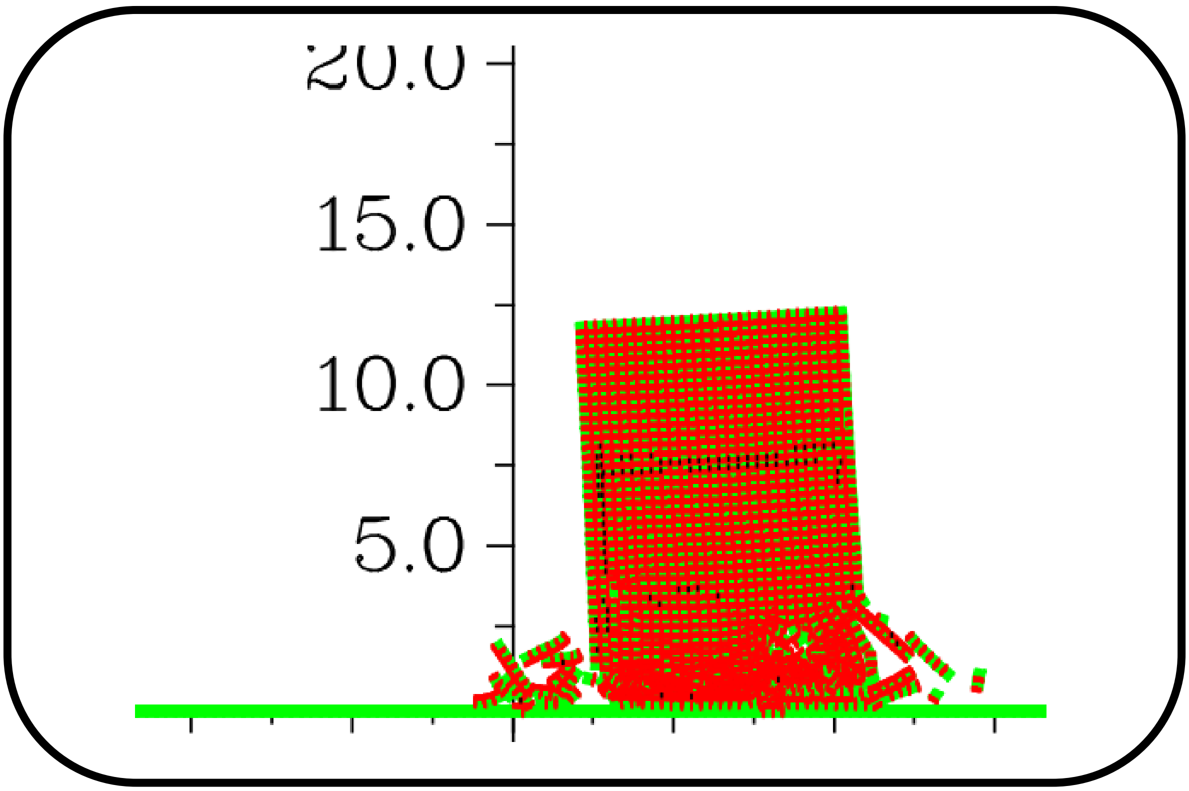
User Manual
Complete Structural Analysis - Applied Element Method (CSA-AEM)



January 22, 2021

1. Introduction

2. Input File

The input file should be prepared in .txt format. The name of the file should be ***input.aem*** and placed in the same folder along with the CSA-AEM Fortran Code.

2.1 GEOMETRY

The first section of the input file consists of the inputs related to the geometry of the structure. It includes the reference coordinates of the structure, element size and no. of elements. The section should start with the string named **GEOMETRY** and this section should be always at the top of the input file.

The **GEOMETRY** section has following syntax.

Section/ Sub-section (String)	**Properties (Numerical Values)**					
GEOMETRY						
DSIZE	DSIZE					
COORD	X1COR	Y1COR	X2COR	Y2COR	NUMX	NUMY
COORD	X1COR	Y1COR	X2COR	Y2COR	NUMX	NUMY
….	….	….	….	….	….	….

DSIZE: The second row of the GEOMETRY section consists the **DSIZE** sub-section. The **DSIZE** is the size of the square shaped elements in meter.

COORD: The third row onwards in the GEOMETRY section consists of the **COORD** subsection. The no. of rows for the **COORD** sub-section depends on the no. of blocks used in the structure. The element numbering is done from left to right and bottom to top manner, so the coordinates shall also be in the same sequence. The **COORD** sub-section has six columns defined as follows:

X1COR: x coordinate of lower left corner of the respective block (in meter).

Y1COR: y coordinate of lower left corner of the respective block (in meter).

X2COR: x coordinate of upper right corner of the respective block (in meter).

Y2COR: y coordinate of upper right corner of the respective block (in meter).

NUMX: Number of elements in X direction.

NUMY: Number of elements in Y direction.

2.2 MATDEF

The properties of the materials used in the structures are defined in the **MATDEF** section. The first row of this section should start with the string **MATDEF**. This section includes the Young's modulus, poisons ratio, tension resistance, compression resistance, number of connected springs, density, damping ratio and thickness. The **MATDEF** section has following syntax.

Section/ Sub-section (String)	**Properties (Numerical Values)**									
MATDEF										
MAT	MATID	PNORMALK	POIS	TENS	COMP	NPSS	GRAV	DAMPING	T	COR
MAT	MATID	PNORMALK	POIS	TENS	COMP	NPSS	GRAV	DAMPING	T	COR
….	….	….	….	….	…..	….	….	….	….	….

MAT: MAT is the only subsection under MATDEF section. Each row of the MAT sub-section represents the material properties of the corresponding material. Each row should start with the string **MAT.** The **MAT** sub-section has 10 columns defined as follows:

MATID: Material ID number. It should start with 1 for the first material, 2 for second and so on.

PNORMALK: Youngs Modulus (E) (N/m^2 for static and in dynamics KN/m^2)

POIS: Poisons ratio

TENS: Tension resistance (N/m^2 for static and KN/m^2 in dynamics) (For elastic analysis TENS=0)

COMP: Compression resistance (N/m^2 for static and KN/m^2 in dynamics) (if zero , Young's modulus is constant)

NPSS: Number of connecting springs between adjacent faces

GRAV: Density of the material (Kg/m^3 for static and t/m^3 in dynamics)

DAMPING: Damping ratio

T: Thickness of the structure (in meter)

COR:

2.3 PARAMS

The parameters used for the structural analysis are defined in the **PARAMS** section. The first row of the PARAMS section should start with the string **PARAMS.** This section includes the definitions regarding planestate condition, geometric residual, scale, numerical damping, writeout condition, and poison effect provision. The PARAMS section has following syntax.

Section/ Sub-section (String)	Properties	
PARAMS		(change this column only)
SET	PLANESTATE	STRESS or STRAIN
SET	GEOMRES	ON or OFF
SET	SCALE	SCALE (Numerical value)
SET	ECHO	ON or OFF
SET	NUMDAMP	ON or OFF
SET	WRITEOUT	ON or OFF
SET	POISONEFFECT	ON or OFF
SET	ELEMENTMEASURE	NDOF

SET: SET is the subsection of the PARAMS section. Each **SET** sub-sections has different parameters definition. The first column of each row of this sub-section should start with string **SET**. The SET subsection has 2 columns; first is the name of the parameter and another column is the definition of the parameters. Only the column with the definition of the parameter should be modified according to the type of structural analysis problem. Each variable under this sub-section is defined as follows.

PLANESTATE: Define the planestate condition. It can be either STRESS or STRAIN.

GEOMRES: Define whether you want the geometric residual ON or OFF.

SCALE: Scaling factor

ECHO: There are flags for debugging the CSA-AEM code. You can ON or OFF the flags from this row.

NUMDAMP: Define whether numerical damping is ON or OFF. If NUMDAMP is on, it uses γ and β as 0.7 and 0.36 (Hilber-Hughes-Taylor (HHT) Method) and if it is OFF, it will take 0.5 and 0.25 (Newmark Beta).

WRITEOUT: Define whether writeout is ON or OFF.

POISONEFFECT: Define whether you want to consider Poison effect or not with ON and OFF.

NDOF: Element number where outputs have to be recorded

2.4 MATASSIGN

The assigning of the defined material properties in Section 2.2 will be assign to the respective elements in **MATASSIGN** section. The first row of the **MATASSIGN** section should start with the string **MATASSIGN.** This section also includes whether we are considering shear modeling of soil element or not. The **MATASSIGN** section has following syntax.

Section/ Sub-section (String)	Properties

MATASSIGN					
MAS	I1	I2	INC	MATID	SOIL or NOSOIL
MAS	I1	I2	INC	MATID	SOIL or NOSOIL
…	…	…	…	…	…

MAS: MAS is the only subsection under MATASSIGN section. Each row of the material assigning should start with the string **MAS** and followed by the element numbers, increment, material ID and soil modeling condition. Each variable under this sub-section is defined as follows.

I1: First element to which the material properties is assigned.

I2: Second element upto which the same material is assigned.

INC: Increment of number of elements while assigning the material properties. If you want to assign the material properties for all the elements from I1 to I2, then INC should be 1.

MATID: Material ID number which is being assigned to those elements. Select the MATID no. from section 2.2.

SOIL or NOSOIL: If you are doing soil shear modeling write SOIL in the Last column of this sub-section otherwise it should be NOSOIL.

2.5 BOUNDARYASSIGN

The boundary condition of the structure under analysis is assigned in **BOUNDARYASSIGN** section. The first row of the **BOUNDARYASSIGN** section should start with the string **BOUNDARYASSIGN**. This section includes element numbers with their corresponding boundary condition. The **BOUNDARYASSIGN** section has following syntax.

Section/ Sub-section (String)	**Properties**					
BOUNDARYASSIGN						
BC	I1	I2	INC	IX	IY	IR
BC	I1	I2	INC	IX	IY	IR
…	…	…	…	…	…	

BC: BC is the only subsection under **BOUNDARYASSIGN** section. Each row of this sub-section should should start with the string **BC** and followed by the element numbers, increment, and boundary condition in x, y dir. And rotation. Each variable under this sub-section is defined as follows.

I1: First element number to which the boundary condition is assigned.

I2: Second element number upto which the boundary condition is assigned.

INC: Increment of number of elements while assigning the boundary condition. If you want to assign the boundary condition for all the elements from I1 to I2, then INC should be 1.

IX: It is the boundary condition code for x-direction. The code should be 0 for free, 1 for restrained, -1 , if X axis is the Normal direction to the T. Boundary.

IY: It is the boundary condition code for y-direction. The code should be 0 for free, 1 for restrained, -1 , if Y axis is the Normal direction to the T. Boundary.

IR: It is the boundary condition code for Rotation. The code should be 0 for free, 1 for restrained.

2.6 REBAR

The properties of rebar and its assigning is done under the **REBAR** section. The first row of the **REBAR** section should start with the string **REBAR**. In this section, we define the orientation and position of the rebars, Young's Modulus, Yield Stress and the reinforcement area. There are two subsections in **REBAR** section. The **REBAR** section has following syntax.

Section/ Sub-section (String)	**Properties**						
REBAR							
STEELFAIL	NSTFAILCODE (0 or 1)						
STEEL	ST_TYPE (V or H)	COOR	CRFTMIN	CRFTMAX	YMOD	YIELD	ASTEEL
STEEL	ST_TYPE (V or H)	COOR	CRFTMIN	CRFTMAX	YMOD	YIELD	ASTEEL
…	…	…	…	…	…	…	…

STEELFAIL: **STEELFAIL** is the first sub-section of the REBAR subroutine. It will have only one row. It defines whether the steel failure is permitted or not. **NSTFAILCODE** will be 0 for no steel failure permitted and 1 for steel failure permitted.

STEEL: STEEL is the second sub-section under the REBAR section. It has orientation and position of the rebars, Young's Modulus, Yield Stress and the reinforcement area. For each rebars there should be one row of **STEEL** sub-section. The variables used in this sub-sections are defines as follows.

ST_TYPE: V for vertical reinforcement and H for Horizontal reinforcement.

COOR: If the ST_ TYPE is V then x-coordinate and if ST_TYPE is H then y-coordination of the rebar position.

CRFTMIN: Lower end identifier; coordinate of lower end of rebar. (0 if infinity).

CRFTMAX: Upper end identifier, coordinate of upper end of rebar. (0 if infinity).

YMOD: Youngs modulus (N/m2) of steel defined in the particular row.

YIELD: Yield stress(N/m2) of the steel defined in the particular row.

ASTEEL: Area (m2) of steel defined in the particular row.

2.7 LOADDEF

The loading properties of the structure is defined in the **LOADDEF** section. The first row of this section should start with the string **LOADDEF**. The **LOADDEF** section has following syntax.

Section/ Sub-section (String)	**Properties**	
LOADDEF		(change this column only)
SET	NLOADCASES	NLOADCASES (in numbers)
SET	LDTYPE	DYN/STA/CYC
SET	DSTYPE	FOR/DIS/ACC
SET	NINC	NINC (in numbers)
SET	NUNLOT	1 or 2
SET	SELFWGT	nselfweightTemporary
SET	DISPMAX	DISPMAX (in numbers)
SET	DRAWSECT	Drawsect (in numbers)
SET	ELEMENTMEASURE	NDOF

SET: SET is the subsection of the LOADDEF section. Each **SET** sub-sections has different loading parameters' definition. The first column of each row of this sub-section should start with string **SET**. The SET subsection has 2 columns; first is the name of the loading parameter and another column is the definition of the parameters. Only the column with the definition of the parameter (Last column) should be modified according to the type of structural analysis problem. Each variable under this sub-section is defined as follows.

NLOADCASES: The number of load cases considered for the structural analysis.

LDTYPE: The type of loading is defined in this subsection. **DYN** for dynamic, **STA** for static and **CYC** for cyclic should be used while defining load type.

DSTYPE: Type of analysis whether it is force control or displacement control or acceleration control is defined in this sub-section. **FOR** for force control (nldcond=0, static or dynamic), **DIS** for displacement control (nldcond=1, static only), and **ACC** for acceleration control (nldcond=2 dynamic only) is used for defining.

NINC: Number of increments is defined in this sub-section. For Static, it is number of increments per loading cycle and for Dynamic, it is number of increments for application of own weight in static way.

NOTE: **NINC** variable represents the number of increments. In static loading conditions, it represents the total number of load or displacement increments. In dynamic loading conditions, this value should be selected carefully. It represents the number of increments in which the own weight is applied. If its value is positive, this means that static analysis is performed first for the own weight (without inertia or damping forces). In some cases, this value can not be positive as the matrix can not be solved statically, like in case of falling rock on a structure. If this value is zero, this means that the all the own weight is applied in the first increment in a dynamic way. If this value is negative, this indicates that no own weight is considered. Only, mass is taken into account.

NUNLOT: This defines the unloading condition during analysis. It is generally used for cyclic case. **"1"** for loading without unloading (in monotonic loading) and **"2"** for loading then unloading (in cyclic loading case) is used to define **NUNLOT.**

SELFWGT: This is used for temporary selfweight calculation and it is not used. (Generally 0)

DISPMAX: The maximum displacement (in meter) value after which collision check starts. If zero, no collision is allowed.

DRAWSECT: Number of increments after which the deformed shape is plotted to the file DRAW.

After the LOADDEF section load data are defined with following syntax :

NDOF: Element number to be monitored at displayed

LOAD DATA				
For Static displacement control data (LDTYPE: STA and DSTYPE: DIS)				
nrow				
i1	i2	inc	xxx	disp
i1	i2	inc	xxx	disp

…	….	…	…	…
For Dynamic ground acceleration data (LDTYPE: DYN and DSTYPE: ACC)				
dtorg	ndtintv	ndtintvcol		
accx	accy			
accx	accy			
….	….			

After the completion of LOADDEF section. The input for loading according to the type of loading is prepared.

For Static displacement control data (LDTYPE:STA and DSTYPE:DIS) following variables are defined in this section.

nrow: nrow represents the no. of rows for loading data. It is defined in the first row of this section. After the first row the loading data are prepared with each row having one loading data.

i1: First degree of freedom in which the loading is applied.

i2: Second degree of freedom until which the loading is applied.

inc: The number of increments of the degree of freedom in which the loading is applied after the previous loading point.

xxx: The maximum permitted displacement in case of load control. If not zero, unloading starts when the maximum displacement reaches this value irrespective to the applied load value. If zero, unloading starts after reaching the load reaches Vload irrespective to the displacement value. In displacement control it should be zero.

disp: The maximum load (vload) or displacement value(in meter). In dynamic load control, this value is meaning less and the load is read in each increment.

For Dynamic ground acceleration data (LDTYPE: DYN and DSTYPE: ACC) following variables are defined in this section.

dtorg: dtorg is the sampling rate of ground motion, △t (in sec.)

ndtintv: ndtintv is the minimum segments of △t (Geometrical residuals are very small)

ndtintvcol: Maximum segments of △t (Geometrical residuals large)

Note: The value of **NDTINTVCOL** should be greater than **NDTINTV.** If **NDTINTVCOL** is negative, this indicate that the value of time increment after collision check starts is constant and the original time increment is divided into equal **" NDTINTVCOL** "intervals. If it is positive, this indicates that variable time increment is used. Its alue ranges from **DTORG/NDTINTV** if the geometrical residual is very small compared to the maximum value obtained till that increment and **DTORG/NDTINTVCOL** if the geometrical residual value approaches the maximum value obtained till that increment.

accx: The ground acceleration in x-direction.

accy: The ground acceleration in y-direction.