

Subject: RE: FEM for a one element structure
Date: Monday, July 25, 2011 1:19 PM
From: Jack Castro <jack.castro@mscsoftware.com>
To: "Wieseman, Carol d." <carol.d.wieseman@nasa.gov>
Conversation: FEM for a one element structure

Hi Carol,

Let me know if you need more input from me on the Beam element questions. Just to close the loop on the one element model you created, this is what you have set up:

Grid A (22) Coordinates: .29513 .10417 0.
Grid B (23) Coordinates: .31591 .15476 0.

Both the placement and displacement coordinate systems of the above grids are in the default "Basic" Nastran coordinate system. The element shear center is offset from the above coordinates in the local displacement coordinate system of each grid. However, since there are no local displacement systems input, the offsets in this case are in the Basic Coordinate System relative to Grid A and Grid B.

The shear center offset from Grid A is: 0.00534 -0.00219 -
0.00091925
The shear center offset from Grid B is: 0.00654 -0.00268 -0.0018564

The PBEAM entry shows no additional offsets of the neutral axis or mass center. Therefore, both the neutral axis and mass center lie on the shear center line in your example.

The section properties input are:

Area: 0.0092645
I1: 0.000100589
I2: 2.60238E-06
I12: -1.44405E-06
J: 0.00010319

Your orientation vector on the CBEAM is: X=1.0, Y=0.0, Z=0.0. These XYZ directions are parallel to the Grid point A displacement coordinate system. However, since there is no displacement coordinate systems input, this orientation vector is parallel to the Basic Coordinate System X axis in this case.

Therefore, Plane 1, which defines the plane of bending for the I1 inertia, is the plane formed by the Shear Center Line, and the Orientation vector parallel to the Basic X Axis emanating from shear center end A. (See graphic below)

I cannot recall what element from Boucke's model you are reporducing, but if you can remind me, then I will compare your inputs versus that element in his model and see if it makes sense.

Have a good day,
Jack

...
Jack Castro
MSC Software Corporation
Phone: 425-891-3177

Subject: Re: FEM for a one element structure
Date: Monday, July 25, 2011 3:16 PM
From: Wieseman, Carol d. <carol.d.wieseman@nasa.gov>
To: Jack Castro <jack.castro@mscsoftware.com>
Conversation: FEM for a one element structure

Looks like I sent the wrong file and the wrong PBEAM card.

With the offset the shear centers should be at at point 3022 and 3023 at the two ends.

```
GRID*,3022,,0.30047,0.10198,+  
*,-0.00091925  
GRID*,3023,,0.32245,0.15208,+  
*,-0.0018564
```

The locations of the bending is at Grids 10xx,
Locations of mass center are grids 20xx']

I'm not convinced I am calculating them properly but I put the grid cards in the file specifically so others could check,
Here is the correct PBEAM card...

```
PBEAM*,24,24,0.0092645,0.000100589,+  
*,2.60238e-06,-1.44405e-06,0.00010319,0.,+  
*,0.0,0.0,0.0,0.0,+  
*,0.0,0.0,0.0,0.0,+  
*,9.3637,9.3637,,+,  
*,,,,,+,  
*,0.0283175,0.000485133,0.0226755,0.0,+  
*,0.0112313,0.0,0.0125929,0.0
```

Haven't tried to run it yet

From original file from Boucke

```
Node => Wing Node 2; 0.29513·m; 0·m; 0.10417·m;  
Node => B Node 2; 0.31086·m; 0.00071382·m; 0.097707·m;  
Node => C Node 2; 0.32667·m; 0.00080760·m; 0.091215·m;  
Node => D Node 2; 0.30047·m; 0.00091925·m; 0.10198·m;
```

```
Node => Wing Node 3; 0.31591·m; 0·m; 0.15476·m;  
Node => B Node 3; 0.33410·m; 0.0017307·m; 0.14729·m;  
Node => C Node 3; 0.34343·m; 0.0017147·m; 0.14346·m;  
Node => D Node 3; 0.32245·m; 0.0018564·m; 0.15208·m;
```

```
-----  
-- Wing Element 3  
-- Span from 0.10417·m to 0.15476·m  
-- Length 0.054694·m  
-- Mass 4.0132·kg
```

```
Elastic_Material => Mat 3  
Rho => 8162.2·kg/m^3;  
-- incl. 2.485kg/m fuer kabel  
E_Mod => 186110000000·kg/m·s^2  
G_Mod => 6224300000·kg/m·s^2  
kappa => 9.3637;
```

```
Cylindric_Module => Wing Beam 3  
Node_1 => Wing Node 2  
Node_2 => Wing Node 3  
Material => Mat 3  
Area => 0.0092645·m^2  
I_Yy => 0.00010061·m^4  
I_Zz => 0.0000025811·m^4  
Alpha => -0.014733  
J => 0.00010319·m^4
```

The inertias have been modified by rotation alpha. Coordinate system is different xnastran=x, ynastran=-z, znastran=y

On 7/25/11 1:19 PM, "Jack Castro" <jack.castro@mscsoftware.com> wrote:

Hi Carol,

Let me know if you need more input from me on the Beam element questions. Just to close the loop on the one element model you created, this is what you have set up:

```
Grid A (22) Coordinates: .29513 .10417 0.  
Grid B (23) Coordinates: .31591 .15476 0.
```

Both the placement and displacement coordinate systems of the above grids are in the default "Basic" Nastran coordinate system. The element shear center is offset from the above coordinates in the local displacement coordinate system of each grid. However, since there are no local displacement systems input, the offsets in this case are in the Basic Coordinate System relative to Grid A and Grid B.

The shear center offset from Grid A is: 0.00534 -0.00219 -
0.00091925

The shear center offset from Grid B is: 0.00654 -0.00268 -0.0018564

The PBEAM entry shows no additional offsets of the neutral axis or mass center. Therefore, both the neutral axis and mass center lie on the shear center line in your example.

The section properties input are:

Area: 0.0092645
I1: 0.000100589
I2: 2.60238E-06
I12: -1.44405E-06
J: 0.00010319

Your orientation vector on the CBEAM is: X=1.0, Y=0.0, Z=0.0. These XYZ directions are parallel to the Grid point A displacement coordinate system. However, since there is no displacement coordinate systems input, this orientation vector is parallel to the Basic Coordinate System X axis in this case.

Therefore, Plane 1, which defines the plane of bending for the I1 inertia, is the plane formed by the Shear Center Line, and the Orientation vector parallel to the Basic X Axis emanating from shear center end A. (See graphic below)

I cannot recall what element from Boucke's model you are reproducing, but if you can remind me, then I will compare your inputs versus that element in his model and see if it makes sense.

Have a good day,
Jack

...
Jack Castro
MSC Software Corporation
Phone: 425-891-3177

-----Original Message-----

From: Wieseman, Carol D. (LARC-D308)
[mailto:carol.d.wieseman@nasa.gov]
Sent: Friday, July 22, 2011 4:41 PM
To: Jack Castro
Subject: Re: FEM for a one element structure

The other difference is that it costs more to run MSC Nastran than MD Nastran for the exact same problem.

I guess that is partially why I don't use Patran to write the BDF file because I do or should have the element.

Didn't have the SPC either.

Here is the latest. Still an error...

On 7/22/11 6:32 PM, "Jack Castro" <jack.castro@mscsoftware.com> wrote:

> Hi Carol,

>
> That is a strange error message. Anyway, the error occurs because
the
> model does not contain any elements. I think when you had Patran
write
> out the model, perhaps the properties were not assigned so Patran
> skipped output the element.
>
> By the way, You should be fine running MD Nastran. There is much
> confusion over what MD Nastran is. MD Nastran is EXACTLY MSC
Nastran,
> but with SOL 400 and SOL 700 added. Plus, SOL 101 is extended to
allow
> nonlinear surface contact. However, if you run MSC Nastran and MD
> Nastran on exactly the same files, you get exactly the same
solutions
> because when you run MD Nastran, you are running the identical
code to
> MSC Nastran. So, the only differences in MD Nastran are the
extensions
> over MSC Nastran. If you attempt to run a model in MSC Nastran
that
> draws on the extensions in MD Nastran, you just get an
authorization
> fatal error.
>
> Sorry for that confusion! :)
>
> Jack
>
> ...
> Jack Castro
> MSC Software Corporation
> Phone: 425-891-3177
>
>
>
> -----Original Message-----
> From: Wieseman, Carol D. (LARC-D308)
> [mailto:carol.d.wieseman@nasa.gov]
> Sent: Friday, July 22, 2011 2:47 PM
> To: Jack Castro
> Subject: FEM for a one element structure
>
> I get fatal error when I try to run it. Can you help me out with
this?
>
> I created the file within Patran. The last 3 cases are the loads
in
> all
> 3 directions. The first subcase is some other loading.
>
> I get a fatal error. I happen to be running MD nastran - because
it
> got installed instead of MSC nastran which will changes on Monday.

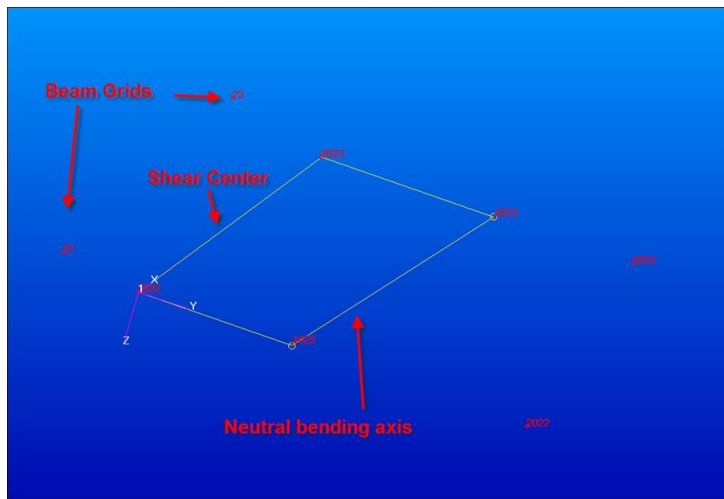
--

Carol D. Wiese

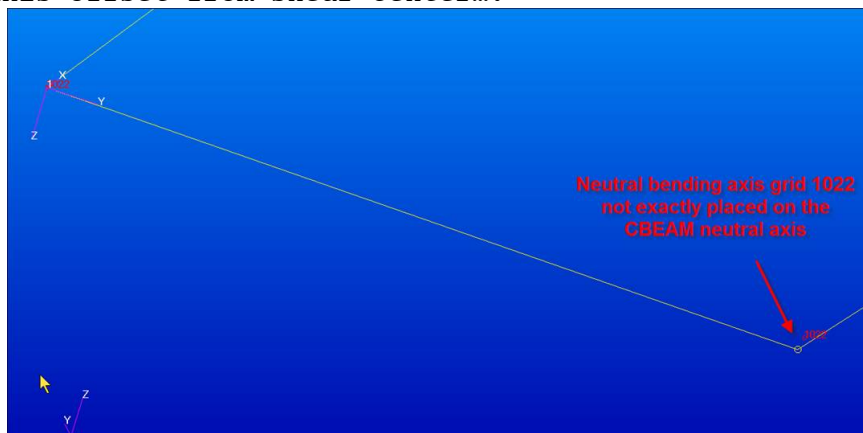
Subject: RE: FEM for a one element structure
Date: Monday, July 25, 2011 4:24 PM
From: Jack Castro <jack.castro@mscsoftware.com>
To: "Wieseman, Carol d." <carol.d.wieseman@nasa.gov>
Conversation: FEM for a one element structure

Carol,

The shear center offsets are correctly placed at nodes 3022 and 3023. The "bending axis" location is pretty close, but not exactly at the grid locations. We cannot get it exactly at the grid locations since Nastran only allows the bending axis to be defined in two dimensions relative to the shear center. If we look at the location of the grids in the plot below, we can see that from a "far field" perspective, the bending grids appear to be placed properly.



However, if I zoom in on the bending grids, then we see there begins to be some separation due to the two-dimensional neutral bending axis offset from shear center...



I have not computed how far off we are yet, but you can see I have created a coordinate system with X aligned with the beam X axis and Y aligned with the Beam Y axis. We can define the bending and mass grids relative to this coordinate system and then transform back to Basic. Then we can have Robert rerun his single beam model with the exact same bending and mass grid locations as Nastran used. I need

to leave now for the rest of the day to go to a customer meeting so I will get back to this tomorrow. Tomorrow I will be at Boeing all day and will try to catch up with Kumar as well.

Jack

...

Jack Castro

MSC Software Corporation

Phone: 425-891-3177

Subject: Re: More weight debugging

Date: Friday, July 22, 2011 5:11 PM

From: Alexander Boucke <Alexander.Boucke@itam-gmbh.de>

To: "Wieseman, Carol d." <carol.d.wieseman@nasa.gov>

Cc: Jack Castro <jack.castro@mscsoftware.com>, "Avery, Christopher R" <christopher.r.avery@boeing.com>

Carol,

sorry for getting back to you so late. The idea of separating out an element is basically a sound one. But I don't think the results will be meaningful when we get the modes of just a single element of 5cm length, but describing a 'beam' with full chord width. It would be better to take the sectional data of this element and create a 1.5m long beam from it using anything between 10 and 50 elements along it's length and then conclude on the lowest mode frequencies as well as compare deflections to defined loads. Using a single element the results will depend on the implementation used for that particular element in the code, whilst a certain length (to justify a beam) and number of elements will be needed.

Regards,

Alexander Boucke

--

ITAM GmbH

Oberforstbacher Strasse 230

52076 Aachen

Tel: +49 2408 146457

Fax: +49 2408 146458

Subject: Re: More weight debugging

Date: Friday, July 22, 2011 5:54 PM

From: Alexander Boucke <Alexander.Boucke@itam-gmbh.de>

To: "Wieseman, Carol d." <carol.d.wieseman@nasa.gov>

Cc: Jack Castro <jack.castro@mscsoftware.com>, "Avery, Christopher R" <christopher.r.avery@boeing.com>

Hi Carol,

the beam does not need to represent a wing, but it needs to represent a beam to get useful results for comparison. Just taking a single element's modal frequencies is not suitable imo.

> OR I can just create one myself.
> Just as long as the geometry is consistent. I guess I can do it myself.
> Assume the points defining the beginning and end of the beam are [0 0 0]
> and [0 1.5 0] - eliminating sweep, eliminating alpha rotation. Have the
> same set of properties, I1,I2,I12,J, Kappa (K1,K2), density etc and pick
> B,C,D locations at each of the 2 end points. What should we pick for
> these? What else do we need?

We can just do that. Best start with a beam with fully constant cross sectional data. We can take the data given at Node 2 (or 3, does not matter) and the cross sectional data from wing beam 3, but ignore Alpha in the first step. Create a beam of 1.5m length, constant axis offsets and constant cross section. The first modal frequency should be stable with a very low amount of elements - say 5. We should be able to get higher modes to match too. So best to a convergence study on the number of elements needed to get at least 8 modes to converge.

I can then try to implement the same beam and do the same convergence study.

We should compare mode frequencies and total mass after this convergence study delivered good results. If frequencies differ, we can attach defined forces and moments to the beam ends and compare deflection, bending and torsional angles.

I am just not sure if I find the time to do all this until next wednesday.

Regards,
Alexander Boucke

--

ITAM GmbH
Oberforstbacher Strasse 230
52076 Aachen

Tel: +49 2408 146457
Fax: +49 2408 146458

Subject: RE: FEM for a one element structure

Date: Tuesday, July 26, 2011 7:56 PM

From: Jack Castro <jack.castro@mscsoftware.com>

To: "Wieseman, Carol d." <carol.d.wieseman@nasa.gov>

Hi Carol,

I updated the one element model to try to more accurately define the bending and mass grids. I somewhat succeeded but it's still not perfect. However, I think we are very close now so this should be close enough. Comparing these grids to your original model shows differences on the order of ~ 4 so you were already very close.

Last week Alexander was suggesting not necessarily comparing one element, but comparing a uniform beam of maybe 20 elements. That would be fine. However we compare, in the end, I think we should do the following until we understand exactly which terms in Nastran are causing the differences:

- 1) Normal modes of beam with no shear center, bending or mass offsets. Check total mass, frequencies and mode shapes
- 2) Normal modes of beam with shear center offset. Bending and mass axes located at shear center
- 3) Normal modes of beam with shear center offset. Mass axis further offset from shear center. No offset for bending axis
- 4) Normal modes of beam with shear center offset. Bending axis further offset from shear center. No offset for mass axis
- 5) Normal modes of beam with shear center offset. Bending and Mass axes further offset from shear center.

All the above can be repeated with linear statics and grav loads.

Also, the test model should have the nodes aligned on the X axis of the default global coordinate system so that the CBEAM element coordinate system axes are parallel to the global coordinate system.

By the way, I do not have a meeting notice for tomorrow. Please forward if

possible. I will try to attend if I don't have a conflict.

Have a nice day,

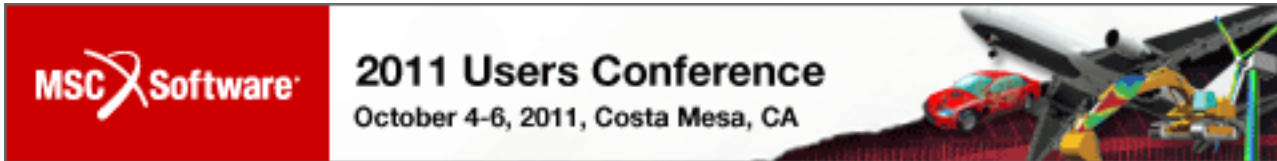
Jack

...

Jack Castro

MSC Software Corporation

Phone: 425-891-3177



From: Wieseman, Carol D. (LARC-D308) [mailto:carol.d.wieseman@nasa.gov]

Sent: Tuesday, July 26, 2011 9:37 AM

To: Jack Castro

Subject: Re: FEM for a one element structure

Great. I'd like us to come up with a few slides and a story for the telecon tomorrow.

Alexander said he would probably not be able to get to it before Wednesday tomorrow. He still needs to give us the cable/non-structural masses at their locations so that they can be added to the solid element model(s). We had been using the HEX model but now are considering using the TET model which has been checked out more by Aachen. We went with the TET because tools were developed at NASA to do the structure to aerodynamic grid interpolation can handle TETs along with QUADs etc but not TET elements. The tool probably will get updated so it doesn't matter.

I am questioning how I should put the loads in at Point 23. Presently I had individual forces in the global x,y,z directions. To extract the stiffness by looking at the displacement I wonder if it should be elemental directions but if the grid is not at the shear center am I still going to get the data I need easily to get stiffness comparisons. If instead of using the CBEAM with 22 and 23 and use 3022 and 3023 with zero offsets would I get the same results. I get

displacements at 22 and 23 not 3022 and 3023.

Alexander has been resisting looking at a one element finite element model (as you probably saw) and would rather generate “a new” wing that actually represents a beam with properties that don’t change but with more elements.

Thanks for looking into this further.
Carol

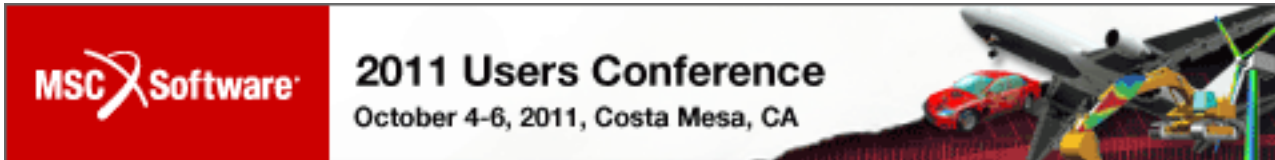
On 7/26/11 12:17 PM, "Jack Castro" <jack.castro@mscsoftware.com> wrote:
Hi Carol,

Yes I meant Alexander! Not sure who I was thinking of when I typed “Robert”. I have not had a chance to get back to this topic yet today as Boeing is having me do a bunch of other things for them today (I do contract work for them 3 days a week.) However, I expect to have time at some point today and it should not take me long.

Jack

...

Jack Castro
MSC Software Corporation
Phone: 425-891-3177



From: Wieseman, Carol D. (LARC-D308) [mailto:carol.d.wieseman@nasa.gov]
Sent: Monday, July 25, 2011 2:19 PM
To: Jack Castro
Subject: Re: FEM for a one element structure

Robert?

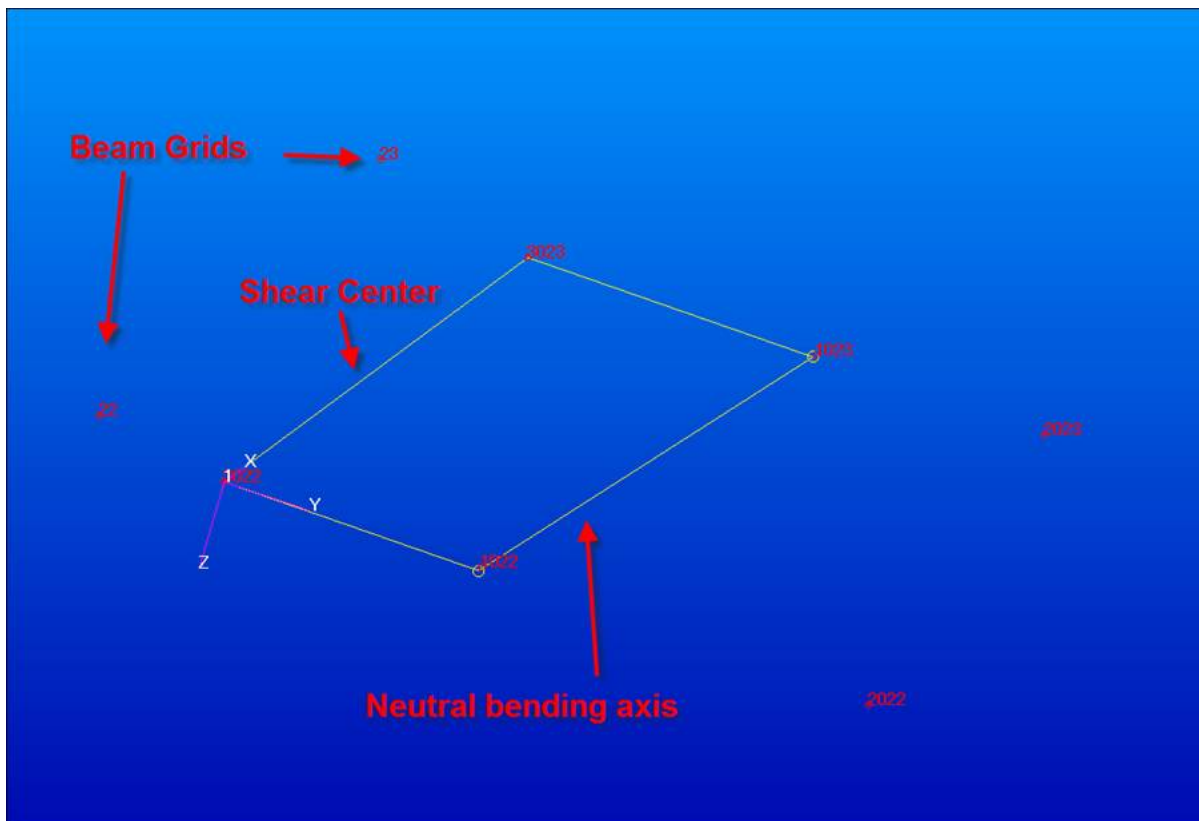
I think you mean Alexander – using his finite element code?

That's a good idea – give me new grid points for the B and C points and see what Alexander gets when he uses them in his model. Shear points were easy because everything was in the global coordinate system.

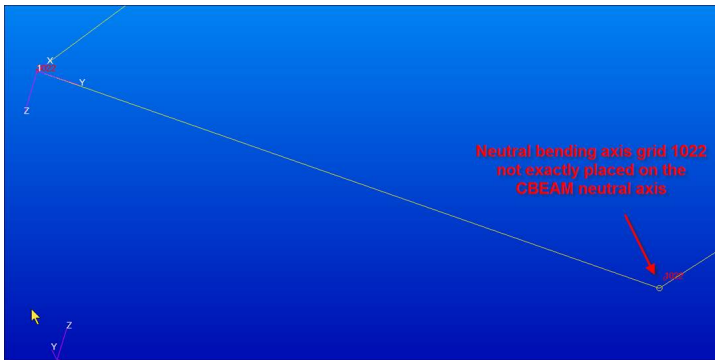
I wasn't sure if I did all the transformations correctly to get the locations right for the bending and center of mass since they involve transformation to the local element coordinate system. Since you have the beam element and all the grid locations you could double check that I did the calculations correctly.

On 7/25/11 4:24 PM, "Jack Castro" <jack.castro@mscsoftware.com> wrote: Carol,

The shear center offsets are correctly placed at nodes 3022 and 3023. The “bending axis” location is pretty close, but not exactly at the grid locations. We cannot get it exactly at the grid locations since Nastran only allows the bending axis to be defined in two dimensions relative to the shear center. If we look at the location of the grids in the plot below, we can see that from a “far field” perspective, the bending grids appear to be placed properly.



However, if I zoom in on the bending grids, then we see there begins to be some separation due to the two-dimensional neutral bending axis offset from shear center....



I have not computed how far off we are yet, but you can see I have created a coordinate system with X aligned with the beam X axis and Y aligned with the Beam Y axis. We can define the bending and mass grids relative to this coordinate system and then transform back to Basic. Then we can have Robert rerun his single beam model with the exact same bending and mass grid locations as Nastran used. I need to leave now for the rest of the day to go to a customer meeting so I will get back to this tomorrow. Tomorrow I will be at Boeing all day and will try to catch up with Kumar as well.

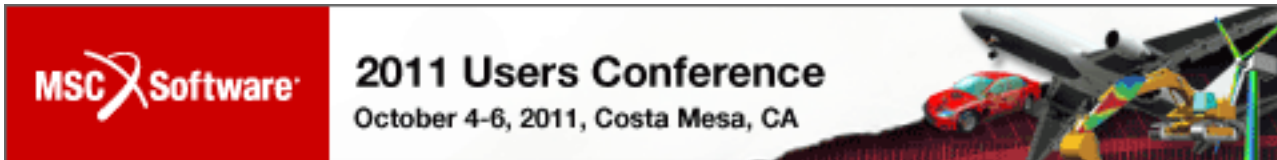
Jack

...

Jack Castro

MSC Software Corporation

Phone: 425-891-3177



From: Wieseman, Carol D. (LARC-D308) [mailto:carol.d.wieseman@nasa.gov]

Sent: Monday, July 25, 2011 12:17 PM

To: Jack Castro

Subject: Re: FEM for a one element structure

Looks like I sent the wrong file and the wrong PBEAM card.

With the offset the shear centers should be at at point 3022 and 3023 at the two ends.

```
GRID*,3022,,0.30047,0.10198,+  
*, -0.00091925  
GRID*,3023,,0.32245,0.15208,+  
*, -0.0018564
```

The locations of the bending is at Grids 10xx,
Locations of mass center are grids 20xx']

I'm not convinced I am calculating them properly but I put the grid cards in the file specifically so others could check,
Here is the correct PBEAM card...

```
PBEAM*,24,24,0.0092645,0.000100589,+  
*,2.60238e-06,-1.44405e-06,0.00010319,0.,+  
*,0.0,0.0,0.0,0.0,0.0,+  
*,0.0,0.0,0.0,0.0,0.0,+  
*,9.3637,9.3637,,,+  
*,,,,,,+  
*,0.0283175,0.000485133,0.0226755,0.0,+  
*,0.0112313,0.0,0.0125929,0.0
```

Haven't tried to run it yet

From original file from Boucke

```
Node => Wing Node 2; 0.29513·m; 0·m; 0.10417·m;  
Node => B Node 2; 0.31086·m; 0.00071382·m; 0.097707·m;  
Node => C Node 2; 0.32667·m; 0.00080760·m; 0.091215·m;  
Node => D Node 2; 0.30047·m; 0.00091925·m; 0.10198·m;
```

```
Node => Wing Node 3; 0.31591·m; 0·m; 0.15476·m;  
Node => B Node 3; 0.33410·m; 0.0017307·m; 0.14729·m;  
Node => C Node 3; 0.34343·m; 0.0017147·m; 0.14346·m;  
Node => D Node 3; 0.32245·m; 0.0018564·m; 0.15208·m;
```

```
-----  
-- Wing Element 3  
-- Span from 0.10417·m to 0.15476·m  
-- Length 0.054694·m  
-- Mass 4.0132·kg
```

```
Elastic_Material => Mat 3  
Rho => 8162.2·kg/m^3;  
-- incl. 2.485kg/m fuer kabel  
E_Mod => 186110000000·kg/m·s^2  
G_Mod => 6224300000·kg/m·s^2  
kappa => 9.3637;
```

```
Cylindric_Module => Wing Beam 3  
Node_1 => Wing Node 2  
Node_2 => Wing Node 3  
Material => Mat 3  
Area => 0.0092645·m^2  
I_Yy => 0.00010061·m^4
```

```
I_Zz    => 0.0000025811·m^4
Alpha    => -0.014733
J        => 0.00010319·m^4
```

The inertias have been modified by rotation alpha. Coordinate system is different xnastran=x, ynastran=-z, znastran=y

On 7/25/11 1:19 PM, "Jack Castro" <jack.castro@mscsoftware.com> wrote:
Hi Carol,

Let me know if you need more input from me on the Beam element questions. Just to close the loop on the one element model you created, this is what you have set up:

Grid A (22) Coordinates: .29513 .10417 0.
Grid B (23) Coordinates: .31591 .15476 0.

Both the placement and displacement coordinate systems of the above grids are in the default "Basic" Nastran coordinate system.

The element shear center is offset from the above coordinates in the local displacement coordinate system of each grid. However, since there are no local displacement systems input, the offsets in this case are in the Basic Coordinate System relative to Grid A and Grid B.

The shear center offset from Grid A is: 0.00534 -0.00219 -0.00091925
The shear center offset from Grid B is: 0.00654 -0.00268 -0.0018564

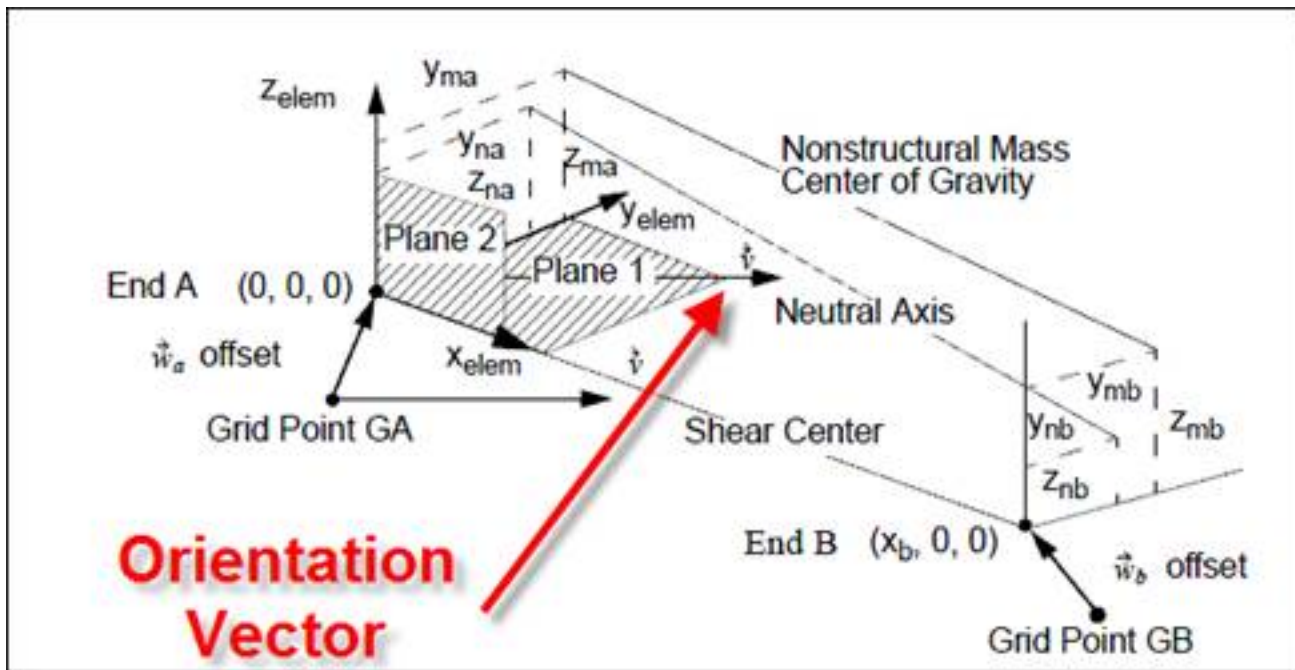
The PBEAM entry shows no additional offsets of the neutral axis or mass center. Therefore, both the neutral axis and mass center lie on the shear center line in your example.

The section properties input are:

```
Area: 0.0092645
I1: 0.000100589
I2: 2.60238E-06
I12: -1.44405E-06
J: 0.00010319
```

Your orientation vector on the CBEAM is: X=1.0, Y=0.0, Z=0.0. These XYZ directions are parallel to the Grid point A displacement coordinate system. However, since there is no displacement coordinate systems input, this orientation vector is parallel to the Basic Coordinate System X axis in this case.

Therefore, Plane 1, which defines the plane of bending for the I1 inertia, is the plane formed by the Shear Center Line, and the Orientation vector parallel to the Basic X Axis emanating from shear center end A. (See graphic below)



I cannot recall what element from Boucke's model you are reproducing, but if you can remind me, then I will compare your inputs versus that element in his model and see if it makes sense.

Have a good day,
Jack

...

Jack Castro
MSC Software Corporation
Phone: 425-891-3177

-----Original Message-----

From: Wieseman, Carol D. (LARC-D308) [mailto:carol.d.wieseman@nasa.gov]
Sent: Friday, July 22, 2011 4:41 PM
To: Jack Castro
Subject: Re: FEM for a one element structure

The other difference is that it costs more to run MSC Nastran than MD Nastran for