JOURNAL OF MATERIALS & CONSTRUCTION
JOURNAL OF VIETNAM INSTITUTE FOR BUILDING MATERIALS
ISSN 2734 - 9438 ISSN 1859 - 381X
Website: www.jomc.vn

Structural analysis of continuous beam using finite element method and ANSYS software

Lam Thanh Quang Khai[1], Do Thi My Dung[1], Truong Van Bang[1]

[1] Faculty of Civil Engineering, Mien Tay Construction University

KEYWORDS
continuous beam
three-moment equation
force method
displacement method
ANSYS
FEM

ABSTRACT
A number of methods were developed to analyze bending moment and shear force diagrams for multi-span continuous beams, including the three-moment equation, force method, displacement method, moment distribution method, and finite element method (FEM), etc. The FEM can solve many kinds of complex problems, many different types of loads, and different boundary conditions that cannot be solved by classical methods. Presently, with the advent of simulation software such as ANSYS, a software specialized in numerical simulation in structural analysis, has made the analysis of complex problems easier than other methods. In this paper, the authors have analyzed a three-span continuous beam using finite element method and ANSYS via GUI method and APDL parameters, this is a simple problem but here is the beginning to study more complex problems with ANSYS

1. Introduction

In the literatures of structural mechanics, many authors have used different methods to calculate internal forces (the bending moment and shear force) for continuous beams such as the force method, displacement method, focusing method, and three-moment equation. From the displacement method two approximate computations were used called Cross and Kani methods [1]. Since the advent of computers, other numerical methods have been developed such as: FEM, and finite difference method. These methods are often applicable to simple problems.

Another method has also proposed to calculate continuous beams and using mathematical method in calculating continuous beams [2]. In FEM, studies have analyzed many different types of problems [3-6], or beam analysis according to Timoshenko [7], and simplified analysis of continuous beams [8]. In this method, with the support of the software, the analysis of continuous beams has become simple, the results are quite accurate.

In addition to the methods mentioned above, the analysis of continuous beams or building structures has become even easier with the advent of ANSYS simulation software. ANSYS can simulate the process of systems with complex boundary conditions, multiple environments, and various interactions that are difficult to simulate using other methods. Therefore, in this paper, the authors have analyzed the three-span continuous beam problem using FEM and guided in ANSYS Release 16.0 software [9-12]. A numerical result is performed and compared with the finite element solution.

2. Materials and methods

A three-span continuous beam has dimensions and loads as shown in Figure 1, E = $2{,}6\times10^7$ kN/m^2 modulus of elasticity of beam, $b\times h$ = 0,3 × 0,6 m rectangular beam cross-section, I = 0,0054 m^4 second moment of area. Calculate rotation displacement and bending moment diagram at the end of elements using FEM and ANSYS.

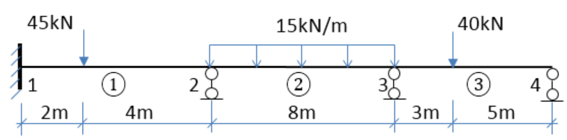


Figure 1. A three-span continuous beam.

3. Results and discussion

3.1. Analysis of the three-span continuous beam problem using FEM

- Step 1: Label the points, and elements. The points are numbered as 1, 2, 3, 4; The elements are numbered as ①, ②, ③. (Figure 1)

- Step 2: Calculate the stiffness matrix of the elements:

$$[K]^1 = \begin{bmatrix} \frac{4EI}{6} & \frac{2EI}{6} \\ \frac{2EI}{6} & \frac{4EI}{6} \end{bmatrix} = \frac{EI}{6}\begin{bmatrix} 4 & 2 \\ 2 & 4 \end{bmatrix}$$

$$[K]^2 = \begin{bmatrix} \frac{4EI}{8} & \frac{2EI}{8} \\ \frac{2EI}{8} & \frac{4EI}{8} \end{bmatrix} = \frac{EI}{8}\begin{bmatrix} 4 & 2 \\ 2 & 4 \end{bmatrix}$$

$$[K]^3 = \begin{bmatrix} \frac{4EI}{8} & \frac{2EI}{8} \\ \frac{2EI}{8} & \frac{4EI}{8} \end{bmatrix} = \frac{EI}{8}\begin{bmatrix} 4 & 2 \\ 2 & 4 \end{bmatrix}$$

- Step 3: Establish global stiffness matrix:

$$[K] = EI\begin{bmatrix} 0{,}667 & 0{,}333 & 0 & 0 \\ 0{,}333 & 1{,}167 & 0{,}25 & 0 \\ 0 & 0{,}25 & 1 & 0{,}25 \\ 0 & 0 & 0{,}25 & 0{,}5 \end{bmatrix}$$

https://doi.org/10.54772/jomc.v11i02.291

Received 03/06/2021, explanation 22/07/2021, Accepted 21/09/2021

- Step 4: Applying boundary conditions: Fixed at the left end of the beam, $\theta_1 = 0$. Proceed to edit row 1 column 1 corresponding to θ_1 in global stiffness matrix. Or the component on the main diagonal is edited to 1, the other element is 0. The stiffness matrix after editing is:

$$[K] = EI\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1{,}167 & 0{,}25 & 0 \\ 0 & 0{,}25 & 1 & 0{,}25 \\ 0 & 0 & 0{,}25 & 0{,}5 \end{bmatrix}$$

- Step 5: Calculate bending moment at fixed and load at equivalent points: From structural mechanics, it is possible to determine:

$$\begin{Bmatrix} M_1^F \\ M_2^F \end{Bmatrix}^1 = \begin{Bmatrix} -40 \\ 20 \end{Bmatrix}, \quad \begin{Bmatrix} M_2^F \\ M_3^F \end{Bmatrix}^2 = \begin{Bmatrix} -80 \\ 80 \end{Bmatrix},$$

$$\begin{Bmatrix} M_3^F \\ M_4^F \end{Bmatrix}^3 = \begin{Bmatrix} -46{,}875 \\ 28{,}125 \end{Bmatrix}$$

Equivalent point load vector:

$$\{F\} = \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} -M_1^{F1} \\ -M_2^{F1} - M_2^{F2} \\ -M_3^{F2} - M_3^{F3} \\ -M_4^{F3} \end{Bmatrix} = \begin{Bmatrix} 40 \\ 60 \\ -33{,}125 \\ -28{,}125 \end{Bmatrix}$$

- Step 6: Determine the angle of rotation of the points:

$$EI\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1{,}167 & 0{,}25 & 0 \\ 0 & 0{,}25 & 1 & 0{,}25 \\ 0 & 0 & 0{,}25 & 0{,}5 \end{bmatrix}\begin{Bmatrix} \theta_1 \\ \theta_2 \\ \theta_3 \\ \theta_4 \end{Bmatrix} =$$

$$= \begin{Bmatrix} 40 \\ 60 \\ -33{,}125 \\ -28{,}125 \end{Bmatrix}$$

Solving system of equations, we get the rotation angle at the points:

$\theta_1 = 0$; $\theta_2 = 59{,}756/EI = 0{,}4256 \times 10^{-3}$ (rad)

$\theta_3 = -38{,}904/EI = -0{,}2771 \times 10^{-3}$ (rad);

$\theta_4 = -36{,}798/EI = -0{,}2621 \times 10^{-4}$ (rad)

- Step 7: Determine the bending moment at the elements end:

$$\begin{Bmatrix} M_1 \\ M_2 \end{Bmatrix}^1 = \frac{EI}{6}\begin{bmatrix} 4 & 2 \\ 2 & 4 \end{bmatrix}\begin{Bmatrix} 0 \\ 59{,}756 \end{Bmatrix}\frac{1}{EI} + \begin{Bmatrix} -40 \\ 20 \end{Bmatrix} =$$

$$= \begin{Bmatrix} -20{,}0783 \\ 59{,}8435 \end{Bmatrix} \text{(kNm)}$$

$$\begin{Bmatrix} M_2 \\ M_3 \end{Bmatrix}^2 = \frac{EI}{8}\begin{bmatrix} 4 & 2 \\ 2 & 4 \end{bmatrix}\begin{Bmatrix} 59{,}756 \\ -38{,}904 \end{Bmatrix}\frac{1}{EI} + \begin{Bmatrix} -80 \\ 80 \end{Bmatrix} =$$

$$= \begin{Bmatrix} -59{,}8435 \\ 75{,}4891 \end{Bmatrix} \text{(kNm)}$$

$$\begin{Bmatrix} M_3 \\ M_4 \end{Bmatrix}^3 = \frac{EI}{8}\begin{bmatrix} 4 & 2 \\ 2 & 4 \end{bmatrix}\begin{Bmatrix} -38{,}904 \\ 19{,}452 \end{Bmatrix}\frac{1}{EI} + \begin{Bmatrix} -19{,}1 \\ 80 \end{Bmatrix} =$$

$$= \begin{Bmatrix} -75{,}4891 \\ 0 \end{Bmatrix} \text{(kNm)}$$

3.2. Analysis of the three-span continuous beam problem using ANSYS simulation

** Perform the problem using the GUI method:*

- Start ANSYS

- Label the name of problem: Utility Menu > File > Change Jobname > DAMLIENTUC

- Label the title of problem: Utility Menu > File > Change Title > TINH TOAN NOI LUC DAM LIEN TUC BA NHIP

- Definition of elements and constants: For ANSYS V16 software, no BEAM3 element is available in the beam element selection directory. So we need to enter the command in the COMMAND window (Figure 2) to call this element and declare the real constant for it.

/PREP7

ET,1,BEAM3

R,1,0.18,0.0054,0.6

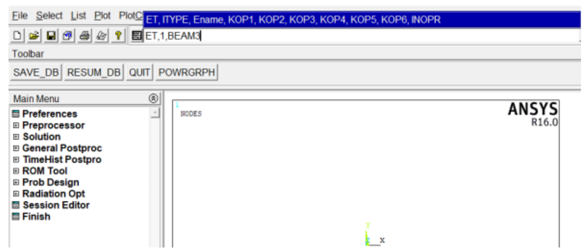


Figure 2. The window used to enter commands.

- Material definition: Main Menu > Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic > EX = 2.6E7 > PRXY = 0.2 > OK. Continue implementation Structural > Density > DENS = 0 > OK > Close the window (Figure 3)

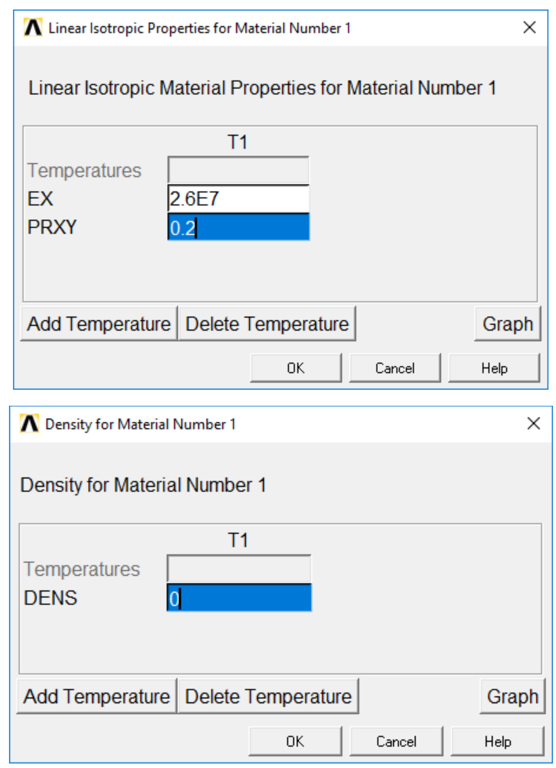


Figure 3. Material property declaration window.

- Creating geometric models:

+ Create points: Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS > In Create Keypoints window in Active CS (Figure 4) > enter NPT = 1, X = 0, Y = 0, Z = 0 > Apply > Similarly, enter points from 2 through 6 according to the table below:

NPT	X	Y	Z	NPT	X	Y	Z
1	0	0	0	4	14	0	0
2	2	0	0	5	17	0	0
3	6	0	0	6	22	0	0

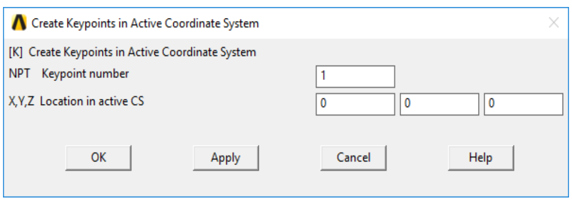


Figure 4. Window in global coordinate system.

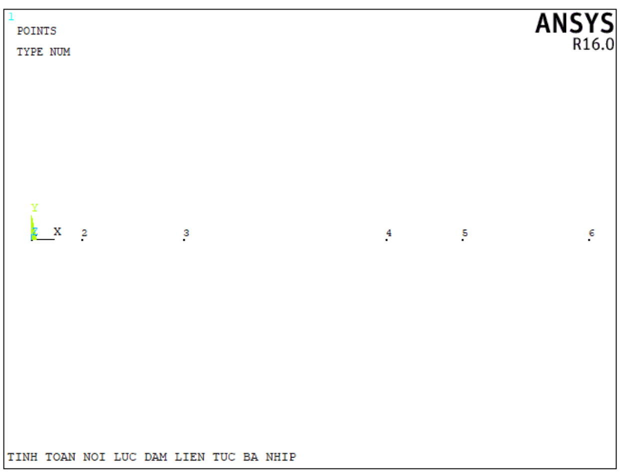


Figure 5. Create points.

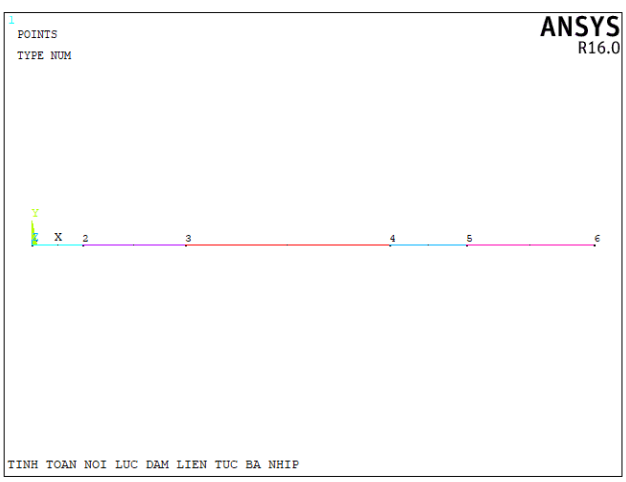


Figure 6. Create the lines.

The result of creating 6 points in the global coordinate system is shown in Figure 5.

+ Create the lines: Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line > Using your mouse to click on points 1 and 2, similar to 2 and 3, 3 and 4, 4 and 5, 5 and 6 > OK. The results are shown in Figure 6.

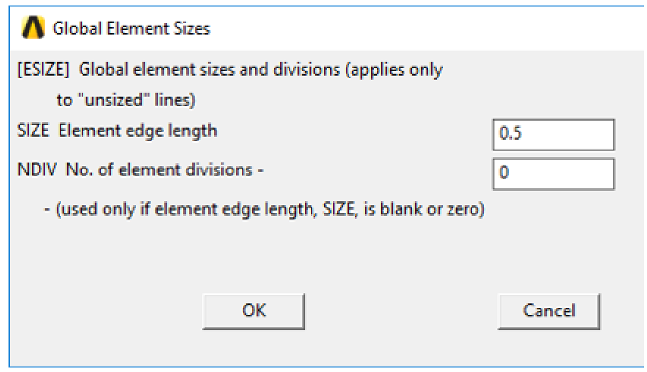


Figure 7. Defines the element sizes.

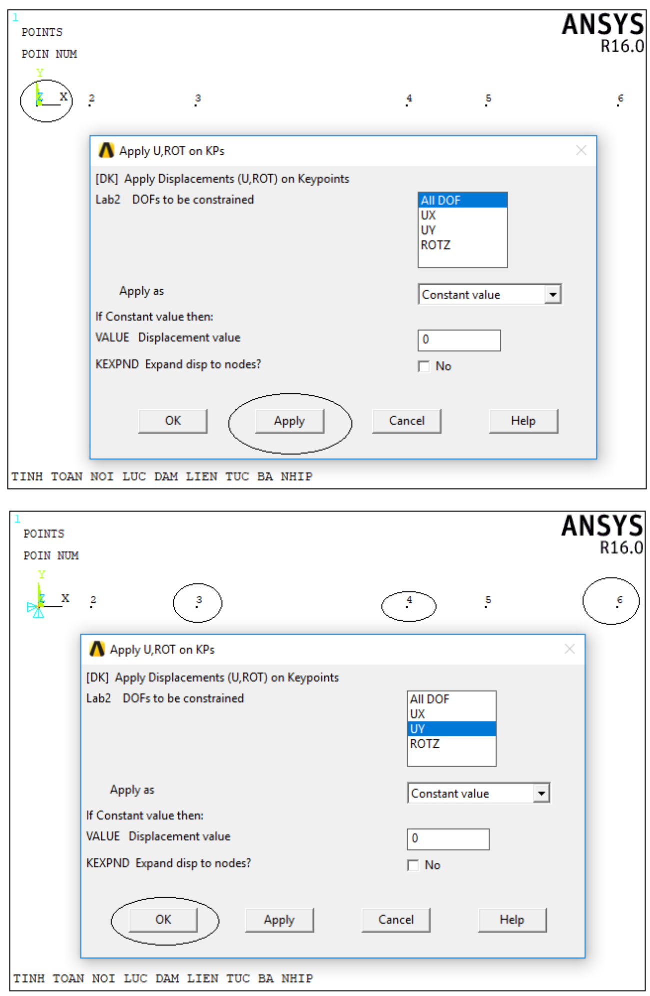


Figure 8. Assign boundary conditions.

- Creating a finite element model:

+ Defines the element sizes:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size > The Global Element Sizes window appears as shown in Figure 7 > SIZE = 0.5 > OK.

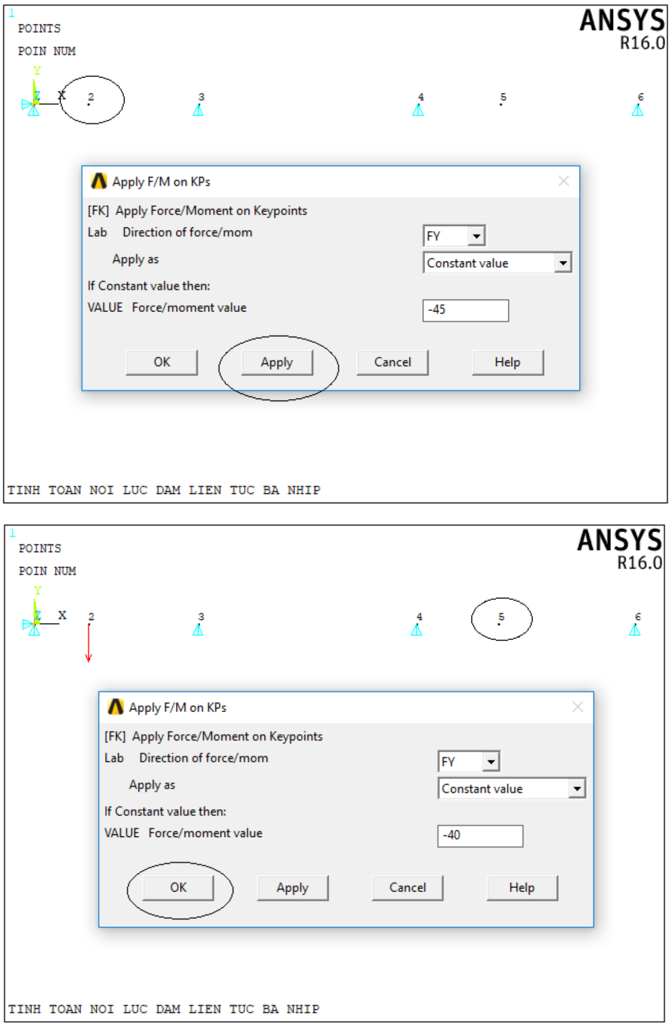


Figure 9. Assign a centralized load.

+ Divide elements:

Main Menu > Preprocessor > Meshing > Mesh > Lines > Picked all.

Main Menu > Finish.

- Assign boundary conditions and loads:

+ Assign gravitational acceleration: Main Menu > Solution > Define Loads > Apply > Structural > Inertia > Gravity > Global > ACELY = 10 > OK.

+ Assign a support boundary conditions:

Utility Menu > PlotCtrls > Numbering > click Keypoint numbers

Utility Menu > Plot > Keypoints

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Keypoints > click point 1 > Apply > Choose All DOF, VALUE = 0 > Apply > Continue using the mouse to select the points 3, 4 and 6 > OK > click UY, VALUE = 0 > OK (Figure 8).

+ Assign a centralized load:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints > click point 2 > Apply > Lab = FY, VALUE = -45 > Apply > Continue using the mouse to select the point 5 > OK > Lab = FY, VALUE = -40 > OK (Figure 9).

+ Assign the load to be distributed at the center span of the beam:

Utility Menu > Plot > Lines

Utility Menu > PlotCtrls > Numbering > Choose Line numbers

Utility Menu > Select > Entities > Select Entities > Lines > By Num/Pick > From Full > OK > click L3 > OK.

Utility Menu > Select > Entities > Select Entities > Elements > Attached to > Lines > Reselect > OK.

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Beams > Picked all > LKEY = 1 > VALI = 15 > OK (Figure 10).

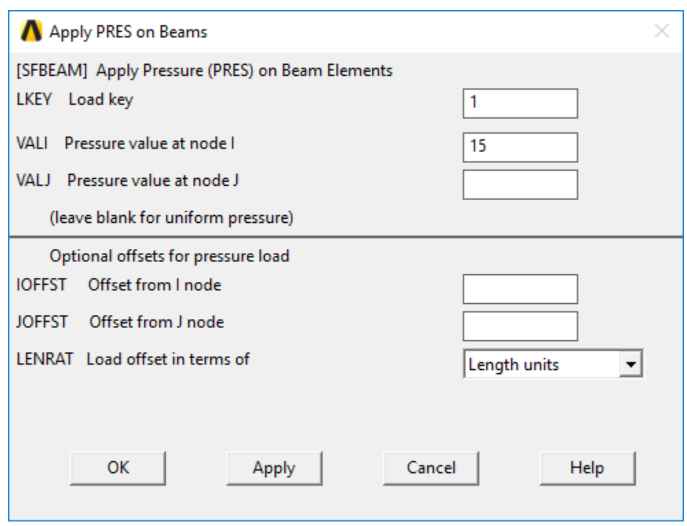


Figure 10. Assign a distributed load.

- Analyze and calculate options set:

Utility Menu > Select > Everything.

Main Menu > Solution > Solve > Current LS.

- Calculation results:

+ Display of internal force diagram:

General Postproc > Element Table > Define Table > Add > Define Additional Element Table Items as Figure 11 > Lab = QI > By sequence num > SMISC > SMISC,2 > Apply > Lab = QJ > By sequence num > SMISC > SMISC,8 > Apply > Lab = MI > By sequence num > SMISC > SMISC,6 > Apply > Lab = MJ > By sequence num > SMISC > SMISC,12 > OK.

General Postproc > Contour Plot > Line Element Result > Plot Line-Element Results > LabI = QI, LabJ = QJ > Fact = 1 > OK, Shear force diagram as shown in Figure 12.

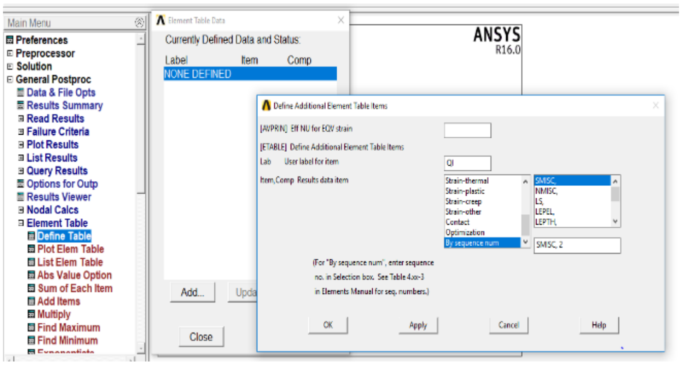


Figure 11. Display element results.

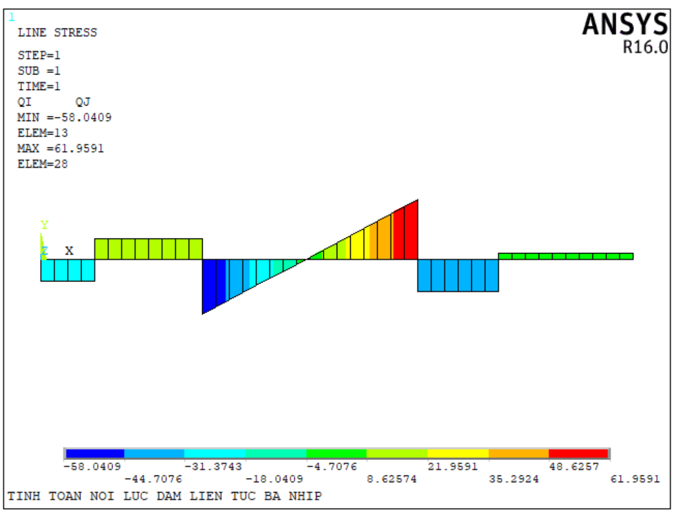


Figure 12. Shear force diagram.

General Postproc > Contour Plot > Line Element Result > Plot Line-Element Results > LabI = MI, LabJ = MJ > Fact = -1 > OK, bending moment diagram as Figure 13.

+ Check the results of the rotation angle:

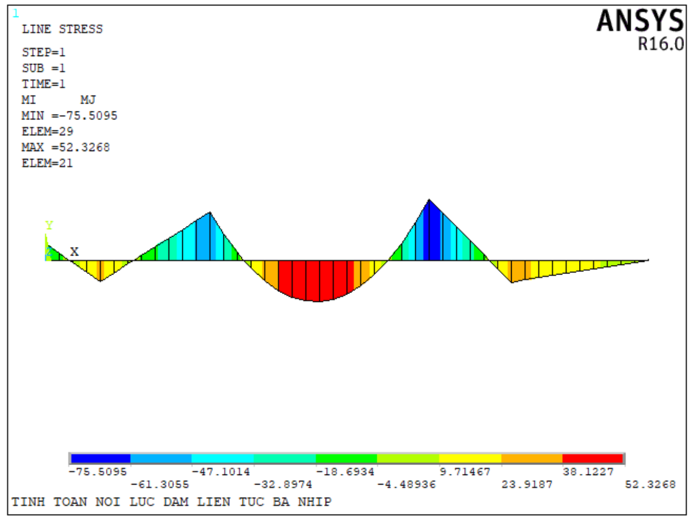
Utility Menu > Plot > Keypoints

Utility Menu > PlotCtrls > Numbering > Choose Keypoint numbers

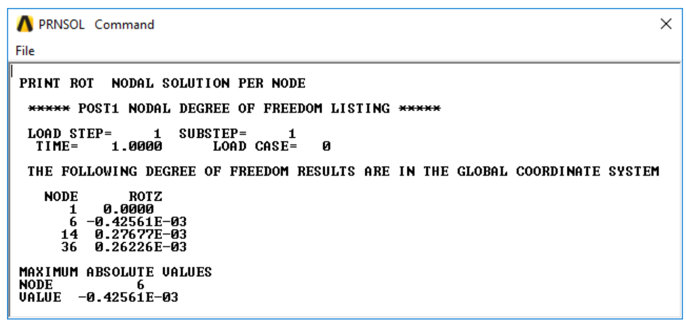
Utility Menu > Select > Entities > in window, Select Entities > Keypoints > By Num/Pick > From Full > OK > Click points 1, 3, 4, 6 > OK.

Utility Menu > Select > Entities > in window, Select Entities > Nodes > Attached to > Keypoints > Reselect > OK.

General Postproc > List Results > Nodal Solution > List Nodal Solution > Choose Nodal Solution > DOF Solution > Z-Component of rotation > OK, The results obtained the rotation angle at the points as shown in Figure 14.



Figur 13. Bending moment diagram.

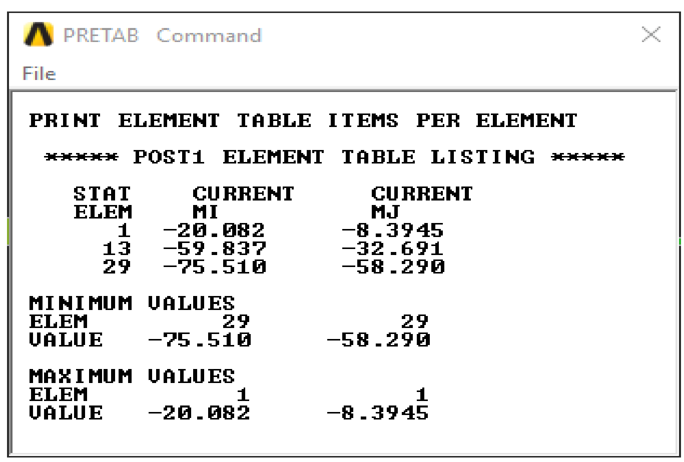


PRNSOL Command

File

```
PRINT ROT   NODAL SOLUTION PER NODE

 ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

 LOAD STEP=     1  SUBSTEP=     1
  TIME=    1.0000      LOAD CASE=   0

 THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

    NODE        ROTZ
       1    0.0000
       6  -0.42561E-03
      14   0.27677E-03
      36   0.26226E-03

MAXIMUM ABSOLUTE VALUES
NODE           6
VALUE   -0.42561E-03
```

Figure 14. Angle displacement at points.



PRETAB Command

File

```
PRINT ELEMENT TABLE ITEMS PER ELEMENT

 ***** POST1 ELEMENT TABLE LISTING *****

    STAT     CURRENT      CURRENT
    ELEM     MI           MJ
       1    -20.082      -8.3945
      13    -59.837      -32.691
      29    -75.510      -58.290

MINIMUM VALUES
ELEM          29            29
VALUE     -75.510       -58.290

MAXIMUM VALUES
ELEM           1             1
VALUE     -20.082       -8.3945
```

Figure. 15. Bending moment values at the elements.

Result of calculating the displacement of rotation at positions 1, 3, 4, 6 in Figure. 14 and the calculation results given in the example at positions 1, 2, 3, 4 are approximately the same.

+ Check the bending moment result:

The value of bending moment calculated by software at positions 1, 3, 4 in Figure 15 is also approximately the same as the results of manual calculation at positions 1, 2, 3 in the three-span continuous beams.

Comment: FEM and ANSYS methods match.

*** Perform the problem according to APDL parameter:***

```
FINI
/CLEAR
/FILNAME,DAMLIENTUC
/TITLE,TINH TOAN NOI LUC DAM LIEN TUC BA NHIP
! Enter the geometry parameter
A=2             $B=4     $C=3     $D=5
L1=A+B $L2=8    $L3=C+D
BD=0.3          $HD=0.6
! Enter the load parameter
F1=45           $F2=40      $Q=15
! Enter the material parameter
E=2.6E7         $M=0.2
$G=2.5  ! Enter 0 when the self weight of the beam is not taken into account
/PREP7
ET,1,BEAM3      ! 2-D beam element
R,1,BD*HD,BD*HD*HD*HD/12,HD
MP,EX,1,E                ! Material definition
MP,PRXY,1,M              $MP,DENS,1,G
K,1,0,0,0 $K,2,A,0,0     $K,3,L1,0,0     K,4,L1+L2,0,0
          $K,5,L1+L2+C,0,0 $K,6,L1+L2+L3,0,0
LSTR,1,2  $LSTR,2,3      $LSTR,3,4
LSTR,4,5  $LSTR,5,6
ESIZE,0.5 $LMESH,ALL
FINISH
/SOLU
ACEL,0,10,0     ! Gravitational acceleration
DK,1,ALL,0      ! Boundary conditions at fixed
DK,3,UY,0       !Roller
DK,4,UY,0       $DK,6,UY,0
FK,2,FY,-F1     ! Assign the centralized load
FK,5,FY,-F2
LSEL,S,,,3 ! Assign the distribution force
ESLL,R
SFBEAM,ALL,1,PRES,Q
ALLSEL   $SOLVE
FINISH
/POST1
ETABLE,QI,SMISC,2
ETABLE,QJ,SMISC,8
ETABLE,MI,SMISC,6
ETABLE,MJ,SMISC,12
PLLS,QI,QJ,1,0,0     !Draw the shear force diagram
PLLS,MI,MJ,-1,0,0   ! Draw the moment diagram
KSEL,S,,,1      $KSEL,A,,,3
KSEL,A,,,4              $KSEL,A,,,6
NSLK,R
PRNSOL,ROT,Z
ALLSEL
ESEL,S,,,1      $ESEL,A,,,13
ESEL,A,,,29
PRETAB,MI,MJ
ALLSEL
FINISH
```

Comment: With the use of the ADPL parameter method, it is possible to use this program code to investigate the effects of different initial parameters. However, using the ADPL parameter, we should know the functions of each command line.

4. Conclusions

Based on the results of the study lead to the following conclusions:

1. Through research shows that with the use of simulation software ANSYS gives quite accurate results with other methods, when using

the ADPL parameter method, it is easy to examine the initial parameters affecting the continuous beam.

2. ANSYS is a multi-environment, multi-structural simulation software, etc., and needs to be studied extensively, especially in the field of civil engineering, they can simulate the working of structures, thermal processes, interact with each other.

3. In ANSYS, the three-span continuous beam problem can create reinforced concrete elements, in order to calculate the reinforced concrete structure in the beams calculation.

References

[1] Leu Tho Trinh, *Structural mechanics, vol. 2: Statically indeterminate structures*. Science and Technology Publishing House, 2010

[2] Pham Van Nam, *FEM for the continuous beam problem*. Master thesis, Hai Phong University, 2017.

[3] Nguyen Xuan Bao, Pham Hong Giang, Vu Thanh Hai, Nguyen Van Le. *FEM applied in calculating constructions, vol 1.* Agricultural Publishing House, 1983.

[4] Chu Quoc Thang, *Finite element method*. Science and Technology Publishing House, 1997.

[5] Lam Thanh Quang Khai, *FEM in calculating the internal force of the planar cycloid arch system*. Journal of Construction Science and Technology, 1, 10-16, 2017.

[6] Pham Van Dat, *Calculate the bar structure according to the FEM*. Construction Publishing House, 2017.

[7] Ji-Qing Jiang, Wei-Qiu Chen and Yih-Hsing Pao, *Reverberation-ray analysis of continuous Timoshenko beams subject to movingloads.* Journal of Vibration and Control, 18(6), 774–784. DOI: 10.1177/1077546310397562

[8] Abdulamir Atalla Almayah, *Simplified Analysis of Continuous Beams.* International journal of applied engineering research, 13(2), 922-928, 2018

[9] ANSYS, *Documentation for ANSYS*. Inc Theory reference, Release 16.0

[10] Lam Thanh Quang Khai, Do Thi My Dung, Vu Hoang Hung, *Analysis of construction structure according to FEM and application in ANSYS software,* Construction Publishing House, 2019.

[11] Vu Hoang Hung, *ANSYS – Practical example of structural analysis of irrigation works, and hydropower*. Construction Publishing House, 2018.

[12] Vu Hoang Hung, Nguyen Quang Hung, *ANSYS structural analysis of irrigation and hydropower works, vol 1: basic problems*, Construction Publishing House, 2012.