



ADAPT

# Technical Issues

- Frequently Asked Questions
- Frequently Encountered Errors

- *Featuring* -  
**ADAPT-ABI 2009**

ADAPT Corporation, USA  
ADAPT International Pvt. Ltd, Kolkata, India

**Tuesday, November 17, 2009**

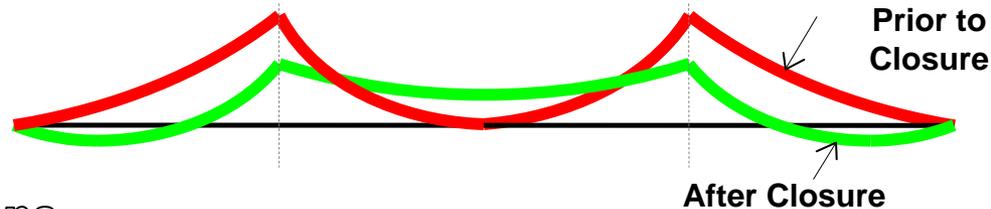




## What is the key advantage of ADAPT-ABI over other structural analysis software?

Unlike general structural analysis software which mainly focus on the completed bridge behavior, ADAPT-ABI models the material properties, time-dependent factors of creep, shrinkage, relaxation in prestressing, elongation of tendons, aging of concrete and provides information for geometry control during construction (for example the pre-camber required during construction of a balanced cantilever bridge to counteracting the short-/long-term deflection), in addition to some post-construction design force envelop calculations like influence line, bending moment and shear force diagrams.

As the loadings during construction phases often causing more severe force resultants for critical structural elements, ADAPT-ABI helps bridge engineer to get a better overview and thorough understanding of the design force envelop and behavior of a bridge throughout its entire life span.





## Apart from construction stages analysis, what type of post-construction analysis ADAPT-ABI can perform?

ADAPT-ABI can calculate the gravity resultant forces due to dead, live and moving loads, and provide graphical diagrams of influence lines and force (moment, shear, torsion\*) envelopes.

Apart from day-wise responses during construction program generates the response for all user defined load cases, total construction loadings, PT, Hyperstatic, Long Term (including Creep, Shrinkage & Relaxation) and all moving loads with their envelop.

From ABI v5 onwards Torsional forces are calculated in using simplified assumptions by Witecki.

Recently version of the program can generate Seismic Loading using co-efficient method.





## Presently I am using Spreadsheet and In-house tools to handle bridge projects. Can you please describe merits and demerits of ADAPT-ABI in comparison to our existing solution?

Often spreadsheet solutions as well as in-house tools are used to analyze and design their bridge geometry. While these solutions quickly produce initial solutions for simple structures, these mostly have following disadvantages:

- May not handle construction stages, requires lot of documentation
- May not produce Moving load result envelope
- Elongation report generation may not be possible
- May produce conservative results for even simpler projects
- Camber calculation is difficult
- Requires up-front investment in model generation



On the other hand though using ABI user needs to idealize the model like any other finite element software, for mostly medium size bridges it offers following advantages:

- Produces initial as well as optimized designs
- Simple procedure supports quick and reliable design times
- Excellent design (training) tool for beginners as well as experts
- Camber calculation even for CIP method is accurate and easy
- Simpler and quicker Moving load analysis and Elongation report

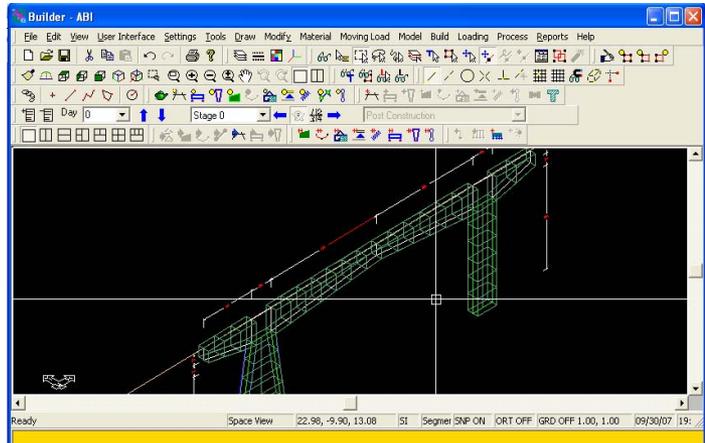




## What are the main differences between v 4.xx and v 2009 (v5) of ADAPT-ABI?

ADAPT-ABI version 5 is a major release to ease the pain of old ADAPT-ABI version 4.x users. In ver. 4.x, users were required to complete the modeling entirely in textual command input. While all the technical features of ver 4.5 are supported, version 5 now provides an intuitive 3D graphical user input interface that allows users to create, modify and view model and results in 3D.

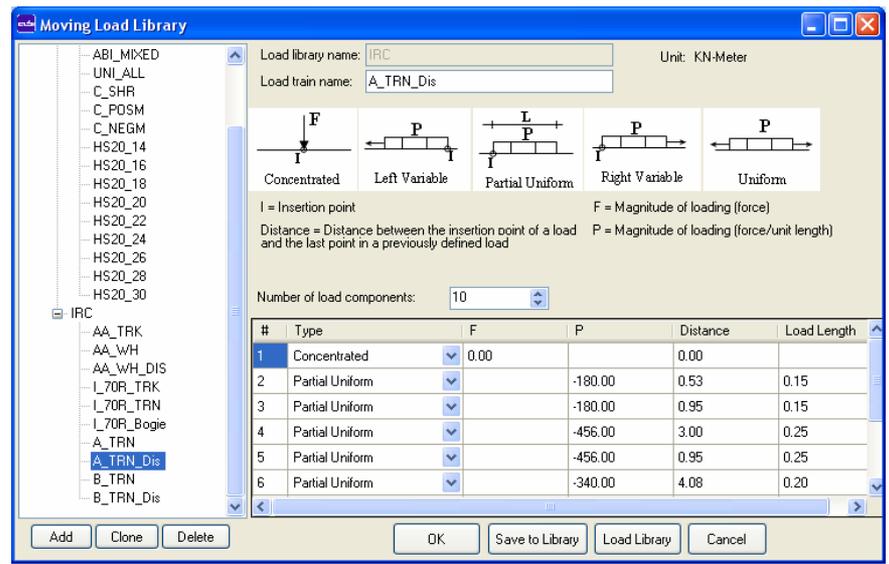
Along with many new features and enhancements of existing capabilities, new version of ABI can open older version of INP and MOV file for processing. Moreover the program can now import DWG or DXF files to generate cross sections, bridge alignments and other important components.



## Does ADAPT-ABI generate moving load as per IRC or BS5400-Part II or EC requirements?

ADAPT-ABI has a general moving load generator that allows user to define any type of truck/vehicle loading patterns. Once the patterns are defined and saved into a library file, they can be used to “run” through the travel paths to obtain the influence lines as well as design force envelops.

Therefore, even though there is no default BS5400 Pt 2 vehicle loadings pre-defined, user are free to create and enhance the moving load library to cater the loading requirements of specific design code that they are using.



The screenshot shows the 'Moving Load Library' window. On the left is a tree view of the library. The main area shows the 'Load library name' as 'IRC' and 'Load train name' as 'A\_TRN\_Dis'. Below this are diagrams for 'Concentrated', 'Left Variable', 'Partial Uniform', 'Right Variable', and 'Uniform' load types. A table below lists the defined load components:

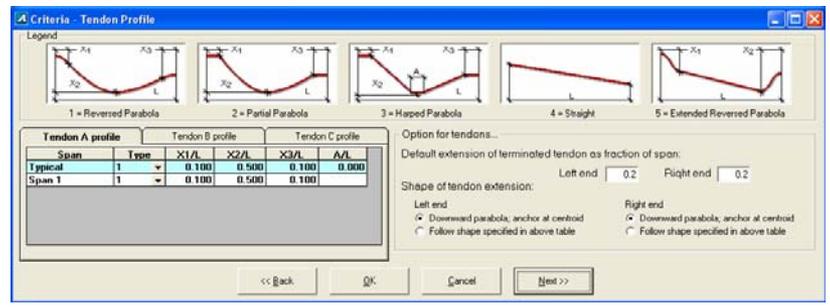
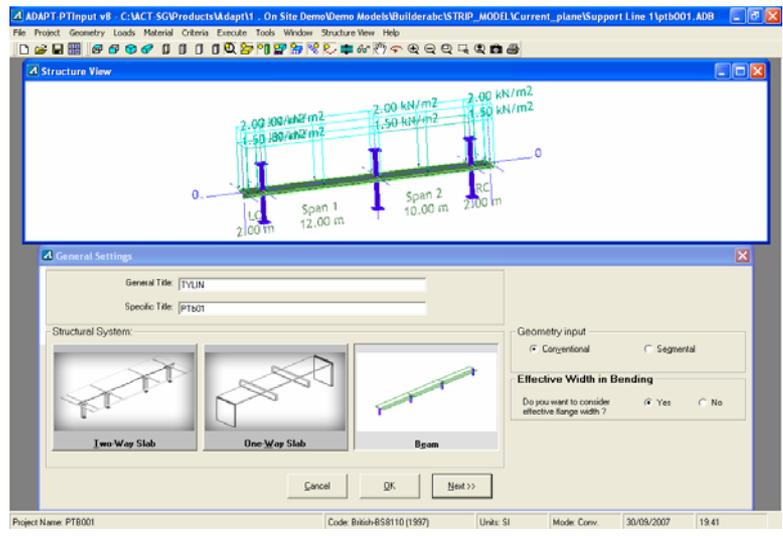
#	Type	F	P	Distance	Load Length
1	Concentrated	0.00		0.00	
2	Partial Uniform		-180.00	0.53	0.15
3	Partial Uniform		-180.00	0.95	0.15
4	Partial Uniform		-456.00	3.00	0.25
5	Partial Uniform		-456.00	0.95	0.25
6	Partial Uniform		-340.00	4.08	0.20



## Does ADAPT-ABI design post-tensioned members (like beams/girder)?

In ADAPT-ABI, users are required to input and specify the tendon shape and stressing forces, and let the program calculates the results which include pre-camber, tendon losses, elongation, and others. ADPAT-ABI doesn't design any structural component.

However, to figure out the right shape and post-tensioning forces to be applied, the calculation can be performed easily in ADAPT-PT PLUS program to the desired allowable stress limits.

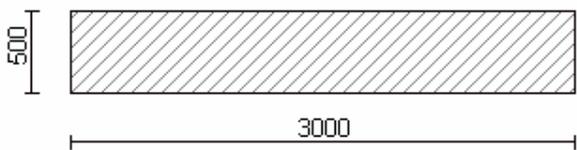




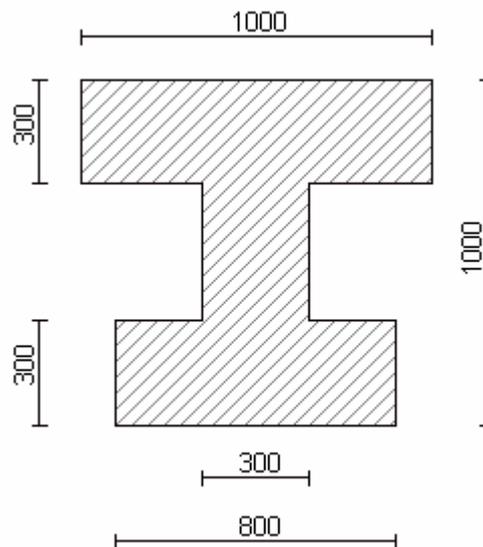
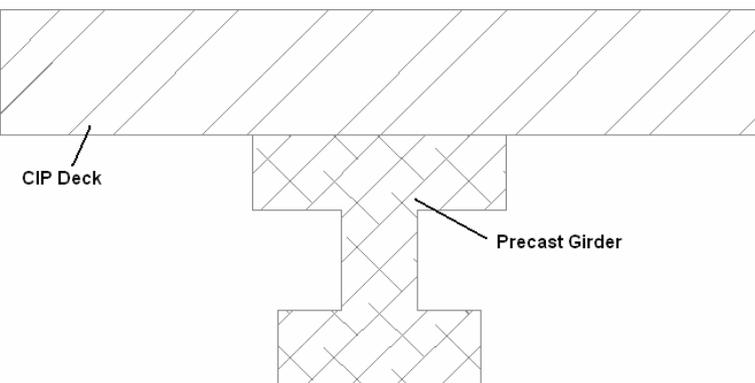
## Description of Problem:

I want to model a CIP deck on top of precast girder. How I can model that in ABI?

Also explain the composite analysis feature in ABI and how I can view combined result in result viewer?



**Deck Section**

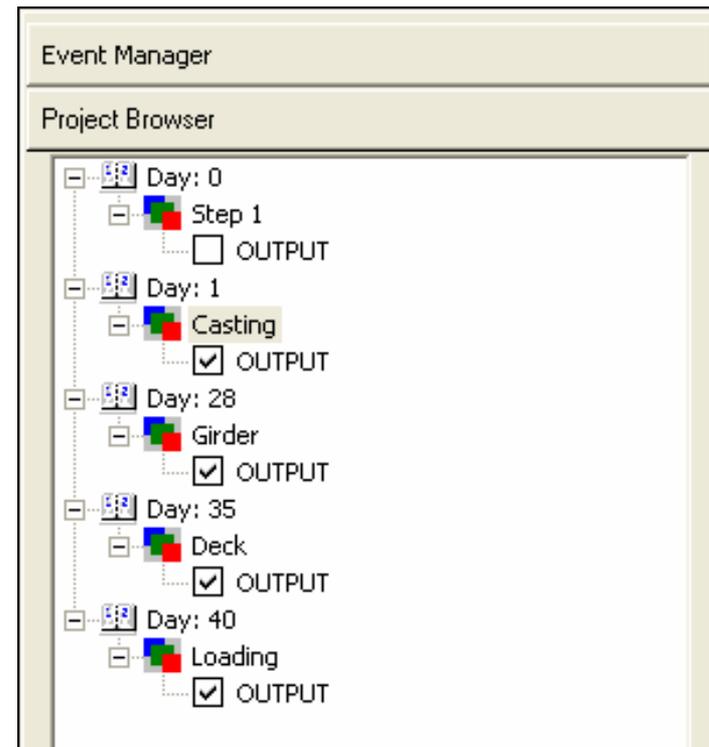


**Girder Section**



## Explanation and Solution:

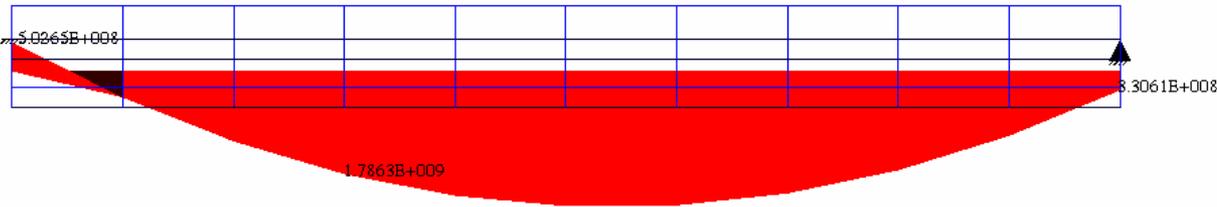
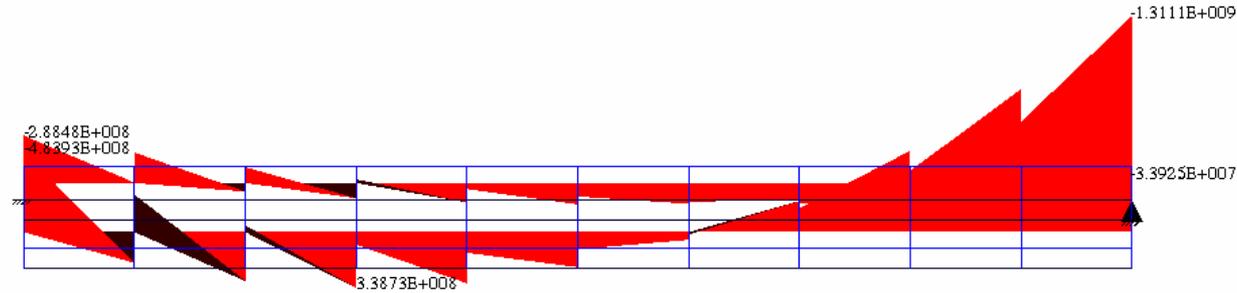
- Model the deck and assign offset 500 mm in Z direction.
- Model the girder and assign its properties.
- Model the Loading -25 kN/m and assign offset 500 mm in Z direction.
- Create Mesh with Composite option.
- Create days 1, 28, 35 and 40.
- Cast Girder on Day 1 and activate self-weight.
- Install Girder and supports in Day 28.
- Install Deck on Day 35.
- Activate UDL on Day 40.
- Generate INP and Execute.





## Explanation and Solution:

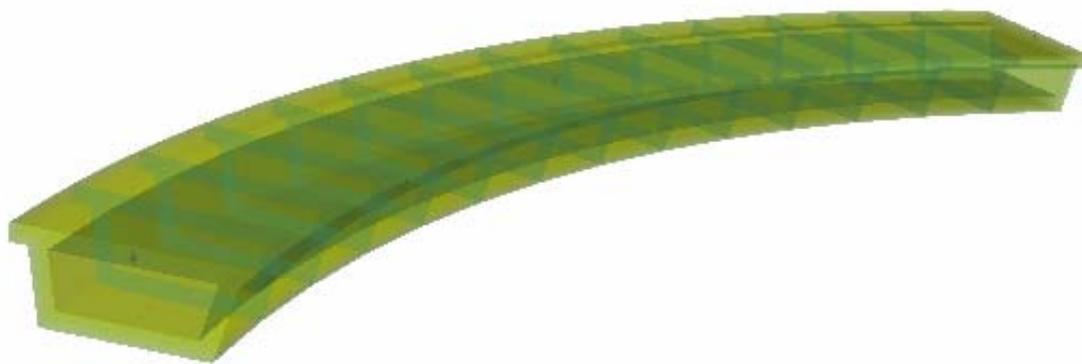
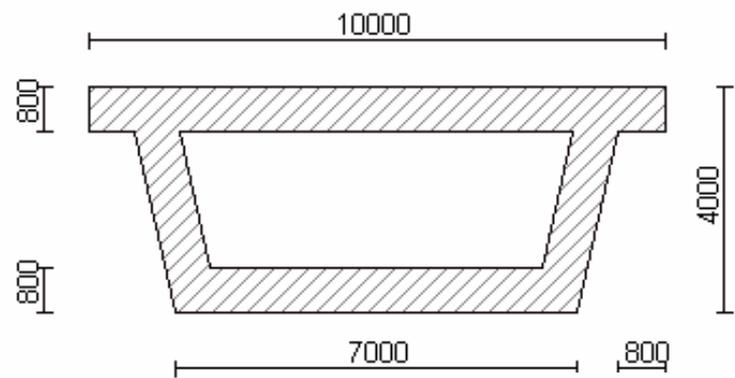
- View Bending Moment on Day 40. Two diagram will be shown, one in the Girder and second on the deck.
- Click on Select Composite Elements.





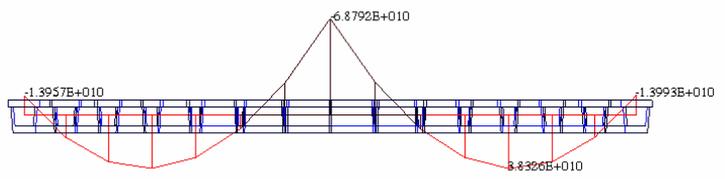
### Description of Problem:

I have a curved deck of radius 90 m and total length of 77.32 m. How I can model such deck with ease in ADAPT ABI?



## Explanation with Solution:

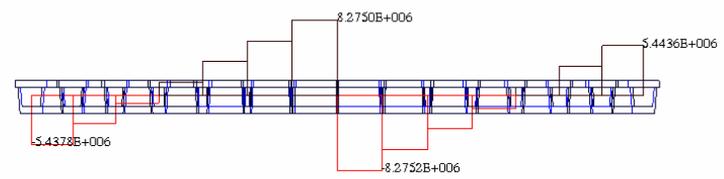
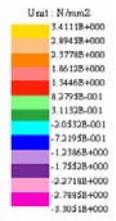
- Draw a reference line and ensure its ends are (0,0,0) and (75,0,0).
- Specify the Span as Ramped Arc with Radius 90 m.
- Model the deck, support and apply self-weight. Solve the structure.



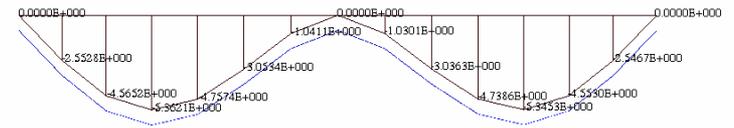
Bending Moment



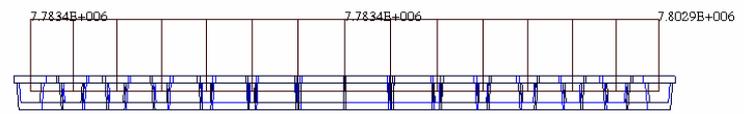
Stress Plot



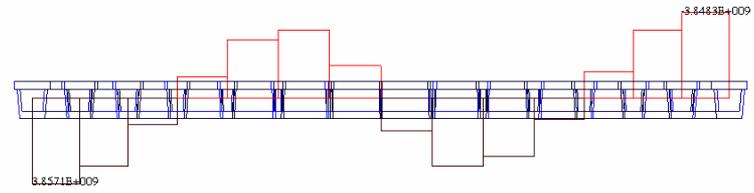
Shear Force



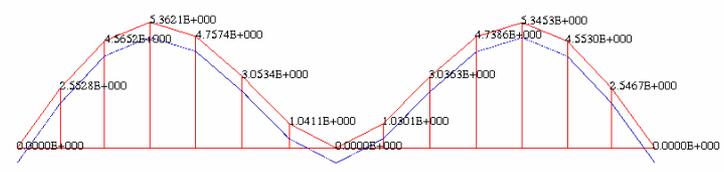
Deflection Profile



Axial Force



Torsion



Camber Profile



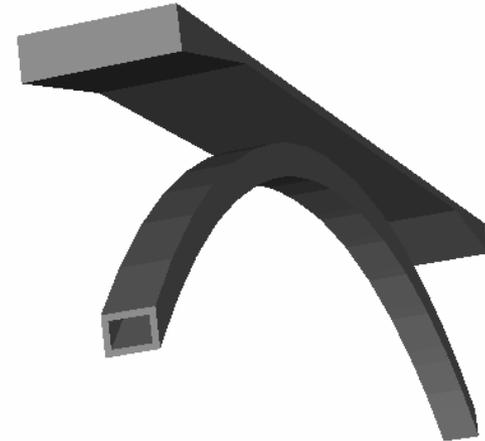


### Description of Problem:

I want to model a deck resting on a arch. The deck is 100 m long including two tapered part of 15 m on both side. Deck is 12 m wide with a thickness of 925 mm at the middle and 3.5 m thickness near abutments.

The arch is a box section of 4 m X 3 m with a wall thickness of 500 mm.

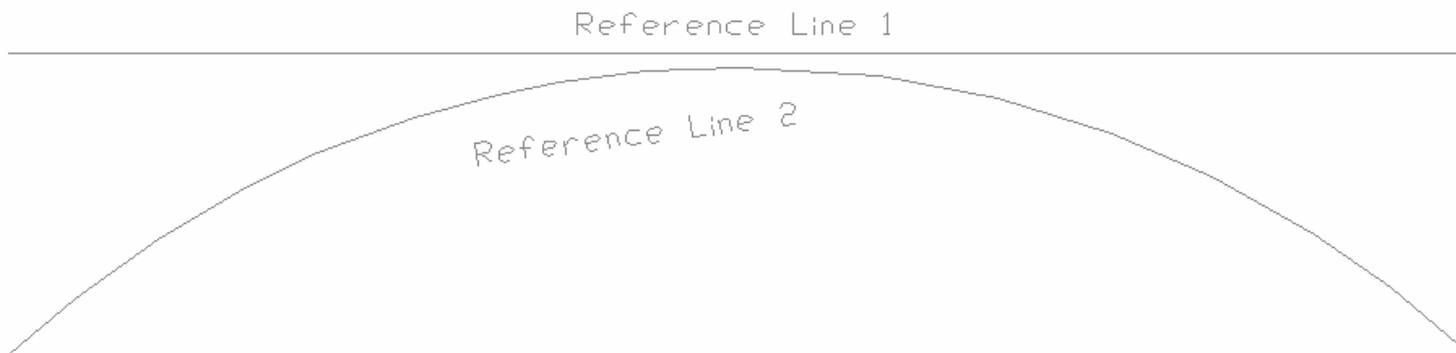
Please explain how I can model this using ADAPT-ABI.



## Explanation and Solution:

It is recommended to create two reference line, one for the deck and the second for the arch. Then select any one of these and go to developed view. Create the girder (snap both ends).

Return back to normal view. You may wish to use copy command to select the same section on the other reference line and use its property box to assign the correct reference line and section.



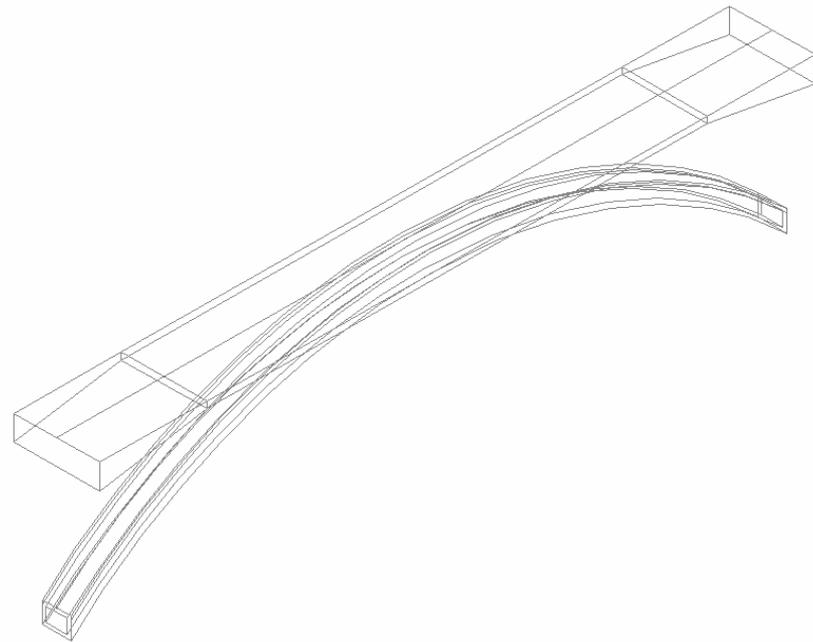
## Explanation and Solution:

contd...

Now select the deck and open its property box. Use define sub-segment command to create three segments of 15 m, 70 m and 15 m. Apply it using green tick mark.

Go to Location tab to assign proper cross sections for each sub-segments.

You may now proceed with this model to assign loadings, construction stages and so on...





## Description of Problem:

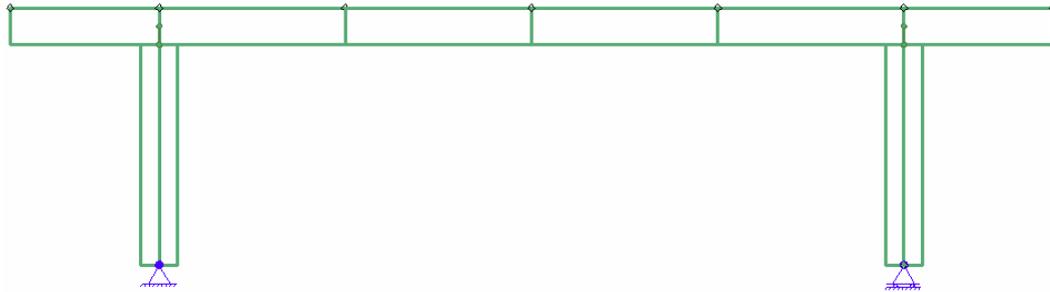
One simple bridge is resting over two piers. While left support is fixed, right support is intended to be a roller one. The installation activities are specified in ABI as follows:

Day 10 – Pier

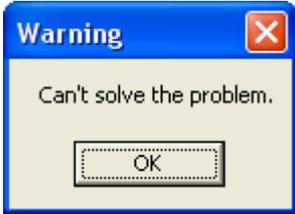
Day 20 – Pier Segments

Day 30 – Middle Segments

However during analysis program is showing “SOLUTION DID NOT CONVERGE” and couldn’t generate the results.



```
SOLVE DAY=20  
  
WARNING: SOLUTION DID NOT CONVERGE  
Terminate the program and refer to the text in  
README.txt in ABI subdirectory for help  
Press any key to Continue or X to exit !
```



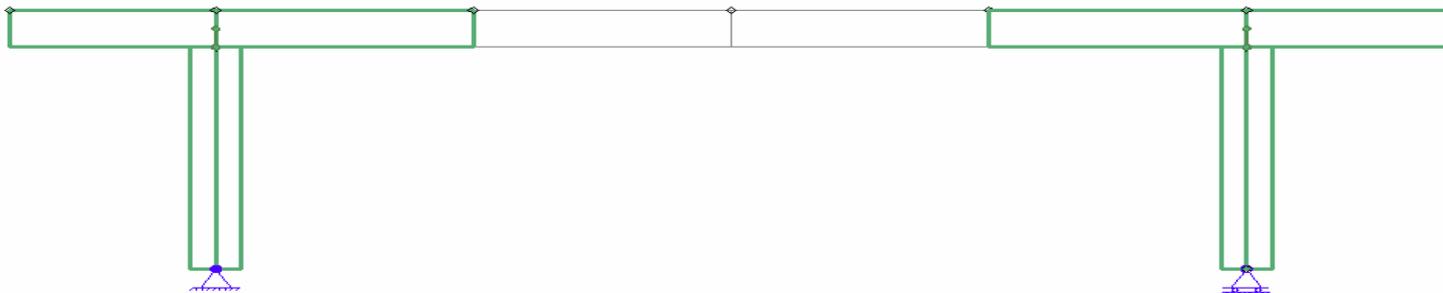
- What is the reason?
- Is there any workaround?



## Explanation:

Though the completed bridge geometry has no stability issue, observe the structure at day 20. Right part of the structure is resting upon a fixed support, but left part is resting on a roller support. Since there is no connectivity between these two parts, left part can float indefinitely in horizontal direction.

In reality some temporary gadget or equipments are used to hold left part in position till the bridge is complete. Hence during construction left support would behave like a hinged or mostly at fixed one.





## Solution:

Initially assign both the supports as fixed. Create one step on day 30 to change support behavior from fixed to roller once entire deck is constructed.

Project Browser

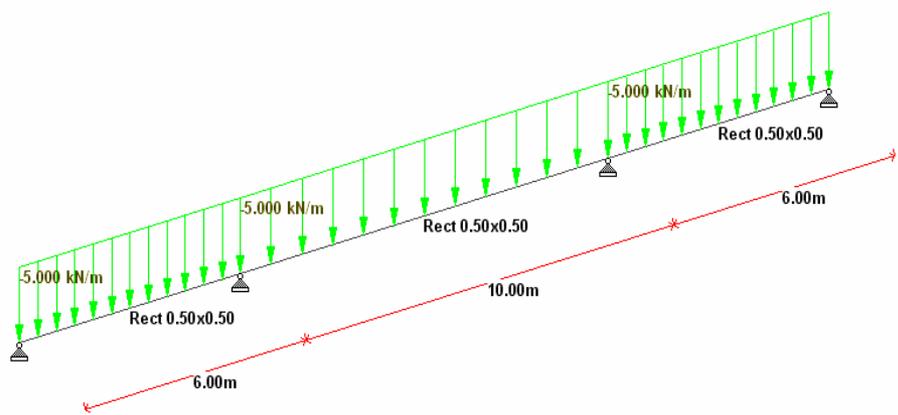
- Day: 0
  - Step 0
    - OUTPUT
- Day: 10
  - Install Pier
    - OUTPUT
- Day: 20
  - Pier Segment
    - OUTPUT
- Day: 30
  - Complete
    - OUTPUT
  - Change Support
    - OUTPUT



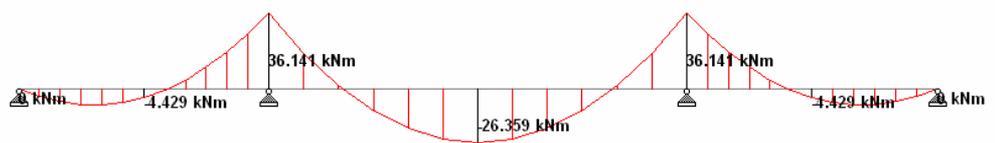
## Description of Problem:

I have modeled one 3 span beam in STAAD. Model parameters are mentioned below:

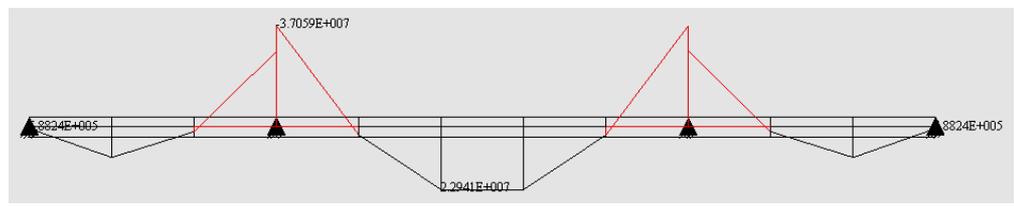
- Spans = 6 m + 10 m + 6 m
- Cross Section = 500 mm X 500 mm
- Material = Concrete
- Member Load = 5 kN/m throughout



However when the same model is created in ABI, Bending Moments are not matching with STAAD results.



Why there is such variations?



### Explanation:

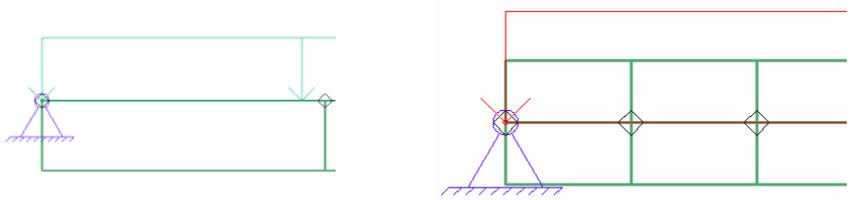
Whenever one wish to match result between two programs, all parameters related to analysis must be same.

Moreover unlike STAAD, ABI considers time dependents parameters like Creep, Shrinkage etc. Hence for comparison, one needs to make sure effect due to such parameters are neutralised.

Also make sure the beam is meshed with adequate numbers of elements.

Nevertheless match E value of concrete with that in STAAD model.

Finally note that by default STAAD consider support along centroid of the section. However ABI places supports at the top of the deck.



Wce (used for calculating Ec):	<input type="text" value="2400.00"/>	kg/m3
Modulus of elasticity, Ec	<input type="text" value="21718.00"/>	N/mm2
Thermal expansion coefficient:	<input type="text" value="0.00001000"/>	/C
Long term effect		
Concrete model:	<input type="text" value="ACI-1978"/>	
Ultimate creep value	<input type="text" value="0"/>	
Ultimate shrinkage value	<input type="text" value="0"/>	



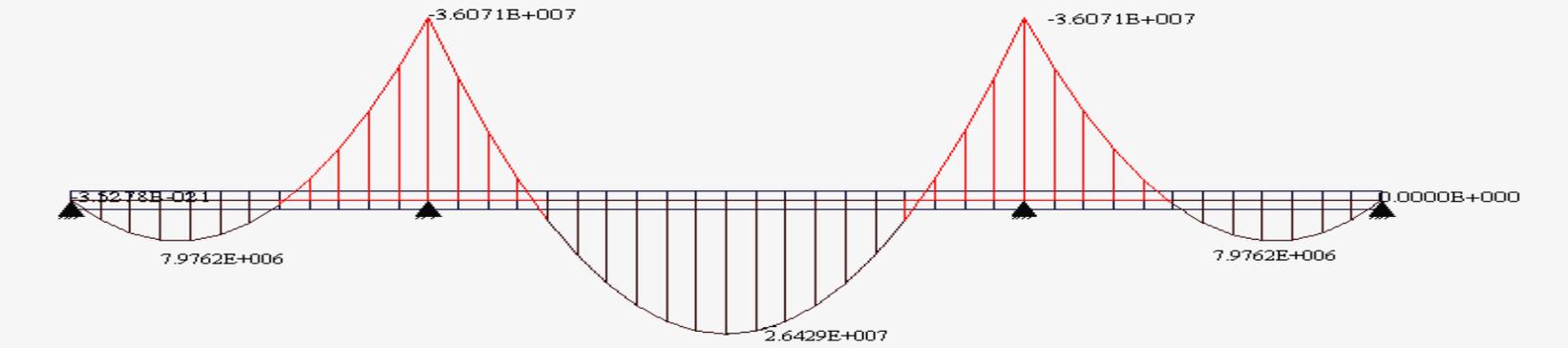
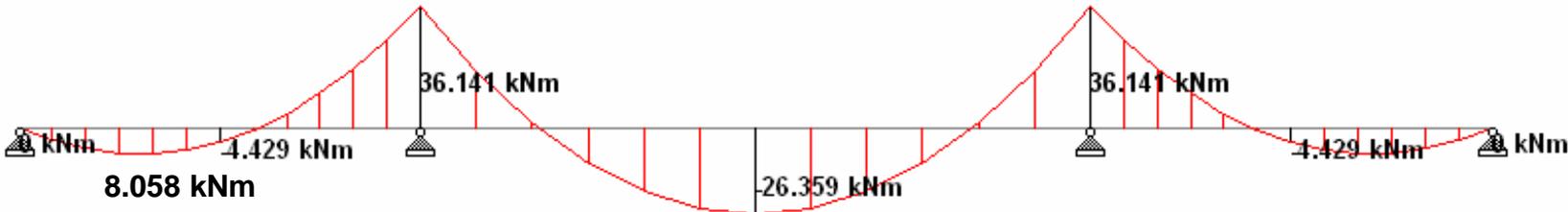
**Solution:**

Set ultimate Creep and Shrinkage values as zero.

Mesh this structure with 0.5 m cell size in ABI. As you might be aware STAAD splits a beam in 12 sub-segments internally.

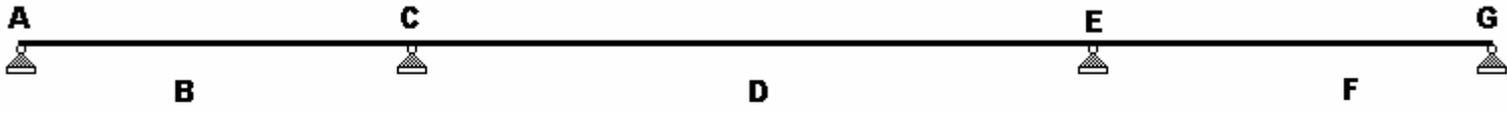
Input E value of Concrete as 21718 N/mm<sup>2</sup>.

Either move the deck or the support to restrain the deck along centroid. It will also bring the load along centroid of the member as in STAAD.





## Comparison of Results:



**B -** } Points with maximum span moment values  
**F -** }

**D -** Mid-point of central span

Locations	STAAD	ABI Default	ABI Final
<b>A</b>	0.000	-0.588	0.000
<b>B</b>	-8.058	11.176	7.976
<b>C</b>	36.141	-37.059	-36.071
<b>D</b>	-26.359	22.941	26.429
<b>E</b>	36.141	-37.059	-36.071
<b>F</b>	-8.058	11.176	7.976
<b>G</b>	0.000	-0.588	0.000

All values are in kN-m. Please note while STAAD reports compression values +ve, ABI reports compression values -ve.



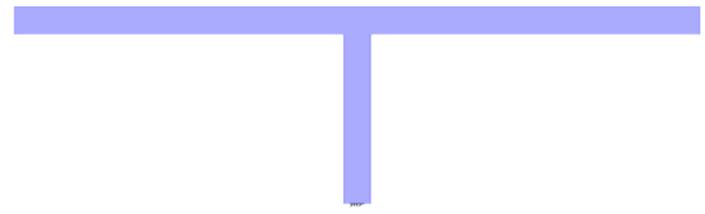


### Description of Problem:

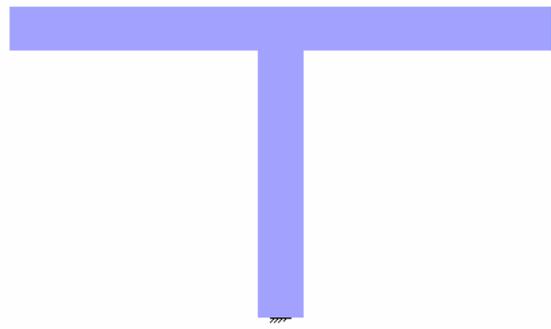
I have used traveler for a balance cantilever model. All segments and supports are installed on intended days. Movement and removal for both the travelers are also specified from their respective property boxes.

However after analysis no traveler is being shown in the result viewer. Am I missing something?

Also clarify while specifying traveler movement how I should specify the cell numbers?



Day 20  
Day 10



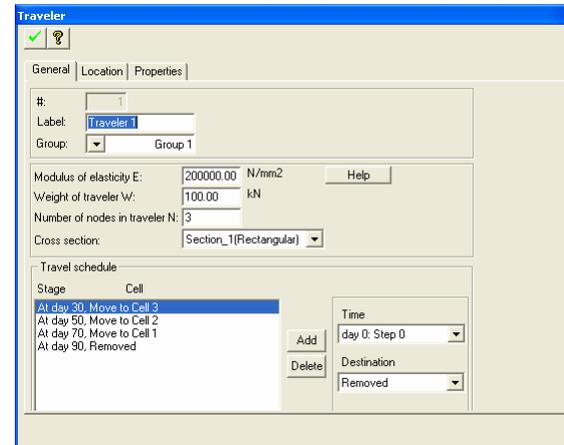
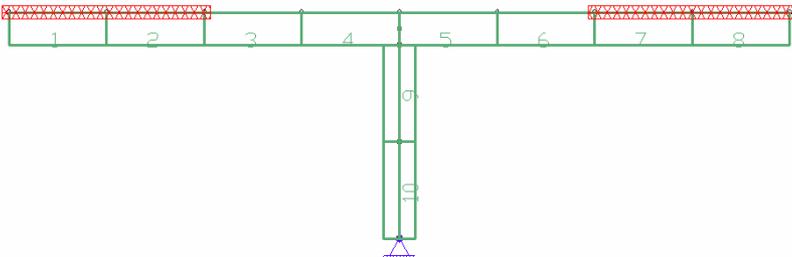


## Explanation:

While using Traveler please note user needs to perform two operations:

1. User needs to select the installation day from left hand project browser and select the Traveler to use “Install at Step” command. Subsequently on the day of removal, select the traveler to use “Remove at Step” command.
2. Double click on the Traveler to specify its location for each day of movement starting from the installation day and finally specify “Removed” on the day of removal.

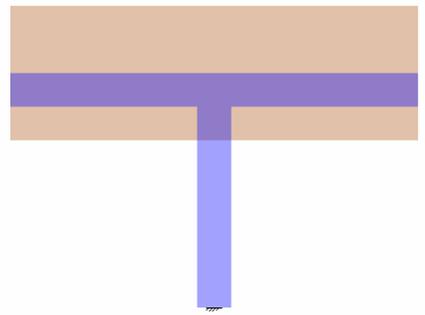
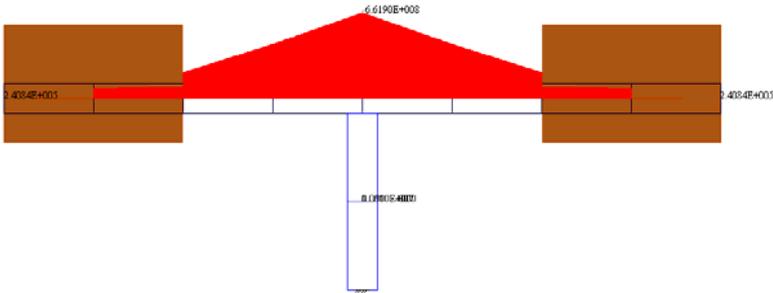
Also note since traveler span from left to right you need to specify left side segment as its location, even if that is not constructed.





## Solution:

- Use "Install at Step" and "Remove at Step" commands.
- Specify position of the travelers properly.
- Analyse and view the results.



Project Browser

- Day: 0
  - Step 0
    - OUTPUT
- Day: 10
  - Pier
    - OUTPUT
- Day: 20
  - Pier Segment
    - OUTPUT
- Day: 30
  - Install Trvlr
    - OUTPUT
- Day: 40
  - 2nd Segment
    - OUTPUT
- Day: 50
  - Move Trvlr 1
    - OUTPUT
- Day: 60
  - 3rd Segment
    - OUTPUT
- Day: 70
  - Move Trvlr 2
    - OUTPUT
- Day: 80
  - Final Segment
    - OUTPUT
- Day: 90
  - Remove Trvlr
    - OUTPUT



## Description of Problem:

I have used spring support in my model. However during analysis program is showing following error.

What is the reason and how I can proceed?

```
STANDARD EXECUTION
DAY:      0      STAGE:  2      STEP:   1      ITERATION:  25

SOLVE DAY=0
OUTPUT

SOLVE DAY=0
LOADING

SOLVE DAY=0

WARNING: SOLUTION DID NOT CONVERGE
Terminate the program and refer to the text in
README.txt in ABI subdirectory for help
Press any key to Continue or X to exit !
```



## Explanation:

Notice the top part of the processing window. When the warning is displayed, number of iteration already reached 25 (default value).

It seems that program needs more number of iterations to converge the result. However since there is no graphical command, hence user needs to set higher iteration limit.

```
STANDARD EXECUTION
```

```
DAY:      0      STAGE:   2      STEP:    1      ITERATION: 25
```



## Solution:

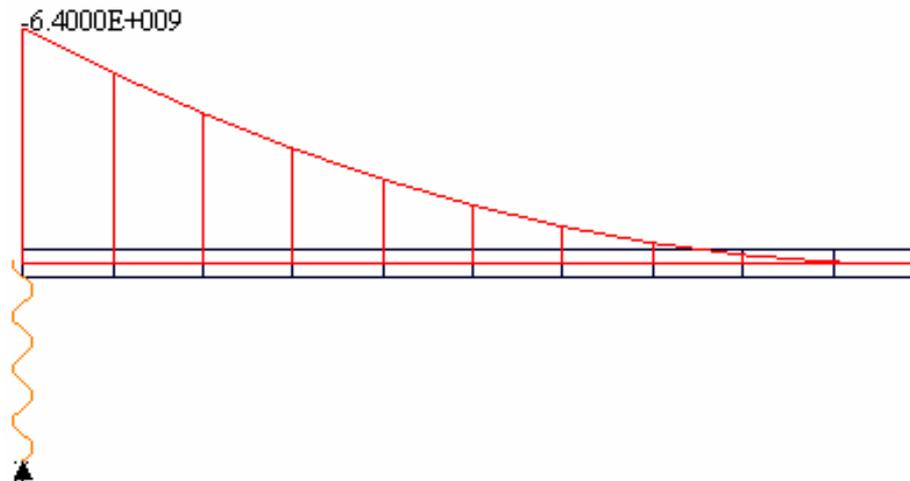
Edit Input Data to open INP file. Go to the line where analysis couldn't converge. The original syntax will be as follows:

```
SOLVE Day=0
```

Set limit to iteration as 500:

```
SOLVE IT=500 Day=0
```

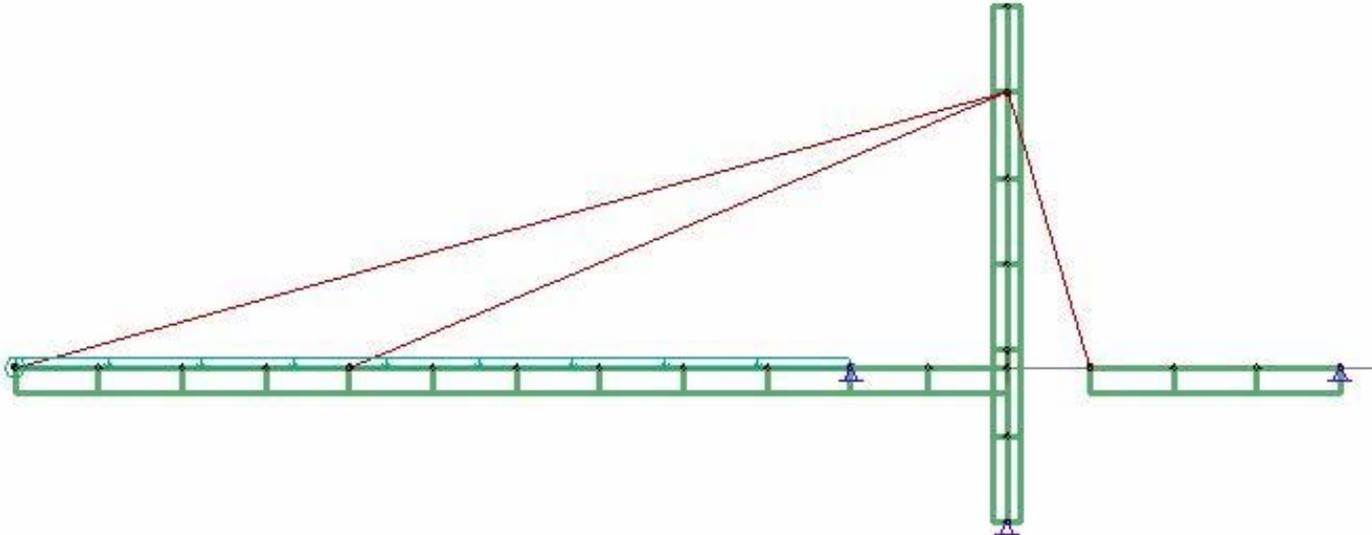
Execute the file. Now this will be processed without any error.





## Description of Problem:

We have a stay model. During construction at one stage, program is showing convergence error. Since there are many steps setting higher iteration limit before final run is very much time consuming. We understand ABI has a strict convergence test where allowable difference in stress for two successive iterations is only 0.007 MPa = 1 psi. How I can increase the toleration limit for the entire run?





## Solution:

Edit Input Data to open INP file. Go to the line where SET commands are specified right after "MESH COMPLETE". The original syntax will be as follows:

```
SET T=20.00 G=0.00,0.00
```

Relax the stress convergence criteria as follows:

```
SET T=20.00 G=0.00,0.00 C=15
```

Execute the file. Now this will be processed without any error.

