Author:	Antonino Vicente Rico	Date:	10/07/2013

FEM	X	HAND		LIN		NOLIN		BUCK		FAT		STATIC		COMP	X	MET	X

HOW TO OBTAIN AND USE STIFFNESS MATRIX

1. INTRODUCTION

There are situations in which some subparts of a structure cannot be implemented in the NASTRAN model (or it is not desired). Some of these situations are the following:

- Analyze different components within a structure.
- To incorporate in a structure a component whose stiffness matrix has been obtained with laboratory test or with highly detailed finite element model.
- Analyze a subpart of a huge structure. For example a rib of an aircraft wing.

To be able to take into account the influence of the part which is not modeled it is necessary to obtain its stiffness matrix. If there is any load or constraint applied on the structure to be condensed, it would be necessary to be taken into account when the condensation is performed.

Depending on the case that is being studied there are two ways to model the structure. It is possible to use GENEL elements or superelements. The use and suitability of each one is explained in the following chapters.

In the same way, the modeling of the structure is going to be explained. The way in which the stiffness matrix can be obtained and the loads and constraints can be considered are discussed.

2. OBTAINING THE STIFFNESS MATRIX

There are two main ways to determine the stiffness matrix of a structure, which are going to be explained below:

Unitary displacements method.

The unitary displacements method barely has practical application because obtaining and post-processing the data is a laborious task. The process consist on introducing unitary displacements in the degrees of freedom in which it is desired to obtain the stiffness matrix. At the same time the unitary displacements are introduced, it is necessary to constraint the remaining desired DOFs. With this process the force vector obtained from

the resolution of the problem has the value of the corresponding stiffness column. To do the process more straightforward an example is going to be carried out below:
For a general structure equation (2.0) shows its behaviour.

$$\begin{bmatrix} K_{11} & K_{12} & \dots & K_{1n} & \dots & K_{1m} \\ K_{21} & K_{22} & \dots & K_{2n} & \dots & K_{2m} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ K_{n-1,1} & K_{n-1,2} & & K_{n-1,n} & & K_{n-1,m} \\ K_{n1} & K_{n2} & \dots & K_{nn} & \dots & K_{nm} \\ K_{n+1,1} & K_{n+1,2} & & K_{n+1,n} & & K_{n+1,m} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ K_{m1} & K_{m2} & \dots & K_{mn} & \dots & K_{mm} \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \\ \vdots \\ \delta_{n-1} \\ \delta_n \\ \delta_{n+1} \\ \vdots \\ \delta_m \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ \vdots \\ F_{n-1} \\ F_n \\ F_{n+1} \\ \vdots \\ F_m \end{Bmatrix} \quad (2.0)$$

If it is desired to obtain the stiffness matrix components for the n first DOFs of the structure, unitary displacements have to be impose in each degree of freedom. And when the problem is solved, the force vector will represent the corresponding stiffness matrix column. Part of the process is displayed in the below sequence of equations ((2.1) (2.2) … (2.n)).

It is important to notice that for each unitary displacement it is necessary to solve the system of equations. On these systems the unknowns are the displacements $\delta_{n+1} \dots \delta_m$ and the loads $\overline{K}_{1i} \dots \overline{K}_{ni}$ with $i = 1 \dots n$ as a function of the DOF in which the displacement is applied.

$$\begin{bmatrix} K_{11} & K_{12} & \dots & K_{1n} & \dots & K_{1m} \\ K_{21} & K_{22} & \dots & K_{2n} & \dots & K_{2m} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ K_{n-1,1} & K_{n-1,2} & & K_{n-1,n} & & K_{n-1,m} \\ K_{n1} & K_{n2} & \dots & K_{nn} & \dots & K_{nm} \\ K_{n+1,1} & K_{n+1,2} & & K_{n+1,n} & & K_{n+1,m} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ K_{m1} & K_{m2} & \dots & K_{mn} & \dots & K_{mm} \end{bmatrix} \begin{Bmatrix} 1 \\ 0 \\ \vdots \\ 0 \\ 0 \\ \delta_{n+1} \\ \vdots \\ \delta_m \end{Bmatrix} = \begin{Bmatrix} \overline{K}_{11} \\ \overline{K}_{21} \\ \vdots \\ \overline{K}_{n-1,1} \\ \overline{K}_{n1} \\ F_{n+1} \\ \vdots \\ F_m \end{Bmatrix} \quad (2.1)$$

$$\begin{bmatrix} K_{11} & K_{12} & \dots & K_{1n} & \dots & K_{1m} \\ K_{21} & K_{22} & \dots & K_{2n} & \dots & K_{2m} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ K_{n-1,1} & K_{n-1,2} & & K_{n-1,n} & & K_{n-1,m} \\ K_{n1} & K_{n2} & \dots & K_{nn} & \dots & K_{nm} \\ K_{n+1,1} & K_{n+1,2} & & K_{n+1,n} & & K_{n+1,m} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ K_{m1} & K_{m2} & \dots & K_{mn} & \dots & K_{mm} \end{bmatrix} \begin{Bmatrix} 0 \\ 1 \\ \vdots \\ 0 \\ 0 \\ \delta_{n+1} \\ \vdots \\ \delta_m \end{Bmatrix} = \begin{Bmatrix} \overline{K}_{12} \\ \overline{K}_{22} \\ \vdots \\ \overline{K}_{n-1,2} \\ \overline{K}_{n2} \\ F_{n+1} \\ \vdots \\ F_m \end{Bmatrix} \qquad (2.2)$$

$$\vdots$$

$$\begin{bmatrix} K_{11} & K_{12} & \dots & K_{1n} & \dots & K_{1m} \\ K_{21} & K_{22} & \dots & K_{2n} & \dots & K_{2m} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ K_{n-1,1} & K_{n-1,2} & & K_{n-1,n} & & K_{n-1,m} \\ K_{n1} & K_{n2} & \dots & K_{nn} & \dots & K_{nm} \\ K_{n+1,1} & K_{n+1,2} & & K_{n+1,n} & & K_{n+1,m} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ K_{m1} & K_{m2} & \dots & K_{mn} & \dots & K_{mm} \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ \vdots \\ 0 \\ 1 \\ \delta_{n+1} \\ \vdots \\ \delta_m \end{Bmatrix} = \begin{Bmatrix} \overline{K}_{1n} \\ \overline{K}_{2n} \\ \vdots \\ \overline{K}_{n-1,n} \\ \overline{K}_{nn} \\ F_{n+1} \\ \vdots \\ F_m \end{Bmatrix} \qquad (2.n)$$

It is important to notice the difference between K and $\overline{K}$. The first one is the stiffness of the structure non-condensed (complete stiffness matrix) and the second one is the stiffness of the structure once it is condensed (condensed stiffness matrix). The stiffness in a DOF of a non-condensed structure is only a function of the elements which are joined with the node corresponding with the mentioned DOF. On the other hand when the structure is condensed, the stiffness corresponding to a DOF take into account the stiffness of the whole structure.

To better understand the difference explained above, imagine the structure shown on Figure 1.

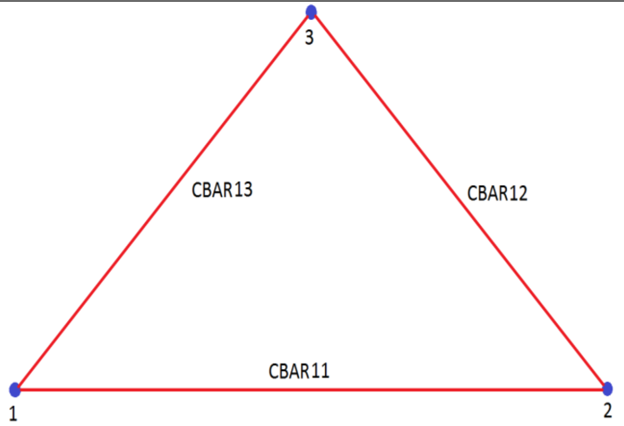

Figure 1

If the complete stiffness matrix of the structure would be displayed, it could be possible to see that the DOFs corresponding to the node 1 would receive the stiffness of Bars 11 and 13. For the node 2, bars 11 and 12. And for node 3, bars 12 and 13. To obtain the complete stiffness matrix, the process explained before should be carried out. For example, to obtain the stiffness corresponding to the horizontal displacement of node 1, the forces of the structure on the condition shown on Figure 2 should be obtained. On this figure, node 1 has a unitary displacement on horizontal direction while all the remaining DOFs of the structure (nodes 1, 2 and 3) are constrained to zero.

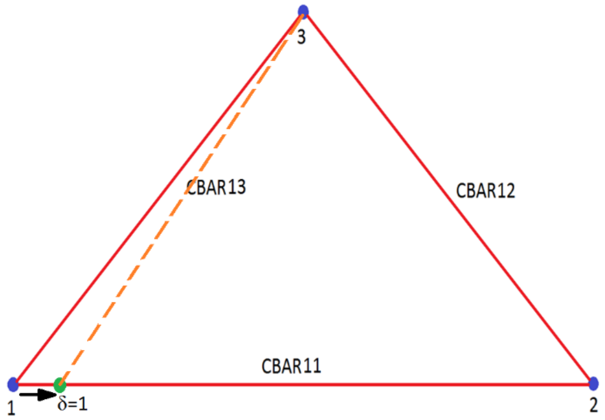

Figure 2

On the other hand, to obtain the condensed stiffness matrix it is necessary to impose unitary displacements on the desired DOFs (not all of them) while the remaining desired

DOF (not all of them) are constrained to zero. But when the condensed stiffness matrix of the structure is obtained, the displacements on non-desired DOFs are free (non-constrained), which makes the differences with the complete stiffness matrix. In the case that has been studied the condensed stiffness matrix is desired on nodes 1 and 2, DOFs associated with node 3 would be free. Therefore when, for example, a unitary horizontal displacement is introduced in node 1, the configuration of the structure would be different regarding the one shown on Figure 2. In this case node 3 would have the displacement obtained from the behavior of the structure, being possible to obtain a behavior as the one shown on Figure 3.

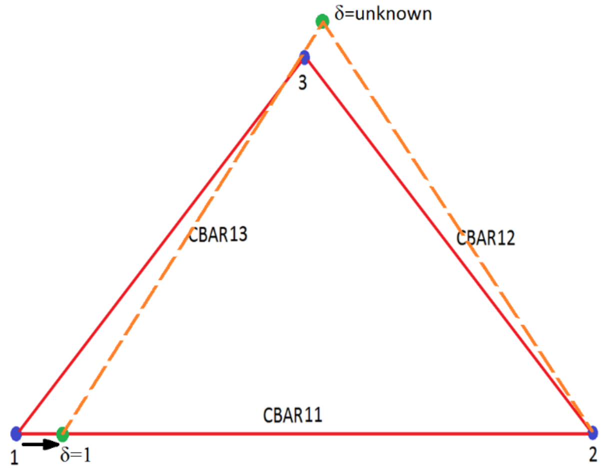

Figure 3

Therefore, as can be noted the configuration of the structure changes, and as a consequence, loads and the stiffness associated with nodes 1 and 2 change as well.

After this paragraph the NASTRAN code corresponding to the method previously commented will be displayed. It is necessary to difference the case in which the complete stiffness matrix of the whole structure is required from the case in which the condensed stiffness matrix is required.

When the complete matrix is required all DOFs would be constrained. Therefore, there is not any DOF to be obtained and a NASTRAN run would report a FATAL error. To avoid this FATAL error an additional structure with free DOFs has to be added. It is important that the new structure would not be related with the original one. A simple example for this purpose is the use of a cantilever beam with two nodes and one element.

When a condensed matrix is required, there is no problem as the previously mentioned one and the run can be carried out normally.

To obtain the forces, and therefore the stiffness matrix components, SPCFORCES should be requested in the NASTRAN run.

Once the NASTRAN run has been performed, the obtained data need to be sorted in order to obtain the stiffness matrix.

```
SOL 101
CEND
…
SUBCASE 1
  SPC = 1
  LOAD = 1
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 2
  SPC = 1
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
…
BEGIN BULK
$1111111222222223333333344444444555555556666666677777777
…
SPCD    1    1    1    1.
SPCD    2    1    2    1.
…
SPC1    1    123456  1
…
```

ASET method.

This method, which has a wide implementation on the industry, use ASET cards and the PARAM EXTOUT. To select the desired degrees of freedom in which the stiffness matrix components are desired, it is only necessary to introduce ASET cards in the bulk data.

With this card also the load vector, damping matrix or mass matrix are obtained, but for the moment only the stiffness matrix will be considered.

It is important to notice that with this process the structure is condensed into the DOFs placed in the ASET cards (the desired DOF). Also it is necessary to remark that the process to obtain the condensed stiffness matrix is carried out once the constrained DOFs are removed from the complete stiffness matrix of the whole structure. Therefore, if it is desired to obtain the matrix of the structure without any constraint, a special NASTRAN run has to be performed (removing all the SPC). As the structure is not constrained there are 6 possible rigid movements. Hence, to avoid the FATAL error due to maxratio exceed, PARAM,BAILOUT,-1 has to be written into the bulk data of the code.

The NASTRAN code for the ASET method is displayed below:

```
SOL 101
CEND
…
BEGIN BULK
$1111111222222223333333344444444555555556666666677777777
…
PARAM   EXTOUT  DMIGPCH
$ FOLLOWING PARAM NECCESARY IF THE STRUCTURE IS NOT CONTRAINET
PARAM,BAILOUT,-1
…
ASET1   123456  1       2
…
```

In the previous code the degrees of freedom have been selected with ASET1 card. A punch archive has been selected with the PARAM, EXTOUT. If it is desired to modify the output of the stiffness matrix the PARAM EXTOUT information can be read in the Quick Reference Guide of NASTRAN (for example to obtain an op2 file instead of a pch file).

3. INSERTION OF THE STIFFNESS MATRIX INTO THE STRUCTURE

Once the stiffness matrix has been obtained, the process of inserting the matrix into the model is going to be explained. As it has been commented on chapter 1, there are two main ways to consider the influence of the stiffness. One, is the use of GENEL elements and the other is the use of superelements. In the following lines both ways are going to be explained as well as the most common examples.

GENEL element.

The GENEL element is a generic element in which it is possible to define the stiffness components on the DOFs desired. Although it is possible to introduce the flexibility matrix, the most common use of this element is with the stiffness matrix. To place this element into the structure, the GENEL card has to be written in the bulk data section. To fill the card, the desired degrees of freedom and the upper triangular part of the corresponding matrix are needed.

The NASTRAN code corresponding to the previous process is presented below. To better understand of how to fill the GENEL card the stiffness matrix inserted in the NASTRAN code is showed on equation (3.0)

```
SOL 101
CEND
…
BEGIN BULK
$111111112222222233333333444444445555555566666666677777777
…
GENEL   1           1       1       1       2       1       3
        1       4       1       5       1       6
        K       200000. 0.      0.      0.      0.      0.      1939.494
        0.      0.      0.      96974.721939.4940.      -96974.70.
        1081866.0.      0.      6515402.0.      6515402.
…
```

200000.	0.	0	0.	0.	0.	
0.	1939.494	0.	0.	0.	96974.72	
0.	0.	1939.494	0.	-96974.7	0.	(3.0)
0.	0.	0.	1081866.	0.	0.	
0.	0.	-96974.7	0.	6515402.	0.	
0.	96974.72	0.	0.	0.	6515402.	

The use of GENEL elements is useful when there are not many degrees of freedom in order not to write a huge GENEL card. Also it is useful when it is desired to introduce multiple similar elements on different places. To do this, only the fields corresponding to the DOFs should be changed.

It is important to notice that when there are loads applied on the structure which is represented by the GENEL element, the application nodes must be placed in the GENEL card. This task is more complicated than the one performed in the superelements.

Care must be taken with the values of truncation of the stiffness. Any difference between corresponding terms of the stiffness matrix can create a spurious constraint on the structure which will generate grounding (OLOAD <> SPCFORCES).

With the purpose of clarifying the previous concepts an example is going to be presented below.

Imagine a bracket as the one showed on Figure 4. This element joins an actuator with the principal structure which is desired to be analyzed. Loads are transferred to the structure through the three fasteners shown on Figure 4. For some reason the bracket is a critical element of the structure, and therefore it is desired to perform a detailed analysis of it.
After the detailed analysis, the influence of the bracket can be introduced into the structure using a GENEL element. In this case the GENEL card would have 24 DOFs corresponding to the three fastener nodes and one for the actuator. Please note that the load coming from the actuator has been modeled as a load acting on a point. It is important to remember that the nodes in which the stiffness factor were obtained, must be included into the new model. Once the GENEL, the four nodes, and the loads are into the NASTRAN code, the analysis can be performed.

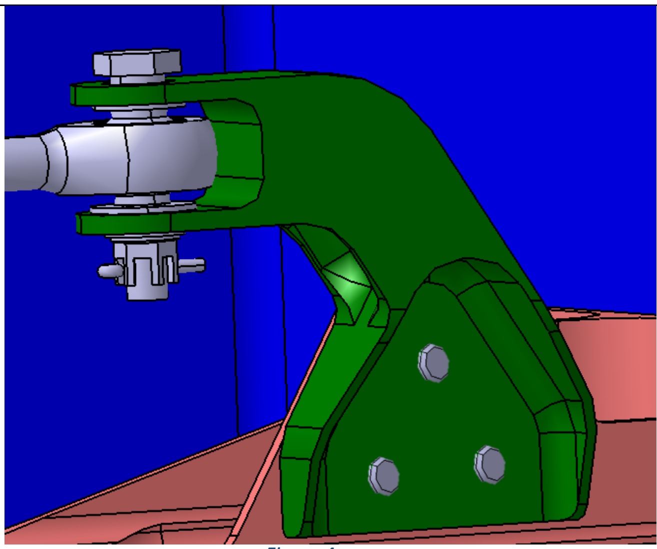

Figure 4

Superelements.

With superelements, as with GENEL elements, it is possible to insert into a structure stiffness in any DOF. But one of the most important difference in comparison with GENEL elements is that with superelements the incorporation of loads is easier. The first task to perform is the inclusion of the stiffness matrix into the NASTRAN code. The usual way in which the stiffness matrix has been obtained is with ASET. Therefore it is only needed to copy the matrix KAAX from the punch file and paste it into the NASTRAN code. If the stiffness matrix has not been obtained with ASET, the matrix should be assembled and introduced with DMIG cards. It is important to highlight that only the non-null components of the upper triangular part of the matrix has to be filled in the DMIG cards.

If the structure which is condensed has any kind loads applied, there is not necessary to carry out a task as in the GENEL. That is, it is not necessary to add the nodes in which loads are applied. The only thing that has to be performed is to copy the PAX vector into the NASTRAN code. This vector is placed in the same file that the KAAX matrix.

Finally, to complete the superelements process, it is necessary to call the stiffness matrix and the vector of loads. To do that “K2GG = KAAX” and “P2G = PAX” sentences have to be introduced into the case control of the final code.

The NASTRAN code with the superelements implementation is presented below:

```
SOL 101
CEND
…
K2GG=KAAX
P2G=PAX
…
BEGIN BULK
$111111122222222333333334444444455555555666666677777777
…
DMIG    KAAX       0      6      2      0                24
DMIG*   KAAX                   1           1
*              1           1 2.000000000D+05…
…
DMIG    PAX        0      9      2      0                1
DMIG*   PAX                    1           0
*            111           1-1.8E+01
…
```

If superelements and GENEL elements are compared, the conclusion is that superelements are more versatile. Consequently, superelements is the most extended method in the industry. Some examples of superelements use can be the analysis of different components within a structure or the analysis of a subpart of a huge structure.

To clarify the superelements method, the analysis of the bracket example from Figure 4 is going to be explained using superelements methodology.

As in the GENEL study, a detailed studied has been done, being able to reduce the structure to the desired DOFs. Due to the fact that with superelements is possible to obtain the reduced loading vector, it is not necessary to consider the DOFs of the nodes in which loads are applied.

In case of multiple load cases analysis, a special management of the NASTRAN run should be done, for further information, please check Aersys Knowledge Unit 7024.