ASIFEM: Finite element code for framed structures using ASI-Gauss technique

(Download site of the analysis code: http://www.kz.tsukuba.ac.jp/~isobe/asifem_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 two elements per member is preferred when using the ASI-Gauss code. In that case, subdivide with two elements between two structural joints in order to obtain highly accurate solutions. Mid-node is to be located at the center of the member. Next, assign the node numbers and element numbers.
2. Fix the coordinates of each node in global Cartesian coordinate system, 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 element and write down the coordinates in input data. As shown in Fig. A.1, the reference nodes will set the elemental coordinate system.
5. Write down the connection data of elements (Node No. of two nodes constituting an element, reference node No., material and sectional property No., pair element 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 numerical conditions for the analysis (Ex. time increment, flags).
9. Set load conditions including consideration of gravity.

After following this procedure, gather up all the input data and save in a file named “input.txt” (see next page).

Execution procedure

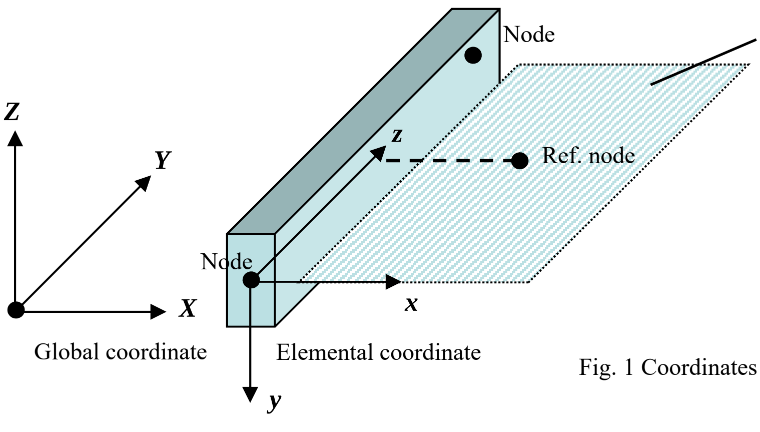
Place the input data “input.txt” in the same folder with “asifem.exe”, and double click the execution file. When the calculation is successfully executed, three files named as “output.txt”, “post.out” and “eqs.out” will be made. All the input data and outputs will be printed out in “output.txt”, and so you can check for the input errors by reading this file. “post.out” and “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 your input data is not printed out completely in the file “output.txt”, while the data in the printed out area is all right, there might be errors at the next section in the input data.

2. Input data is completely printed out in “output.txt”, however, outputs are not printed out.
→ Set the error message output flag on. The file “post.out” for the constructed model will be printed out so that you can view and check the model by using “Graphic.exe”.

3. There seems to be no problem in the model itself by checking the graphic software.
→ Read the “Searching for input errors” section in the file “output.txt” which will be printed out when the error message output flag is on. Are all the elements confirmed as “O.K.”? If not, check the element information which is not “O.K.”. It is highly possible that the coordinates for reference nodes are not properly set.



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

<u>Outline of input data (input txt) of FEM code for framed structures (ASIFEM)</u>

16 16 : Total No. of elements, total No. of nodes

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

3 2 2 2 1 2 2 1 0 2 : 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]

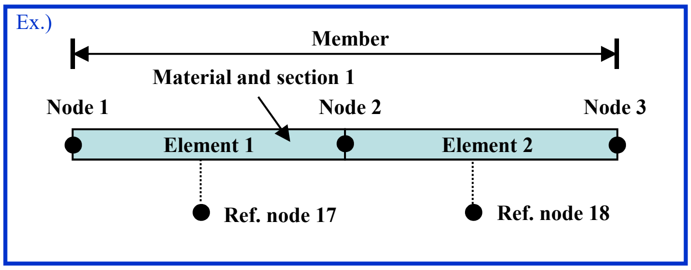
24 : No. of fixed degrees of freedom

10 1 10 2 10 3 10 4 10 5 10 6
12 1 12 2 12 3 12 4 12 5 12 6
14 1 14 2 14 3 14 4 14 5 14 6
16 1 16 2 16 3 16 4 16 5 16 6

: (Fixed node No., fixed freedom No.), (Fixed node No., fixed freedom No.), ･･･ (Total of 24 pairs of data are aligned in this example)

2 1 17 1 2
2 3 18 1 1
4 3 19 1 4
4 5 20 1 3
6 5 21 1 6
6 7 22 1 5
8 7 23 1 8
8 1 24 1 7
9 1 25 1 10
9 10 26 1 9
11 3 27 1 12
11 12 28 1 11
13 5 29 1 14
13 14 30 1 13
15 7 31 1 16
15 16 32 1 15

Connection data of elements (specialized for ASI-Gauss technique)
: Node No. (located at the middle of a member), node No. (located at the end of a member), ref. node No., material and sectional property No., pair element No.
(Aligned from the upper row in order as: 1st element, 2nd element, ･･･, for 16 elements in this example)



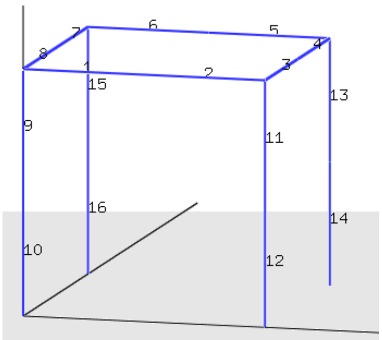
1 0.0000d+00 0.0000d+00 2.0000d+03
2 1.0000d+03 0.0000d+00 2.0000d+03
3 2.0000d+03 0.0000d+00 2.0000d+03
4 2.0000d+03 1.0000d+03 2.0000d+03
5 2.0000d+03 2.0000d+03 2.0000d+03
6 1.0000d+03 2.0000d+03 2.0000d+03
7 0.0000d+00 2.0000d+03 2.0000d+03
8 0.0000d+00 1.0000d+03 2.0000d+03
9 0.0000d+00 0.0000d+00 1.0000d+03
10 0.0000d+00 0.0000d+00 0.0000d+00
11 2.0000d+03 0.0000d+00 1.0000d+03
12 2.0000d+03 0.0000d+00 0.0000d+00
13 2.0000d+03 2.0000d+03 1.0000d+03
14 2.0000d+03 2.0000d+03 0.0000d+00
15 0.0000d+00 2.0000d+03 1.0000d+03
16 0.0000d+00 2.0000d+03 0.0000d+00
17 5.0000d+02 -5.0000d+02 2.0000d+03
18 1.5000d+03 -5.0000d+02 2.0000d+03
19 2.5000d+03 5.0000d+02 2.0000d+03
20 2.5000d+03 1.5000d+03 2.0000d+03
21 1.5000d+03 2.5000d+03 2.0000d+03
22 5.0000d+02 2.5000d+03 2.0000d+03
23 -5.0000d+02 1.5000d+03 2.0000d+03
24 -5.0000d+02 5.0000d+02 2.0000d+03
25 0.0000d+00 -5.0000d+02 1.5000d+03
26 0.0000d+00 -5.0000d+02 5.0000d+02
27 2.0000d+03 -5.0000d+02 1.5000d+03
28 2.0000d+03 -5.0000d+02 5.0000d+02
29 2.0000d+03 2.5000d+03 1.5000d+03
30 2.0000d+03 2.5000d+03 5.0000d+02
31 0.0000d+00 2.5000d+03 1.5000d+03
32 0.0000d+00 2.5000d+03 5.0000d+02

Coordinates of each node
: Node No., X, Y, Z coordinates in global coordinate system [mm]
(1st ~16th rows are for normal nodes, 17th~32nd rows are for ref. nodes in this example)

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

<u>*Details of the flags</u>

Scheme flag	: 1=Conventional method 2=ASI technique 3=ASI-Gauss 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)