LASER INTERFEROMETER GRAVITATIONAL WAVE OBSERVATORY

LIGO Laboratory / LIGO Scientific Collaboration

LIGO-T030273-04-D *ADVANCED LIGO* 3rd November 2003

Case Study: Using ANSYS to predict the Lowest Flexural Internal Mode frequency of the MC and RM upper Blades

R.Jones, C.I.Torrie

Distribution of this document:
LIGO Science Collaboration

This is an internal working note
of the LIGO Project.

California Institute of Technology
LIGO Project – MS 18-34
1200 E. California Blvd.
Pasadena, CA 91125
Phone (626) 395-2129
Fax (626) 304-9834
E-mail: info@ligo.caltech.edu

Institute for Gravitational Research
University of Glasgow
Kelvin Building
Glasgow G12 8QQ
Phone: +44 (0)141 330 3340
Fax: +44 (0)141 330 6833
Web: www.physics.gla.ac.uk/gwg

Massachusetts Institute of Technology
LIGO Project – NW17-161
175 Albany St
Cambridge, MA 02139
Phone (617) 253-4824
Fax (617) 253-7014
E-mail: info@ligo.mit.edu

LIGO Hanford Observatory
P.O. Box 1970
Mail Stop S9-02
Richland WA 99352
Phone 509-372-8106
Fax 509-372-8137

LIGO Livingston Observatory
P.O. Box 940
Livingston, LA 70754
Phone 225-686-3100
Fax 225-686-7189

http://www.ligo.caltech.edu/

Revision 00: Limited release
Revision 01: Review & comments (CIT)
Revision 02: Further updates (CIT and RAJ)
Revision 03: Latest update based on comments from CIT (RAJ)
Revision 04: based on comments from CIT

1.0 Introduction

This document summarises the steps taken to build a finite element model (FEM) that would help us to estimate lowest flexural mode frequency (f_{INT}) of specific blade designs (i.e. MC and RM upper blades). The work, carried out during the visit of Russell Jones to the California Institute for Technology in November 2003, was performed as an extension of the ANSYS tutorial[1] created by Eoin Elliffe on the deflection analysis of cantilever blades.

The main purpose of the work was to familiarise the authors with a basic approach to modal analysis in ANSYS, prior to considering the altogether more difficult prospect of the frequency analyses on the structures for the HAM and BSC suspensions[2]. This study was of further use to us because the assessment of the lowest flexural mode frequency (f_{INT}) of these blades was relevant to the review of the GEO600 blade ECD design[3], considered for application in the MC and RM Controls Prototypes.

2.0 Experimental Measurement of "f_{INT}"

The Mode Cleaner Upper Blade (D020205)[4] was used as the subject in the following test setup:

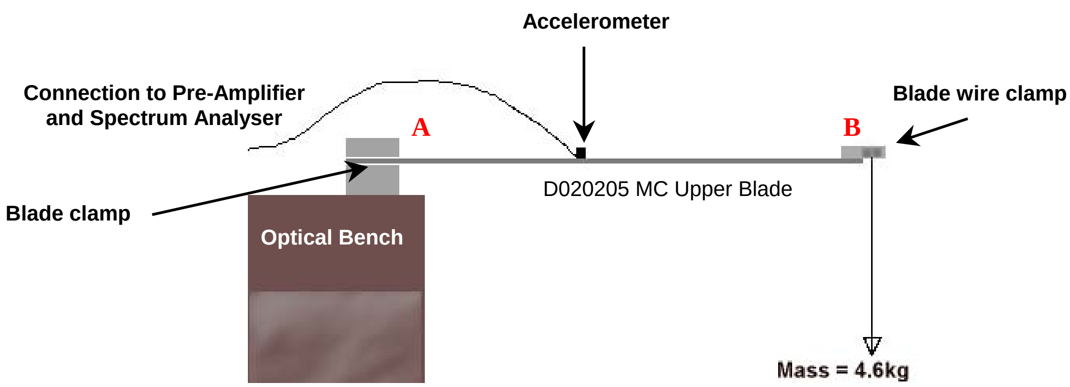


Figure (i): The Experimental Test set-up

Please note that in every test the blade was fully clamped at thick end ("**A**" in figure (i)), and also that a safety cover was constructed above the blade.
The experimental results are shown in TABLE I below:

No.	DESCRIPTION OF TEST	1st Internal Mode (Hz)
1	Rigidly clamped flat at the tip of the blade (i.e. No mass suspended from the tip of the blade)	(117*) 118
2	Loaded flat with 4.6kg, without the use of a blade wire clamp	90
3	Loaded flat with 4.6kg, inc. blade wire clamp (12g) (*i.e. As in the diagram above*)	84
4	Loaded flat with 4.6kg, inc. blade wire clamp (12g), and a spacer (simulating the mass of a magnet/bolt = 7g) placed at midpoint of blade of blade	77.8
5	Loaded flat with 4.6kg - blade wire clamp used + 5g at the tip (**B** in figure (i))	82
6	Loaded flat with 4.6kg - blade wire clamp used + 10g at the tip (**B** in figure (i))	80
7	Loaded flat with 4.6kg - blade wire clamp used + 20g at the tip (**B** in figure (i))	77

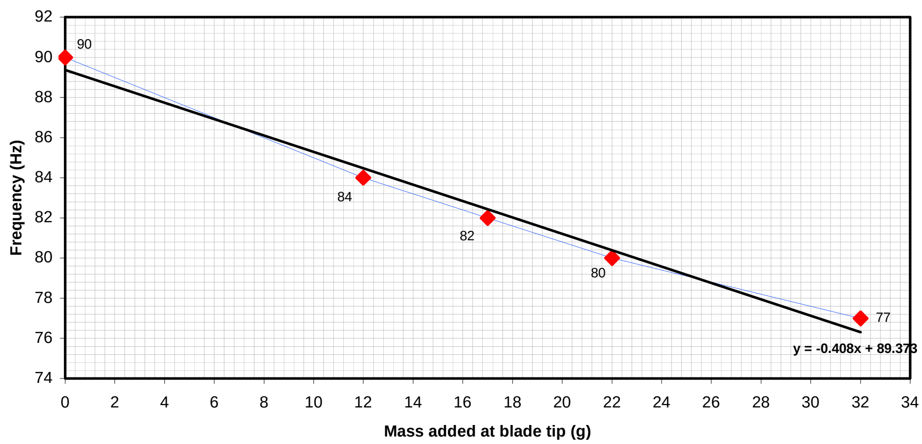
TABLE I

[1] T030056-00-D - Ansys 7.0 tutorial in non-linear blade deflection analysis (E.Elliffe)
[2] LIGO-T030278 - Frequency Analysis of the MC Structure
[3] LIGO-T040004 – Review of the GEO 600 Blade Eddy Current Damper
[4] LIGO-D020205 MC Upper Blade

In Table I above, #1 was performed to mimic an early attempt to model the blade in Ansys. In this model we used a flat blade and applied displacement constraints to both the base and tip of the blade (i.e. zero displacement of certain nodes in all degrees of freedom). Experimentally, it proved difficult to clamp the blade flat at the tip and this was the main source of error. This is evident in #1 where our initial measurement of f_{INT} was 117Hz but when the clamp was tightened further, f_{INT} increased to 118Hz. This devalues the fact that the result was within 5% of the 123Hz Ansys had predicted.

In the experiment, #5, #6 and #7 were taken to assess the effect of adding mass to the tip of the blade. These results have been graphed in figure (ii) below, and indicate a linear relationship between the internal mode and the mass of the clamp as the clamp mass is varied.

Figure (ii) **: Effect on the First Internal Mode Frequency of adding mass**



As can be seen for the above graph of 1st internal frequency against the mass added to the tip of the blade the relationship is linear. The blade used in this experiment weighed ~ 80 g. Summary

An additional 27 % of mass, 22 g, the resonance changes by 11 %.
An additional 15 % of mass (typical stainless steel clamp, although GEO used titanium) the resonance changes by 7 %.

3.0 Development of an FEA Model

One approach is considered using an imported 3-D solid file, selecting an appropriate element type and using the automatic mesh tool

3.1Recipe of Steps

In order to minimise the time involved in building and running a model, we used SolidWorks to create the geometry then export it using a translator .SAT file (version 7), which was in turn imported into ANSYS. It is vital that 'metres' are selected as units when performing an export from SolidWorks.

Once the file had been imported, the following parameters were employed in each model:

ELEMENT TYPE:	Solid 92
Material:	Maraging Steel
Density:	8000 kg/m^3
Young's Modulus:	186 GPa
Poisson's ratio:	0.3
MESH TYPE:	Smart size 8-10

3.2 Process of Refinement

We assumed an iterative approach to the FEA, consistently questioning our results as we attempted to converge upon a model that gave results closely matching those from experiments. Our observations are discussed in the following section, and the results of the analyses are listed in 'Table II' alongside the relevant experimental data.

The first issue addressed was that of the blade geometry (i.e. a flat blade or curved blade in the FE model) that would achieve the best fit to our experimental results. This could only be done by simultaneously considering how to address the experimental 4.6kg load applied to the tip of the blade in order to make it flat. This process is illustrated in 'Table II', #3- #7.

It was decided that it was more suitable to model a flat blade in ANSYS, given the finite amount of time available. Following this decision, we concentrated on how best to apply displacement constraints[5] to the model (Table II, #9 - #13) and in doing so were again forced to question the input geometry. Discussions with Larry Jones regarding the significance of the flat clamped area, led to FE analysis of the blade both with and without this area, whilst simultaneously varying the number of fully constrained nodes (Table II, #9 - #13).

Further detailing of our FE model concentrated on inclusion of the mass added by the blade wire clamp and also the magnet. In order to consider accurate geometries for the magnet and wire clamp, it was necessary to 'join' multiple parts in SolidWorks to create a single unified part prior to exporting a .SAT file.

[5] Appendix 1: Selection of screen captures showing iterations of fixed nodes

4.0 Table of Comparison

	Geometry	Analysis	Description of Loadings and Constraints	1st Frequency Hz	within "…" %
EXPERIMENT					
1	-	-	Fully clamped at base / clamped flat at tip	118	-
2	-	-	Fully clamped at base / tip loaded with 4.6kg, but no blade wire clamp	**90**	*compare with 3 - 13
ANSYS FEA					
3	full MC, flat	modal	All nodes selected at base, 3 nodes picked at tip	123	36.67%
4	full MC, flat	modal	All nodes selected at base, all nodes picked at tip	128	42.22%
5	full MC, flat	modal	All nodes selected at base, centre node at tip loaded with 4.6kg	28.5 (125)	(38.89%)
6	full MC, curved	modal	All nodes selected at base, centre node at tip loaded with 4.6kg	29.5 (101.7)	(13.00%)
7	full MC, curved	modal	All nodes selected at base, all nodes picked at tip	229	154.40%
8	full MC, curved	**Static**	Curved MC upper blade loaded with 4.6kg: deflection = 157mm (within 10% of the specified 143mm)	-	-
9	full MC, flat	modal	4 + 4 at base (top + bottom at corners), 1 central node at tip (on top surface)	97	7.77%
10	sawn off, flat	modal	2 + 2 at base (top + bottom), 1 central node at tip (on top surface)	85	5.55%
11	sawn off, flat	modal	5 + 5 at base (top + bottom), 1 central node at tip (on top surface)	97	7.77%
12	sawn off, flat	modal	All nodes at base, 1 central node at tip (on top surface)	103	14.44%
13	sawn off, flat	modal	3 + 3 at base (top + bottom), 1 central node at tip (on top surface)	**94**	4.33%
EXPERIMENT					
14	-	-	Fully clamped at base / tip loaded with 4.6kg, plus blade wire clamp (12g)	**84**	*compare with 15,16
ANSYS					
15	full MC, flat	modal	4 + 4 at base (top + bottom at corners), 1 central node at tip (on top surface)	92.3	9.88%
16	sawn off, flat	modal	3 + 3 at base (top + bottom), 1 central node at tip (on top surface)	**88**	4.76%
EXPERIMENT					
17	-		Fully clamped at base / tip loaded with 4.6kg, plus blade wire clamp (12g) + spacer at midpoint of blade	**78**	*compare with 18
ANSYS					
18	sawn off, flat	modal	3 + 3 at base (top + bottom), 1 central node at tip (on top surface)	**83**	7.79%

TABLE II

KEY:
full MC: MC upper blade including clamped area at the base
sawn off: MC upper blade with clamped area removed prior to export from SolidWorks

4.1 Observations

4.1.1 Flat or curved

Comparing (Table II) #2, with #4 - #7, it is clear that the best approximation to experiment is achieved by using a flat blade as the subject of the FE model under our conditions.

The analyses of the flat and curved blades where we applied a 4 kg static force to a central node at the blade tip wielded peculiar results (#5 and #6 above), and this is illustrated in Appendix 2[6]. In both cases we were finding a first frequency at around 29Hz, but this did not appear to be f_{INT}. A static analysis was performed to check that the applied load was being recognized, and this proved to be the case (#8 above). However it was clear that although the application of a 4.6kg force 'looked' like the best approximation to the reality case, it was inappropriate.

4.1.2 Consideration of Claming

Although there are many variables affecting this problem, we chose to concentrate on the issue of clamping as we refined our model. Again looking at Table II, #9 - #13 describe steps taken to optimize the number and position of the displacement constraints. It is clear that increasing the number of fully constrained nodes also increases the lowest flexural mode frequency (f_{INT}). Figure (iv) shows our final iteration on fixed nodes, which came within 5% of both the experimental value of 90Hz and the theoretical prediction[7] of 90Hz.

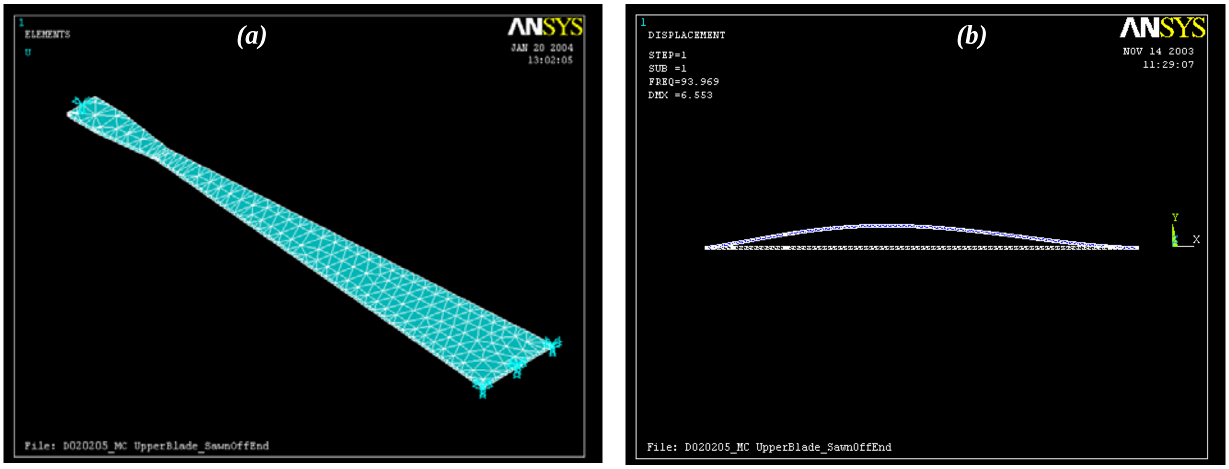


Figure (iv): Pictures related to Table II, #13
(a) Screenshot from ANSYS showing displacement constraints
(b) Graphical representation of lowest flexural mode frequency (f_{INT}), 94 Hz

4.1.3 Adding Mass in the model

As stated previously, in order to introduce the clamp and magnet to the model accurately, we chose to 'join' components in SolidWorks prior to the exporting of a SAT file. Two examples of joined assemblies are shown in figure (v) below.

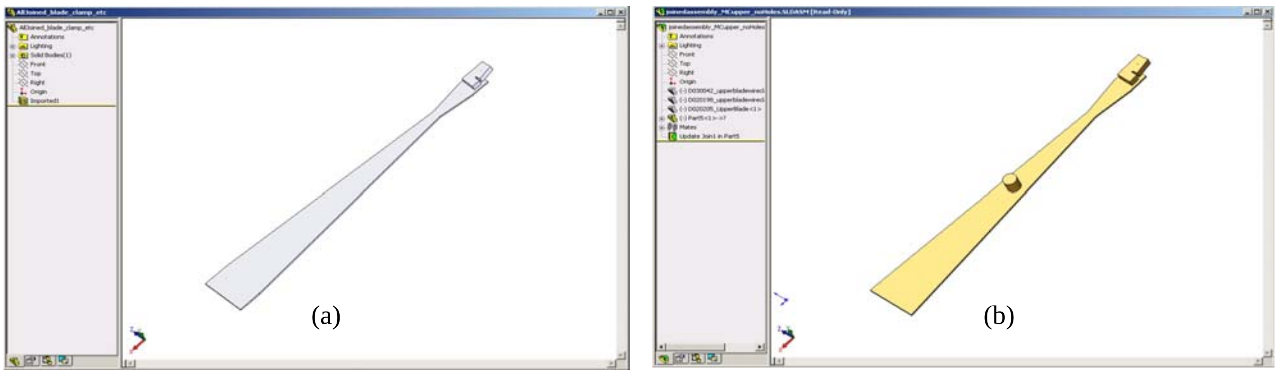


Figure (v): Screen captures from SolidWorks
(a) Full MC blade joined with blade wire clamp, Table II #15
(b) MC blade without clamping region, joined with blade wire clamp and magnet, Table II #18

[6] Appendix 3: Including a 4.6kg static load in Modal Analysis (Table II #5 and #6)
[7] EXCEL document: "Bladespec_MC" (M. Plissi, N. Robertson, C.I.Torrie)

When the blade clamp was introduced, the fixed node at the tip of the blade was offset to coincide with the approximate position of the wire break off point. This position is suggested in figure (vi) below.

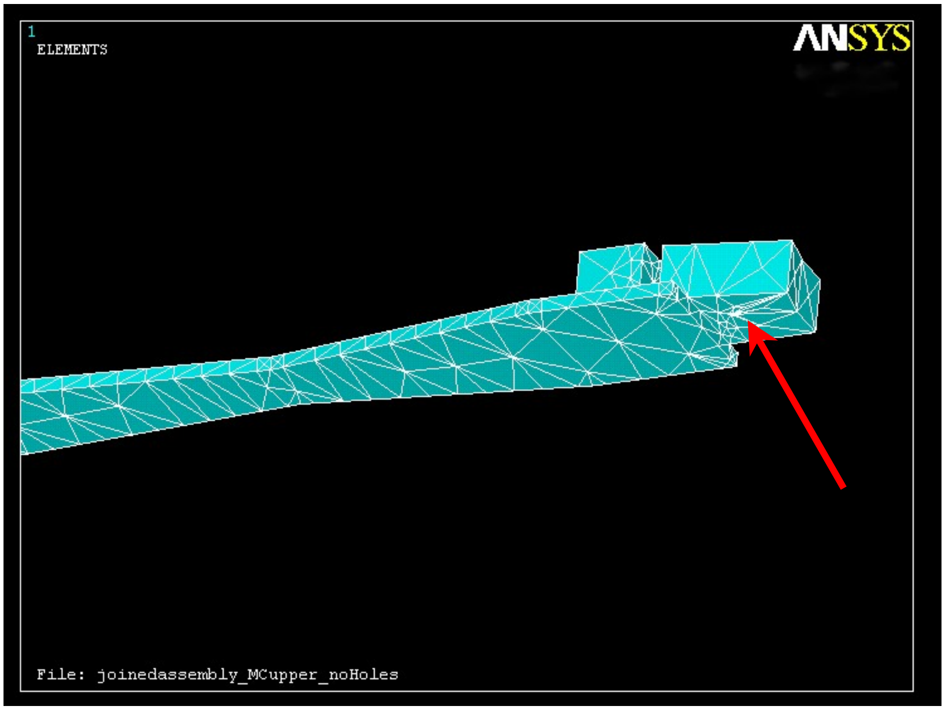


Figure (vi): Screen capture showing a meshed MC blade and blade wire clamp; the red arrow points to the position of the selected fixed node

This highlights a source of error in our FEA; that of consistent selection of fixed node position. The use of Solid92 element type and the 'Smart Mesh' command creates irregular element geometries and in some places a high concentration of nodes (as can be seen in figure (vi) around the region of the groove in the wire clamp). This made consistency of selection difficult from model to model, as mesh geometries varied significantly using our approach.

4.2 Proposed model

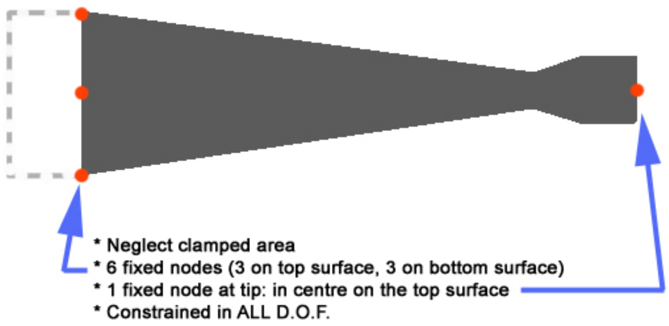
We propose that the following model be used as a template during the early stages of analyses on blades of a similar size to the Mode cleaner Upper blade.

1. Export SAT file (version 7, units = metres) from SolidWorks and import into ANSYS.

2. Use the following input parameters:

ELEMENT TYPE:	Solid 92
Material:	Maraging Steel
Density:	8000 kg/m^3
Young's Modulus:	186 GPa
Poisson's ratio:	0.3
MESH TYPE:	Smart size 8-10

3. Apply the constraints as depicted below:



4.3 Model Validation

With our model we have been able to predict the internal mode to within 10 % of both results from experiment (Table I) and those calculated from blade theory. These results are summarized in the following table:

	THEORY (Hz)	EXPERIMENT (Hz)	FEA (Hz)	Within…
Load = 4.6kg, no wire clamp, no magnet	90	90	94	4.3%
Load = 4.6kg, inc. wire clamp, no magnet	-	84	88	4.76%
Load = 4.6kg, inc. wire clamp and magnet	-	78	83	7.79%

Table III: Summary of Results on MC Upper blade

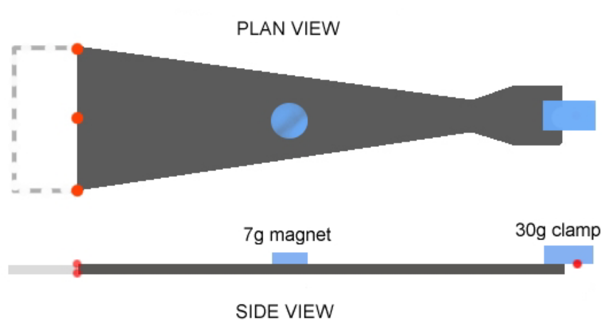
In order to validate the proposed model, it was applied to a Basic Tapered Section and the 'Original' GEO 600 Upper Blade to assess its accuracy.

The first flexural internal mode of the Basic Tapered Section[8] was predicted at ~60Hz using an early FEA model in IDEAS. ANSYS predicted the first internal mode to be 55 Hz using the proposed template which is within 10% .

The first internal mode of the 'Original' GEO 600 Upper Blade[9] was measured at 55Hz, and ANSYS predicted 51 Hz, again well within 10%.

5.0 Application of the Model

The ANSYS model was applied to the RM upper blade design incorporating both a magnet and clamp (as shown below):



[8] (Thesis) Development of Suspensions for the GEO600 Gravitational Wave Detector, p.135, C.I.Torrie
[9] (Thesis) Development of Suspensions for the GEO600 Gravitational Wave Detector, p.131-2, C.I.Torrie

Applying this model to the above case resulted in a first internal mode of 123 Hz. This compares to an internal mode of the RM upper blade, as calculated from blade theory[10], of 139 Hz. However, this value *did not* incorporate additional mass representing the wire clamp or magnet. Referring back to Table I (and figure (i)) in Section 2.0, it can be seen that adding mass reduces the lowest flexural mode frequency (f_{INT}), so it is understandable that the FE model predicted a lower frequency than theory.

6.0 Conclusions: FEA Model

Using an iterative process with particular focus on the application of displacement constraints, we have created a model that predicts the first internal mode frequency of the MC upper blade to within 10 % of both the experimental measurement and the theoretical prediction. The model has been tested on several other examples of a similar size to good effect, most notably the RM upper blade.

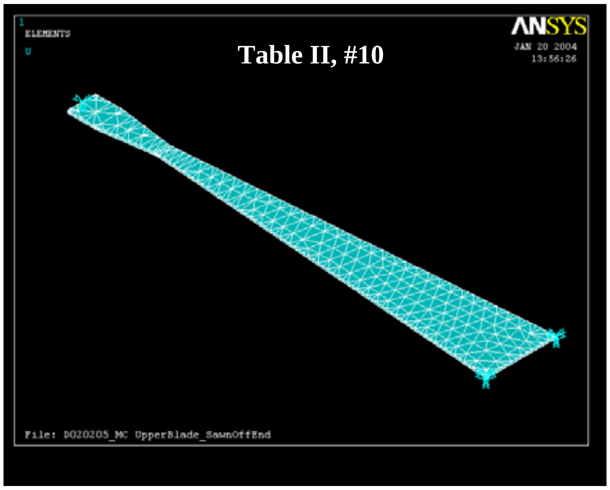
It is clear from this study that the mass contributed by a wire clamp (and also a magnet for blade ECD) has a significant affect on the first internal mode of a blade. In our case study, an additional 15 % of mass (associated with the inclusion of a typical stainless steel clamp) reduces the internal resonance by 7 %; and an additional 27 % of mass reduces it by 11 %.

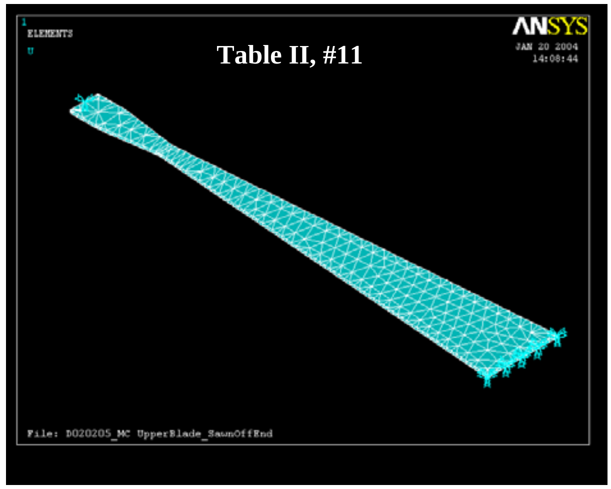
References

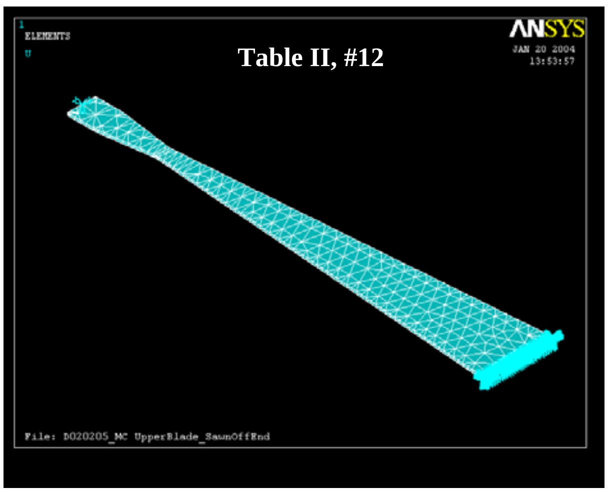
T030056-00-D - Ansys 7.0 tutorial in non-linear blade deflection analysis (E.Elliffe)
Development of Suspensions for the GEO600 Gravitational Wave Detector, C.I.Torrie (Thesis)
EXCEL document: "Bladespec_MC" (M. Plissi, N. Robertson, C.I.Torrie)
EXCEL document: "Bladespec_RM" (M. Plissi, N. Robertson, C.I.Torrie)
T030107-00-D Cantilever blade analysis for Advanced LIGO (M. V. Plissi)

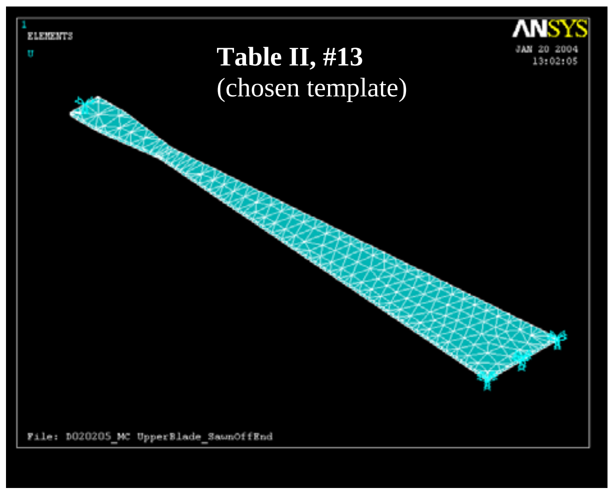
[10] EXCEL document: "Bladespec_RM" (M. Plissi, N. Robertson, C.I.Torrie)

Appendix 1: Selection of screen captures showing iterations of fixed nodes



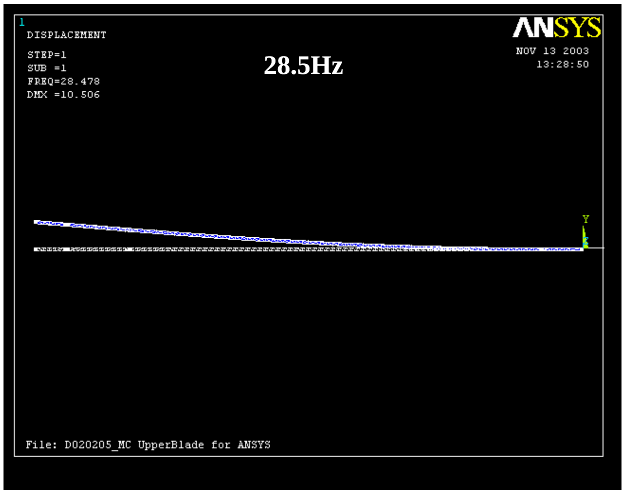


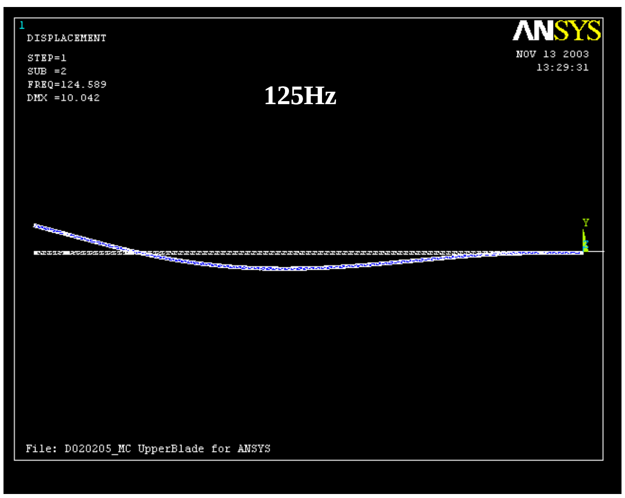




Appendix 2: Including a 4.6kg static load in Modal Analysis (Table II #5 and #6)

Table II, #5





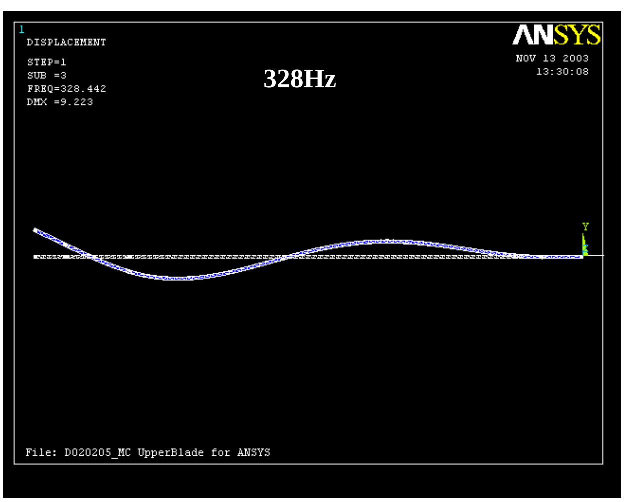


Table II, #6