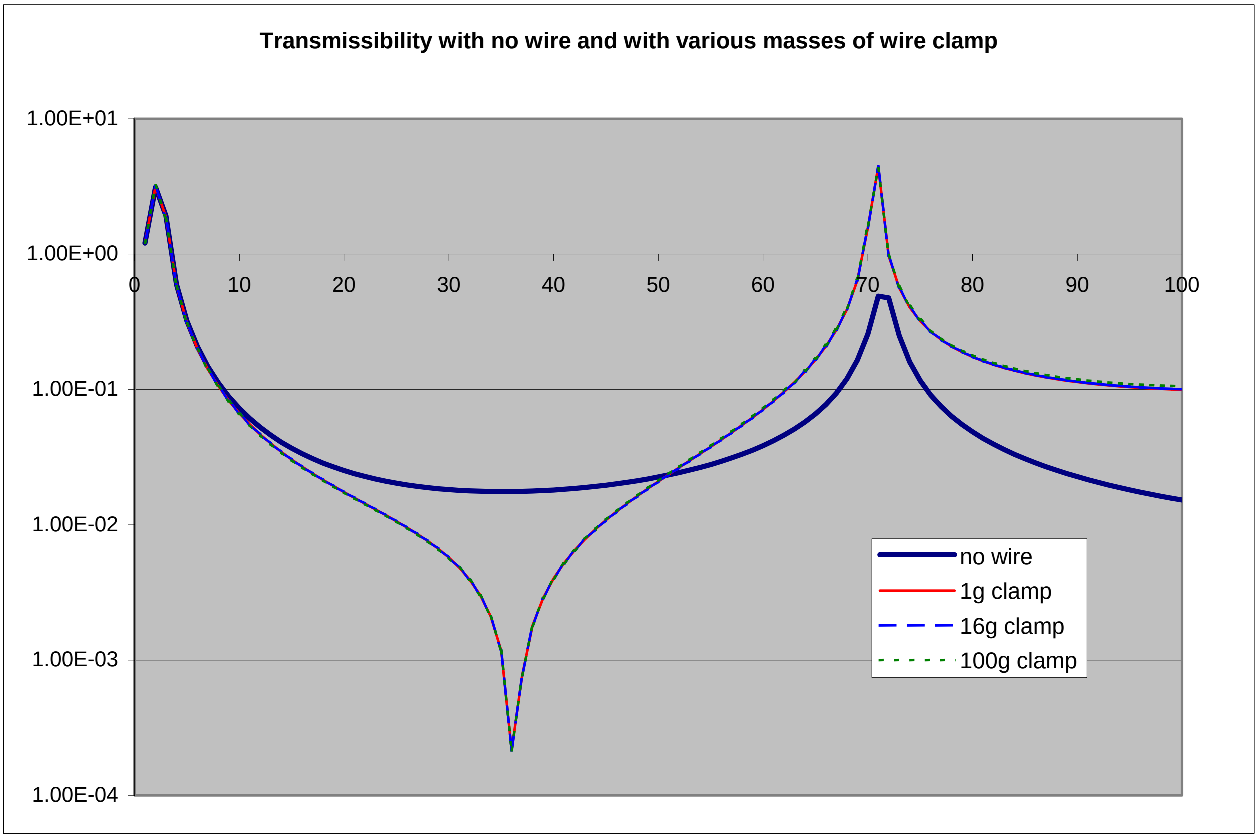
6.4 Transmissibility with various wire clamp masses

Taking results for 1-100 Hz with

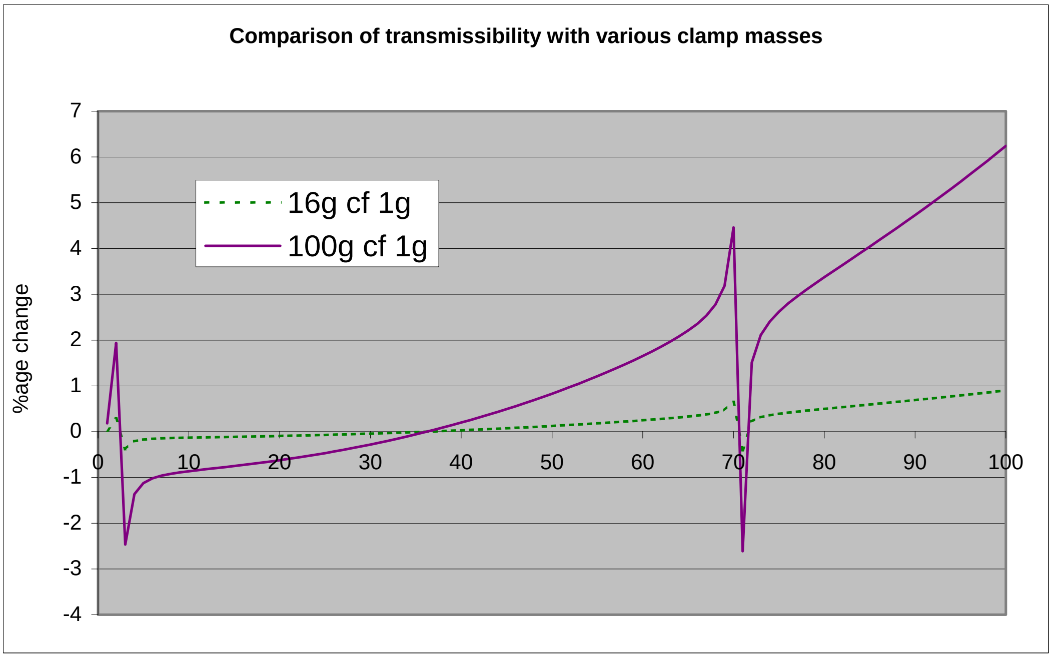
- Mass on end of blade (case described in 3.2 above)
- Wire and very small clamp (0.001kg)
- Wire with 16 g clamp (6.2 + 6.3 above)

- Wire with 100g clamp

```
blength=0.48
bthick=0.00450
rootwidth=0.096
tipwidth=0.013
hroot=rootwidth/2
htip=tipwidth/2
maryoung=1.76e11
marpoiss=0.3
mardens=7800
wireyoung=2e11
wirepoiss=0,3
wiredens=7800
dampratio=1e-4
tipmass=.1
wiredia=7e-4*2
wirelen=0.54
testmass=11

HARFRQ,0,100
NSUBST,100
```



The transmissibility numbers are slightly different for the different clamp masses but no enough to show on the graph. So (partly to check I really have changed the model!) here are the percentage changes 16g/1g ad 100/1g:



The effects of the clamp mass look modest – even a 100g clamp changes things by only a few percent at 100 Hz.

Appendix 1. Macro for "reference" blade

```
finish
/CLEAR,START
*abbr,doit,doit
/input,start71,ans,'C:\Program
Files\Ansys
Inc\v71\ANSYS\apdl\',,,,,,,,,,,,,,,,1
/PREP7
!*
! values of parameters
blength=0.37
taperl=0.30
rootwidth=0.082
hroot=rootwidth/2
tipwidth=0.016
htip=tipwidth/2
bthick=0.002
maryoung=1.76e11
marpoiss=0.3
mardens=7800
dampratio=1e-4

!*
ET,1,SHELL93
R,1,bthick, , , , , ,
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,maryoung
MPDATA,PRXY,1,,marpoiss
MPTEMP,1,0
MPDATA,DENS,1,,mardens

K,1 ,0,-hroot,0,
K,2 ,taperl,-htip,0,
K,3 ,blength,-htip,0,
K,4,blength,htip,0,
k,5,taperl,htip,0
K,6 ,0,hroot,0,

LSTR,1,2
,2,3
,3,4
,4,5
,5,6
,6,1

AL,1,2,3,4,5,6


aplot
ESIZE,hroot/4,0
amesh,1,2


DL,6,,all,0
DL,3,,UZ,0

! modal analysis
FINISH
/SOL
ANTYPE,2
!*
ANTYPE,2
MSAVE,0
!*
MODOPT,LANB,20
EQSLV,SPAR
MXPAND,0, , ,0
LUMPM,0
PSTRES,0

!*
MODOPT,LANB,20,0,500, ,OFF
/STATUS,SOLU
SOLVE
FINISH
/POST1
SET,LIST
*get,freq1,mode,1,freq
*get,freq2,mode,2,freq
*get,freq3,mode,3,freq
```

7. APPENDIX 2. MACRO FOR BLADE FROM CONCEPTUAL DESIGN DOCUMENT

```
finish
/CLEAR,START
*abbr,doit,doit
/input,start71,ans,'C:\Program
Files\Ansys
Inc\v71\ANSYS\apdl\',,,,,,,,,,,,,,,,1
/PREP7
!*
! values of parameters
!analysis type at is 0 for modal and 1
for harmonic
at=1
blength=0.48
rootwidth=0.096
hroot=rootwidth/2
tipwidth=0.013
htip=tipwidth/2
bthick=0.0045
maryoung=1.76e11
marpoiss=0.3
mardens=7800
dampratio=1e-4
tipmass=11

!*
ET,1,SHELL93
ET,2,MASS21
!*
R,1,bthick, , , , , ,
!*
R,2,11,11,11,0,0,0
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,maryoung
MPDATA,PRXY,1,,marpoiss
MPTEMP,1,0
MPDATA,DENS,1,,mardens

K, ,0,0,0,
K, ,0,hroot,0,
K, ,0,-hroot,0,
K, ,blength,0,0,
K, ,blength,htip,0,
K, ,blength,-htip,0,

LSTR,       3,       6
LSTR,       6,       4
LSTR,       4,       1
LSTR,       1,       3
LSTR,       4,       5
LSTR,       5,       2
LSTR,       2,       1

AL,4,1,2,3

AL,3,5,6,7
aplot
ESIZE,hroot/4,0
amesh,1,2

TYPE,   2
MAT,       1
REAL,       2
KSEL,S,KP,,4
NSLK
*GET,massnode,NODE,0,NUM,MAX
NSEL,ALL
Ksel,all

E,massnode


DL,4,,all,0
DL,7,,all,0
!next two lines for harmonic analsysis
*if,at,eq,1,then
DL,4,,uz,1
DL,7,,uz,1
*endif

*if,at,eq,0,then
! modal analysis
FINISH
/SOL
ANTYPE,2
!*
ANTYPE,2
MSAVE,0
!*
MODOPT,LANB,20
EQSLV,SPAR
MXPAND,0, , ,0
LUMPM,0
PSTRES,0

!*
MODOPT,LANB,20,0,1000, ,OFF
/STATUS,SOLU
SOLVE
FINISH
/POST1
SET,LIST
*get,freq1,mode,1,freq
*get,freq2,mode,2,freq
*get,freq3,mode,3,freq
! end modal analysis
!*

*else
! harmonic analysis
FINISH
/SOL
!*
ANTYPE,3
!*
HROPT,FULL
HROUT,ON
LUMPM,0
DMPRAT,dampratio,
!*
EQSLV,FRONT,0,
PSTRES,0
!*
HARFRQ,0,100
NSUBST,100,
KBC,1
!*
/STATUS,SOLU
SOLVE
/POST26
FILE,'file','rst','.'
NUMVAR,200
SOLU,191,NCMIT
STORE,MERGE
PLCPLX,0
PRCPLX,1
FILLDATA,191,,,,1,1
REALVAR,191,191
!*
NSOL,2,massnode,U,Z, UZ_tip
nsol,3,421,u,z,uz-mid
STORE,MERGE
/GROPT,LOGX,On
/GROPT,LOGY,ON
XVAR,1
PLVAR,2,3


*endif
```

8. APPENDIX 3 – MACRO FOR SIMPLE NONLINEAR SOLUTION

```
finish
/CLEAR,START
*abbr,doit,doit
/input,start71,ans,'C:\Program
Files\Ansys
Inc\v71\ANSYS\apdl\',,,,,,,,,,,,,,,,1
/PREP7
!*
! values of parameters


blength=0.48
bthick=0.0045
rootwidth=0.096
tipwidth=0.013
hroot=rootwidth/2
htip=tipwidth/2
maryoung=1.76e11
marpoiss=0.3
mardens=7800
dampratio=1e-4
tipmass=11

!*
ET,1,SHELL93
ET,2,MASS21
!*
R,1,bthick, , , , , ,
!*
R,2,11,11,11,0,0,0
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,maryoung
MPDATA,PRXY,1,,marpoiss
MPTEMP,1,0
MPDATA,DENS,1,,mardens

K, ,0,0,0,
K, ,0,hroot,0,
K, ,0,-hroot,0,
K, ,blength,0,0,
K, ,blength,htip,0,
K, ,blength,-htip,0,

LSTR,       3,       6
LSTR,       6,       4
LSTR,       4,       1
LSTR,       1,       3
LSTR,       4,       5
LSTR,       5,       2
LSTR,       2,       1

AL,4,1,2,3

AL,3,5,6,7
aplot
ESIZE,hroot/4,0
amesh,1,2

TYPE,   2
MAT,       1
REAL,       2
KSEL,S,KP,,4
NSLK
*GET,massnode,NODE,0,NUM,MAX
NSEL,ALL
Ksel,all

E,massnode

DL,4,,all,0
DL,7,,all,0
FK,4,FZ,600
FINISH

!* nonlinear analysis to determine
curved form
/SOL
!*
ANTYPE,0
NLGEOM,1
NSUBST,10,0,0
/STATUS,SOLU
SOLVE
FINISH
/POST1
PLDISP,2
FINISH

!*Now apply curved form to model
/PREP7
UPCOORD,1,OFF
nplot
!* and force the tip back where it
came from
dk,4,uz,-.187


!* nonlinear analysis to determine
straight form
/SOL
!*
ANTYPE,0
NLGEOM,1
NSUBST,10,0,0
/STATUS,SOLU
SOLVE
FINISH
/POST1
PLDISP,2
FINISH

*go,:END
! modal analysis
FINISH
/SOL
ANTYPE,2
!*
ANTYPE,2
MSAVE,0
!*
MODOPT,LANB,20
EQSLV,SPAR
MXPAND,0, , ,0
LUMPM,0
PSTRES,0

!*
MODOPT,LANB,20,0,1000, ,OFF
/STATUS,SOLU
SOLVE
FINISH
/POST1
SET,LIST
*get,freq1,mode,1,freq
*get,freq2,mode,2,freq
*get,freq3,mode,3,freq
! end modal analysis
!*

:END
```

9. APPENDIX 4 – MACRO FOR MODAL ANALYSIS WITH PRESTRESS

```
finish
/CLEAR,START
*abbr,doit,doit
/input,start71,ans,'C:\Program
Files\Ansys
Inc\v71\ANSYS\apdl\',,,,,,,,,,,,,,,,1
/PREP7
!*
! values of parameters

blength=0.48
bthick=0.0045
rootwidth=0.096
tipwidth=0.013
hroot=rootwidth/2
htip=tipwidth/2
maryoung=1.76e11
marpoiss=0.3
mardens=7800
dampratio=1e-4
tipmass=11

!*
ET,1,SHELL93
ET,2,MASS21
!*
R,1,bthick, , , , , ,
!*
R,2,11,11,11,0,0,0
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,maryoung
MPDATA,PRXY,1,,marpoiss
MPTEMP,1,0
MPDATA,DENS,1,,mardens

K, ,0,0,0,
K, ,0,hroot,0,
K, ,0,-hroot,0,
K, ,blength,0,0,
K, ,blength,htip,0,
K, ,blength,-htip,0,

LSTR,       3,       6
LSTR,       6,       4
LSTR,       4,       1
LSTR,       1,       3
LSTR,       4,       5
LSTR,       5,       2
LSTR,       2,       1

AL,4,1,2,3

AL,3,5,6,7
aplot
ESIZE,hroot/4,0
amesh,1,2

TYPE,   2
MAT,       1
REAL,       2
KSEL,S,KP,,4
NSLK
*GET,massnode,NODE,0,NUM,MAX
NSEL,ALL
Ksel,all

E,massnode

DL,4,,all,0
DL,7,,all,0
FK,4,FZ,600

FINISH

!* nonlinear analysis to determine
curved form
/SOL
!*
ANTYPE,0
NLGEOM,1
NSUBST,10,0,0
/STATUS,SOLU
SOLVE
FINISH
/POST1
PLDISP,2
FINISH

!*Now apply curved form to model
/PREP7
UPCOORD,1,OFF
nplot
!* and force the tip back where it
came from
dk,4,uz,-.187


!* nonlinear analysis to resume
straight form
/SOL
!*
ANTYPE,0
NLGEOM,1
NSUBST,10,0,0
pstres,on
ematwrite,yes
/STATUS,SOLU
SOLVE
FINISH
/POST1
PLDISP,2
FINISH

! modal analysis
*MSG,UI
Starting modal analysis

/SOL
ANTYPE,2
upcoord,1.0,on
pstres,on
MSAVE,0
!*
MODOPT,LANB,20
EQSLV,SPAR
MXPAND,0, , ,0
LUMPM,0

!*
MODOPT,LANB,20,0,1000, ,OFF
/STATUS,SOLU
pSOLVE,triang
psolve,eiglanb
FINISH
*MSG,UI
Starting mode expansion

/solu
expass,on
psolve,eigexp
finish
*MSG,UI
Macro complete
```

10. APPENDIX 5. MACRO FOR MODEL WITH CLAMP, WIRE, AND MASS

```
finish
/CLEAR,START
*abbr,doit,doit
/input,start71,ans,'C:\Program
Files\Ansys
Inc\v71\ANSYS\apdl\',,,,,,,,,,,,,,,,1
/PREP7
!*
! values of parameters
!analysis type at is 0 for modal and 1
for harmonic
at=0
blength=0.48
bthick=0.00450
rootwidth=0.096
tipwidth=0.013
hroot=rootwidth/2
htip=tipwidth/2
maryoung=1.76e11
marpoiss=0.3
mardens=7800
wireyoung=2e11
wirepoiss=0,3
wiredens=7800
dampratio=1e-4
tipmass=.016
wiredia=7e-4*2
wirelen=0.54
testmass=11

!*
ET,1,SHELL93  !for the blade
ET,2,MASS21   !for the clamp and the
test mass
ET,3,LINK8    !for the wire
!*
!*

!*
R,1,bthick, , , , , , !for the blade
R,2,tipmass,tipmass,tipmass,0,0,0 !for
the clamp
R,3,testmass,testmass,testmass,0,0,0
!for the test mass
R,4,3.14*wiredia*wiredia/4, ,  !for
the wire

MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,maryoung
MPDATA,PRXY,1,,marpoiss
MPDATA,DENS,1,,mardens

MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,wireyoung
MPDATA,PRXY,2,,wirepoiss
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,2,,wiredens

K, ,0,0,0,
K, ,0,hroot,0,
K, ,0,-hroot,0,
K, ,blength,0,0,
K, ,blength,htip,0,
K, ,blength,-htip,0,
K, ,blength,0,-wirelen

LSTR,       3,       6
LSTR,       6,       4
LSTR,       4,       1
LSTR,       1,       3
LSTR,       4,       5
LSTR,       5,       2
LSTR,       2,       1

AL,4,1,2,3

AL,3,5,6,7
aplot
ESIZE,hroot/4,0
amesh,1,2

!clamp
TYPE,   2
MAT,       1
REAL,       2
kmesh,4

! add wire
lstr,4,7  !line 8

type,3
mat,2
real,4
lmesh,8

!add testmass
type,2
mat,1
real,3
kmesh,7

!constrain the wire
DL,8, ,UX,0
DL,8, ,UY,0

! encaster
DL,4,,all,0
DL,7,,all,0
!next two lines for harmonic analsysis
*if,at,eq,1,then
DL,4,,uz,1  !root displacement
DL,7,,uz,1
*endif

*if,at,eq,0,then
! modal analysis
FINISH
/SOL
ANTYPE,2
!*
ANTYPE,2
MSAVE,0
!*
MODOPT,LANB,20
EQSLV,SPAR
MXPAND,0, , ,0
LUMPM,0
PSTRES,0

!*
MODOPT,LANB,20,0,1000, ,OFF
/STATUS,SOLU
SOLVE
FINISH
/POST1
SET,LIST
*get,freq1,mode,1,freq
*get,freq2,mode,2,freq
*get,freq3,mode,3,freq
! end modal analysis
!*

*else
! harmonic analysis
FINISH
/SOL
!*
ANTYPE,3
!*
HROPT,FULL
HROUT,ON
LUMPM,0
```

```
DMPRAT,dampratio,
!*
EQSLV,FRONT,0,
PSTRES,0
!*
HARFRQ,0,100
NSUBST,100,
KBC,1
!*
/STATUS,SOLU
SOLVE
/POST26
FILE,'file','rst','.'
NUMVAR,200
SOLU,191,NCMIT
STORE,MERGE
PLCPLX,0
PRCPLX,1
FILLDATA,191,,,,1,1
REALVAR,191,191
!*
NSOL,2,92,U,Z, UZ_tip
nsol,3,421,u,z,uz-mid
STORE,MERGE
/GROPT,LOGX,On
/GROPT,LOGY,ON
XVAR,1
PLVAR,2,3




*endif
```

11. APPENDIX 6 NOTES ON Q, PHI ETC BY NORNA ROBERTSON

7/8 Jan 2004

<u>Simple harmonic motion, loss and damping.</u>

Consider the following equation.

$$m\ddot{x} + b\dot{x} + k_0 x = F \qquad (1)$$

This is a simple oscillator with damping.

Now "damping ratio", ζ is defined as the ratio between actual damping and critical damping. Critical damping is given by quality factor Q = ½.

Thus the damping ratio can be written as

$$\zeta = 1/(2Q) \qquad (2)$$

We also note that Q = $\omega_0 m/b$, where ω_0^2 = k_0/m (3)

Now when we talk about loss, ϕ, we are thinking about a spring constant which has a small imaginary term, i.e.

$$k = k_0(1 + i\phi) \qquad (4)$$

We can rewrite our simple oscillator equation as

$$m\ddot{x} + kx = F \qquad (5)$$

and using (4) we get

$$m\ddot{x} + k_0(1 + i\phi)x = F \qquad (6)$$

Comparing (6) and (1), and using $\dot{x} = i\omega x$ we find the equivalence

$k_0\phi = b\omega$, and hence

$$b = \frac{k_0\phi}{\omega} \qquad (7)$$

Comparing this to (3) gives us the relationship between Q and ϕ. Note that ϕ in general could be a function of frequency. We find

$$Q = 1/\ \phi(\omega_0) \qquad (8)$$

Hence the damping ratio could be expressed as
$\zeta = \phi(\omega_0) / 2$
If we have structural damping, ϕ is constant with frequency and so $\phi(\omega_0) = \phi$ and the damping constant is given by

$$\zeta = \phi / 2 \qquad (9)$$

Now linking to the ANSYS formula, comparing it to the simple oscillator as expressed in (1) we have the equivalence

$$b \equiv \beta_c K \qquad (10)$$

By comparing (7) and (10), noting that k_0 is equivalent to K, and using (9) we see that

$$\beta_c = \frac{\phi}{\omega} = \frac{\phi}{2\pi f} = \frac{\zeta}{\pi f}$$

The full definition is Q = 2pi times energy stored in system divided by energy lost per cycle. it is also often defined in terms of the width of a resonance. If you consider power resonance curve (e.g. a graph of power supplied by a driving force to a driven oscillator plotted against frequency) then Q is the ratio of peak frequency to full width at half maximum (halfway down down the curve). If on the other hand you plot a displacement/amplitude curve against freq., Q is given by peak freq. divided by full width at point 1/root2 from peak (i.e. 3 db down). Power involves displacement squared - hence difference in the curves. And finally for large Q values the peak height on a displacement or amplitude curve is approx Q times the value off resonance.

And from the ANSYS manuals:

Equation 15.20.

$$[C] = \alpha[M] + (\beta + \beta_c)[K] + \sum_{j=1}^{NMAT} \beta_j[K_j] + \sum_{k=1}^{NEL} [C_k] + [C_\xi]$$

where:

[C] = structure damping matrix
α = constant mass matrix multiplier (input on **ALPHAD** command)
[M] = structure mass matrix
β = constant stiffness matrix multiplier (input on **BETAD** command)
β_c = variable stiffness matrix multiplier (see Equation 15.23)
[K] = structure stiffness matrix
NMAT = number of materials with DAMP input
β_j = constant stiffness matrix multiplier for material j (input as DAMP on **MP**
K_j = portion of structure stiffness matrix based on material j
NEL = number of elements with specified damping
C_k = element damping matrix
C_ξ = frequency-dependent damping matrix (see below)

Equation 15.23.

$$\beta_c = \frac{\xi}{\pi f}$$

where:
ξ = constant damping ratio (input on **DMPRAT** command)
f = frequency in the range between f_B and f_E
f_B = beginning frequency (input as FREQB,**HARFRQ** command)
f_E = end frequency (input as FREQE,**HARFRQ** command)