SSTAN Capabilities

Objectives

1) Use the SSTAN analysis program

The general analysis program SSTAN is available for download from www.ce.ufl.edu/software.htm . It is a 3-D finite element program that has a graphics post processor. It is a batch program that assumes the structural information is contained in an input file named “name.inp”. The results are put into a file named “name.out”. The users manual is available in the help file (from STANPLOT). The input is prepared and stored in the input file. The general rules for the input file are:

The ***first line*** in the file is an analysis title. IT MUST BE THE FIRST LINE. It is also important that it does not begin with any of the header names like COORDINATE, FRAME etc.

The ***second line*** is the analysis control information line. It is of the form:

N1,N2,N3,N4

Where N1 is the number of nodes in the structure,
N2 is the number of different element types in the structure. Again, this means if you are using both truss and beams there are two, regardless of the number of member properties.
N3 is the number of load cases.
N4 is the number of load combinations.

The rest of the INPUT is on a free formatted - header basis. All data is arranged by groups and is signified by a header. For example, nodal coordinates are signified by the header COORDINATE. This data is order independent. That, is these data blocks can be placed in the INPUT file in any order. All data groups must end with a blank line.

The following headers are available:

<u>COORDINATE</u>

This specifies the nodal coordinates of the structure to be analyzed. Note that the program assumes the structure is 3-D.

<u>BOUNDARY</u>

This specifies the boundary conditions or nodal DOF. Nodes can be either released or fixed and only in the global X-Y-Z coordinate system.

TRUSS

This specifies the truss element data. Trusses can have initial tension, rigid end offsets and the ability to not take either compression or tension (not both). The zero compression option is useful for slender bracing members. The zero tension option is useful for gap elements or uplift problems.

BEAM

This specifies the bending member data. These elements can be used for beams as well as columns. Beams can have uniform loads applied to the member, rigid end offsets and can include P-Δ effects.

LOADS

This specifies the concentrated loads applied to the structure. They can be concentrated loads or moments.

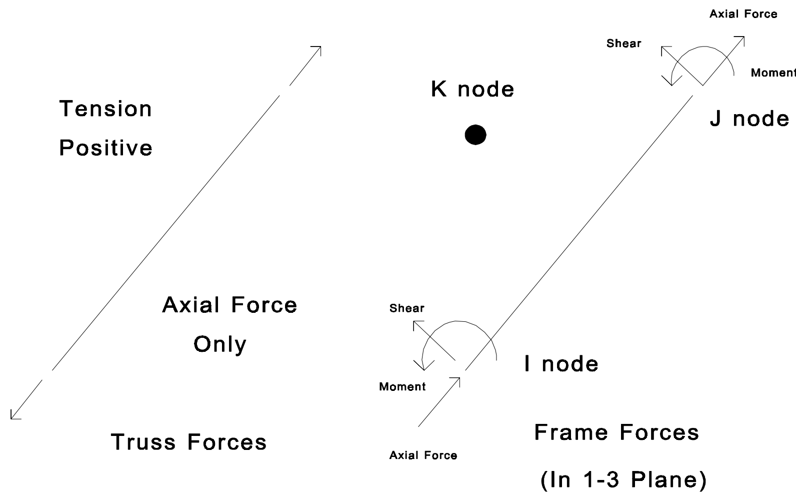
COMBINATIONS

This section allows linear combinations of the basic load cases. This option is not valid when using non-compression/tension trusses or P-Δ effects.

PLATE, MEMBRANE, SHELL, BRICK and AXISYMMETRIC

These are other types of members available. These constitute what are usually called finite elements and will not be discussed here but covered in later chapters. Several examples will be given to demonstrate the use of SSTAN.

The coordinate system assumed is SSTAN is a right hand rule system. All displacements, forces and moments are given in this right-handed system. The results for element forces are given at the nodes of a member in the local coordinate system. The local coordinate system for the truss and beam members are given below.



Force Output for Truss and Beam

Use of STANPLOT

STANPLOT is a graphics post processor for the SSTAN program. STANPLOT can plot any of the SSTAN element types, displaced shapes, axial and moment results for beams and truss members, and stress contours for membranes, plates, shells and solid elements. The program allows windowing and changing of the structures view as well as many other useful options.

STANPLOT has many additional capabilities. It is an essential tool in checking the structure to see that its input and analysis are correct. Using the plotting program things like connectivity, nodal coordinates, symmetry in displacements and many other things can be checked. Verifying the structure and results is the most important part of a structural analysis.

SSTAN Solution Errors

If a structure is unstable, the stiffness matrix generated is singular. As a result, during the equation solving process an error will be generated specifying a singular matrix or that there is a **Negative** on the diagonal. Some typical causes of a singular matrix are:

1) The structure is unstable. The cause of this is generally improper specification of boundary conditions. As in the above example, all out of plane DOF were fixed, if this were not the case, an error would have occurred giving a singular matrix.

Another possible cause is if the structure were to be placed on rollers. At least ONE horizontal fixity must be supplied for stability. In the direct stiffness method, all active DOF must have some stiffness associated with them. Even if no load is applied to the horizontal direction, the structure must be stabilized.

2) The elements are not connected correctly, causing an unstable structure. Usually, the real structure is stable but the model is unstable due to a typographical error in element connectivity. Check to be sure the elements are connected to the proper nodes. STANPLOT can be very helpful for this type of error.

3) The element properties are specified incorrectly. If an element property is left out or read in as zero, this can cause the structure to be unstable. This is common in 3-d structures where the torsional property is left out.

One final point to note is that when distributed loads are used, the maximum moment in the span is given along with the beam member forces. This is useful for design where the maximum moment is required.

A good way to check the structure is through a graphical display of the structure and its results. SSTAN has a graphical post processor that allows display of the structure, displaced shapes and stress and displacement contours for finite elements. To use the graphics post processor, just run the program (STANPLOT) in the directory where the result data is contained.