ASIFEM3: Finite element code for framed structures using ASI technique (cubic beam element version)

(Download site of the analysis code: http://www.kz.tsukuba.ac.jp/~isobe/asifem3_e.html)

Procedure of making input data

1. Draw the model you want to analyze, and decide the number of element subdivision per member. Please note that only one element per member is needed when using the ASI technique for cubic beam element. Next, assign the node numbers and element numbers.
2. Fix the coordinates of each node in global Cartesian coordinate, and write down in input data.
3. Fix the material and sectional properties of structural members, and write down in input data.
4. Fix one reference node per one element and write down the coordinates in input data. (Ref. node sets the direction of elemental section. See Fig. 1.)
5. Write down the elemental data. (Node No. of two connected nodes, ref. node No., material and sectional property No.)
6. Fix the boundary conditions and write down in input data.
7. Write down the total number of elements, nodes, calculation step number and output step number.
8. Fix and write down other parameters for the analysis. (ex. time increment, flags)
9. Set load conditions (including consideration of gravity).

With the upper procedure, gather up all the input data in a file named “input3.txt” (see next page).

Execution procedure

Place the input data “input3.txt” in the same folder with asifem3.exe, and double click the execution file. An analysis will be executed and three files named as “output.txt”, “post.out”, “eqs.out” will be made. All the input data and outputs will be printed out in “output.txt”, and you can check the input errors by checking this file. “post.out”, “eqs.out” are the files used for the graphic software “Graphic.exe”. You can check the outputs graphically by double clicking “Graphic.exe”. Please read “manual(Graphic.exe)_e.txt” for the operation.

How to handle errors

1. Input data is not printed out completely in “output.txt”.

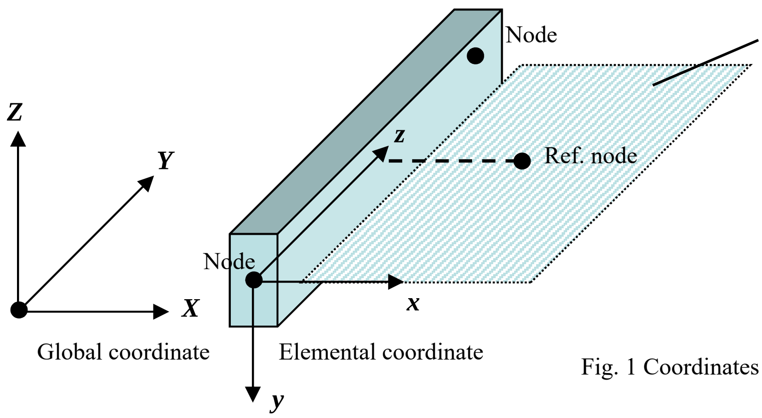
→If the data for the printed out area is all right, you can check the next input data which is not printed out in “output.txt”. It is possible that there are input errors in that area.

2. Input data is completely printed out in “output.txt”, however, outputs are not printed out.

→ The file “post.out” for the constructed model will be printed out by setting the error message output flag on. By viewing the model using “Graphic.exe”, you can check if there is any problem in the model itself.

3. There seems to be no problem in the model itself by checking the graphic software.

→Please read the ”Searching for input errors” section in the “output.txt” printed out when the error message output flag is on. Are all the elements confirmed as “O.K.” ? If not, please check the elemental data which is not ”O.K.”. Especially, there is a possibility that reference node coordinates are not properly fixed.



Assignment area for a reference node

Locate the node in the area further away from *z*-axis and vertically from *y*-axis of the elemental coordinate system. This figure shows the case for a member section when the moment area inertia around *x*-axis is larger than that around *y*-axis.

Fig. 1 Coordinates and location of a reference node

Outline of input data (input3 txt) of FEM code for framed structures (ASIFEM3, cubic beam element version)

8 8 : Total No. of elements, total No. of nodes

1000 100 : No. of calculation steps, No. of output steps

2 2 2 2 1 2 2 1 0 0 : Scheme flag, incremental theory flag, deformation theory flag, elastic/elasto-plastic analysis flag, mass matrix flag, static/dynamic analysis flag, truss/rahmen flag, gravity flag, error message output flag, precision flag (see * for details)
0.44444d+00 0.83333d+00 1.0d-03 : Integration parameters β, δ for Newmark's β method, time increment [s]

1000 1 : No. of steps in loading, No. of nodes in loading
1 1.00d+05 0.0d+00 0.0d+00 0.0d+00 0.0d+00 0.0d+00 : Node No. in loading, subjected forces [N] in *X*, *Y*, *Z* directions of global coordinate system, subjected moments [Nmm] around *X*, *Y*, *Z* -axes

1 : No. of material and sectional properties
2.060d+05 3.000d-01 2.450d+02 2.060d+03 7.900d-06
: Young's modulus [MPa], Poisson's ratio, yield stress [MPa], tangent modulus after yielding [MPa], density [kg/mm^3]

2.500d+03 2.083d+06 1.302d+05 4.391d+05 6.250d+04 1.563d+04
: Cross-sectional area [mm^2], moment of area inertia around *x*-axis [mm^4], moment of area inertial around *y*-axis [mm^4], torsional coefficient [mm^4], plastic section modulus for *x*-direction [mm^3], plastic section modulus for *y*-direction [mm^3]

24 : No. of fixed degrees of freedom
5 1 5 2 5 3 5 4 5 5 5 6 : (Fixed node No., fixed freedom No.), (Fixed node No., fixed freedom No.), ・・・
6 1 6 2 6 3 6 4 6 5 6 6 (Total of 24 pairs of data are aligned in this example)
7 1 7 2 7 3 7 4 7 5 7 6
8 1 8 2 8 3 8 4 8 5 8 6

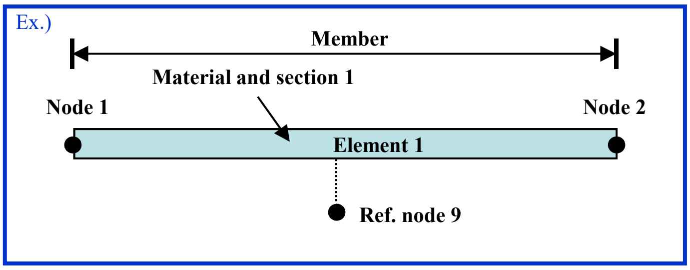
1 2 9 1
2 3 10 1
3 4 11 1
4 1 12 1
5 1 13 1
6 2 14 1
7 3 15 1
8 4 16 1

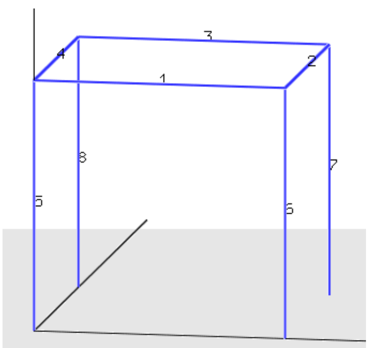
Connection data of elements
: Node No. (located at the start end of an element), node No. (located at the last end of an element), ref. node No., material and sectional property No.
(Aligned from the upper row in order as: 1st element, 2nd element, ・・・, for 8 elements in this example)

1 0.0000d+00 0.0000d+00 2.0000d+03
2 2.0000d+03 0.0000d+00 2.0000d+03
3 2.0000d+03 2.0000d+03 2.0000d+03
4 0.0000d+00 2.0000d+03 2.0000d+03
5 0.0000d+00 0.0000d+00 0.0000d+00
6 2.0000d+03 0.0000d+00 0.0000d+00
7 2.0000d+03 2.0000d+03 0.0000d+00
8 0.0000d+00 2.0000d+03 0.0000d+00
9 1.0000d+03 -1.0000d+03 2.0000d+03
10 3.0000d+03 1.0000d+03 2.0000d+03
11 1.0000d+03 3.0000d+03 2.0000d+03
12 -1.0000d+03 1.0000d+03 2.0000d+03
13 0.0000d+00 -1.0000d+03 1.0000d+03
14 2.0000d+03 -1.0000d+03 1.0000d+03
15 2.0000d+03 3.0000d+03 1.0000d+03
16 0.0000d+00 3.0000d+03 1.0000d+03

Coordinates of each node
: Node No., *X*,*Y*,*Z* coordinates in global coordinate system [mm]
(1st ~8th rows are for normal nodes, 9th~16th rows are for ref. nodes in this example)

Attention: 1.0000d+03 is an expression for double precision of 1.0×10^3





*Details of the flags

Scheme flag	: 1=Conventional method 2=ASI technique
Incremental theory flag	: 1=TLF 2=ULF
Deformation theory flag	: 1=Infinitesimal deformation 2=Finite deformation
Elastic/elasto-plastic analysis flag	: 1=Elastic analysis 2=Elasto-plastic analysis
Mass matrix flag	: 1=Consistent mass matrix 2=Lumped mass matrix
Static/dynamic analysis flag	: 1=Static analysis 2=Dynamic analysis
Truss/rahmen flag	: 1=Truss structure 2=Rahmen structure
Gravity flag	: 0=No gravity 1=Considering gravity
Error message output flag	: 0=No output 1:Error message output
Precision flag	: 0=High 1:Midium high 2:Medium 3:Low

Model of this input data
(No.: element number)