Esap User Manual

General Program Description

Esap is a matrix structural analysis program for elastic pin-connected truss, 2D, and 3D rigid frame structures. The program uses a stiffness formulation to solve for the global displacements of the structure given the applied loads. The global and member end forces are back calculated from the displacements. The support reactions are assembled from the global member end forces.

The intent is for the program to be fast, easy to use, and accessible. Build a structure, apply some loads, get some deflections, forces, and reactions all natively in Excel.

User Notes

* While there are currently references in this document to truss and 3D rigid frame analysis capabilities they have not yet been implemented
* There is no error handling to assist the user at this time. If there is a mistake in the input, the errors are going to be the raw VBA errors
* Anything with a line through it in this document has not been implemented but is a place holder for future expansion of the analysis capabilities
* Do not change the location of the cells in the worksheets, otherwise the program will not read the input correctly
* Do not change the names of the worksheets (the tabs at the bottom), otherwise the program will not read the input correctly
* You can add additional sheets to the workbook. Do not use the names of the worksheets already in the workbook
* All input must be entered in consecutive rows or columns, i.e. don’t leave a blank row between joint definition inputs or a blank column between load ids for the load case input

Program Control and Execution

* Select a structure type
  + ~~Truss – Pin connected bar elements~~
  + 2D Frame – Rigidly connected 2D beam elements including axial stiffness
  + ~~3D Frame – Rigidly connected 3D beam elements including axial stiffness~~
* Run Analysis
  + After entering the geometry, supports, section properties, material properties, assigning the section and material properties to the geometry, entering the joint and element loads, and specifying the load cases to be run press this button to run the analysis.
  + Pressing this button automatically clears the existing results, if any.

Coordinate System

* Local Coordinate System
  + Truss & 2D Frame
    - X – defined positive from beginning to end along the axis of the member
    - Y and Z
      * Case 1 – Member not parallel to global Y-axis
        + The local Y-axis is determined by a plane parallel to the global Y-axis and containing the member
        + The local Y-axis for the member lies in this plane perpendicular to the member. The positive direction is defined to be the one in the same direction as the positive direction of the global Y-axis. Then use the right hand rule to define the local Z-axis.
      * Case 2 – Member parallel to global Y-axis
        + In this case the local Y-axis cannot be defined as above because any line perpendicular to the member is also perpendicular to the global Y-axis. There is no unique plane parallel to the global Y-axis and containing the member
        + To uniquely define the local Y-axis the local Z-axis is defined first. In this case it is defined to be in the same direction as the global Z-axis. The local Y-axis is then defined using the right hand rule.
* Global Coordinate System
  + X – positive to the right
  + Y – positive up
  + Z – out of the page
  + Follows right hand rule convention

Units

* The units are user determined although consistent units are required.
  + For example, if the joint coordinates are entered in feet the section properties and material properties and any load input need to be entered in terms of feet; the output will also be in terms of feet
* It is recommended to pick a set of units and be consistent with the input.
* The recommended set of units is inches and kips. However, feet and kips or kilonewtons and meters could be used.
* The output will be in the same units as the input

Joint Definition

* Joint ID
  + Unique joint identifier
  + Two joints cannot have the same name
  + Can be a string, number, special character, or a combination
* X-coordinate
  + Global x-coordinate of joint
  + Decimal number
* Y-coordinate
  + Global y-coordinate of joint
  + Decimal number
* Z-coordinate
  + Global z-coordinate of joint
  + Decimal number
  + Only used for 3D frame

Element Definition

* Element ID
  + Unique element identifier
  + Can be a string, number, special character, or a combination
* Beg Joint ID
  + Joint at the beginning of the member
  + Must be defined in the Joint Definition
* End Joint ID
  + Joint at the end of the member
  + Must be defined in the Joint Definition

Support Definition

* All supports are given in the global coordinate system
* Enter a value of 1 to fix the support in the given degree of freedom
* Enter a value of 0 to free the support in the given degree of freedom
* Values for all degrees of freedom must be entered for a given support
* Joint ID
  + Joint ID at location of support
  + Must be defined in the Joint Definition
* DX
  + Displacement in the x-direction
* DY
  + Displacement in the y-direction
* DZ
  + Displacement in the z-direction
  + 3D frames only
* RX
  + Rotation about the x-axis
  + 3D frames only
* RY
  + Rotation about the y-axis
  + 3D frames only
* RZ
  + Rotation about the z-axis
  + 2D and 3D frames only

Section Property Definition

* All section properties given in the element local coordinate system
* Section Property ID
  + Unique section property identifier
  + Can be a string, number, special character, or a combination
* A
  + Area of section
  + Used to compute the axial stiffness
  + Decimal number
* Avz
  + Not currently used by program, shear deformations are not currently considered
  + ~~Shear area in the local z-direction~~
  + ~~Used to compute the shear deformation component of the bending stiffness~~
  + ~~Only used for 3D frames~~
  + ~~Decimal number~~
* Avy
  + Not currently used by program, shear deformations are not currently considered
  + ~~Shear area in the local y-direction~~
  + ~~Used to compute the shear deformation component of the bending stiffness~~
  + ~~Only used for 2D and 3D frames~~
  + ~~Decimal number~~
* Izz
  + Moment of area about the local z-axis
  + Used to compute the bending stiffness
  + Only used for 2D and 3D frames
  + Decimal number
* Iyy
  + Moment of area about the local y-axis
  + Used to compute the bending stiffness
  + Only used for 3D frames
  + Decimal number
* J
  + Torsional moment of area about the local z-axis
  + Used to compute the torsional stiffness
  + Only used for 3D frames
  + Decimal number

Material Property Definition

* Material Property ID
  + Unique material property identifier
  + Can be a string, number, special character, or a combination
* E
  + Elastic modulus of material
* G
  + Shear modulus of material
* v
  + Poisson’s ratio of material

Section Property Assignment

* Element ID
  + Element ID of element to assign a section property
  + Must be defined in the Element Definition
* Section Property ID
  + Section property ID of assigned section property
  + Must be defined in the Section Property Definition

Material Property Assignment

* Element ID
  + Element ID of element to assign a material property
  + Must be defined in the Element Definition
* Material Property ID
  + Material property ID of assigned material property
  + Must be defined in the Material Property Definition

Joint Load Definition

* Joint loads applied in the global coordinate system
* Load ID
  + Unique load identifier
  + Joint and element load IDs cannot be duplicates
  + Can be a string, number, special character or a combination
* Description
  + User description of the load
  + This is not used by the program
  + Can be a string, number, special character or a combination
* Joint ID
  + Unique joint ID
  + ID of the joint to which the load is applied
  + Must be defined in the Joint Definition and used in the Member Definition
* Magnitude
  + Magnitude of the load to be applied
  + +/- Value of Load
  + +/- indicate direction of load in the applied axis
* Direction
  + Global coordinate axis on which the load acts
  + The following are valid input for the different structure types
    - Truss
      * FX, FY
    - 2DFrame
      * FX, FY, MZ
    - 3DFrame
      * FX, FY, FZ, MX, MY, MZ

Element Load Definition

* Load ID
  + Unique load identifier
  + Joint and element load IDs cannot be duplicates
  + Can be a string, number, special character or a combination
* Description
  + User description of the load
  + Not used by program
  + Can be a string, number, special character or a combination
* Element ID
  + Unique element ID
  + ID of the element to which the load is applied
  + Must be defined in the Element Definition and have section and material properties defined
* Load Type
  + Input Options
    - CONC
      * Concentrated point load
    - UNIF
      * Uniformly distributed load
    - ~~LINR~~
      * ~~Linearly distributed load~~
* Coordinate System
  + Coordinate system used to apply the load
  + Input Options
    - LOCAL
      * Member local coordinate system (See Coordinate System Definition)
    - GLOBAL
      * Structural global coordinate system (See Coordinate System Definition)
* Magnitude Beg
  + Must be same sign as Magnitude End
  + CONC Load Type
    - Magnitude of concentrated load
  + UNIF Load Type
    - Magnitude of uniformly distributed load
  + ~~LINR Load Type~~
    - ~~Magnitude at beginning of linearly distributed load~~
* ~~Magnitude End~~
  + ~~Must be same sign as Magnitude Beg~~
  + ~~CONC Load Type~~
    - ~~Not used, can be left blank~~
  + ~~UNIF Load Type~~
    - ~~Not used, can be left blank~~
  + ~~LINR Load Type~~
    - ~~Magnitude at end of linearly distributed load~~
* Direction
  + Defines the coordinate axis on which the load acts
    - Forces will act parallel to the specified axis
    - Force direction will be based on the sign given with the magnitude
    - Moment will act about the specified axis
    - Moment direction will be based on the sign given with the magnitude
  + The following are valid input for the different structure types
    - Truss
      * FX, FY
    - 2DFrame
      * FX, FY, MZ
    - 3DFrame
      * FX, FY, FZ, MX, MY, MZ
* Length Type
  + Location of load can be input as fractional length along the element or as absolute length along the element
  + Input Options
    - FRACT
      * Use fractional length input
    - ABS
      * Use absolute length input
* Begin Location
  + CONC Load Type
    - FRACT
      * Fractional distance from beginning of element to concentrated load application point
    - ABS
      * Absolute distance from beginning of element to concentrated load application point
  + UNIF Load Type
    - FRACT
      * Fractional distance from beginning of element to beginning of distributed load
    - ABS
      * Absolute distance from beginning of element to beginning of distributed load
  + ~~LINR Load Type~~
    - ~~FRACT~~
    - ~~ABS~~
* End Location
  + CONC Load Type
    - Not used, should be left blank
  + UNIF Load Type
    - FRACT
      * Fractional distance from beginning of element to end of distributed load
    - ABS
      * Absolute distance from beginning of element to end of distributed load
  + ~~LINR Load Type~~
    - ~~FRACT~~
    - ~~ABS~~

Load Case Definition

* Load Case ID
  + Unique load case identifier
  + Can be a string, number, special character or a combination
* Load ID
  + The list of Load IDs to be applied to structure at the same time and analyzed
  + The Load IDs must be entered in successive columns across the page in the same row as the Load Case ID
  + Must be defined in the Joint Load Definition or the Element Load Definition

Joint Displacement Output

* Joint displacements are given in the global coordinate system
* LC
  + Load Case ID
* JT
  + Joint ID
* DX
  + Deflection in the x-direction
* DY
  + Deflection in the y-direction
* DZ
  + Deflection in the z-direction
* RX
  + Rotation about the x-axis
* RY
  + Rotation about the y-axis
* RZ
  + Rotation about the z-axis

Element Force Output

* Element forces are given in the element local coordinate system
* LC
  + Load Case ID
* MEM
  + Element ID
* JT
  + Joint ID
* FX
  + Force in the x-direction
* FY
  + Force in the y-direction
* FZ
  + Force in the z-direction
* MX
  + Moment about the x-axis
* MY
  + Moment about the y-axis
* MZ
  + Moment about the z-axis

Structure Reaction Output

* Reactions are given in the global coordinate system
* LC
  + Load Case ID
* JT
  + Joint ID of support
* FX
  + Reaction in the x-direction
* FY
  + Reaction in the y-direction
* FZ
  + Reaction in the z-direction
* MX
  + Reaction about the x-axis
* MY
  + Reaction about the y-axis
* MZ
  + Reaction about the z-axis

Solution Routine

May talk about LU decomposition routine, and forward and backward substitution. The advantage of the LU decomposition over a full inversion. How the load vector for many load cases can be evaluated more computationally efficiently with the LU decomposition.