SUPERELEMENTS IN MULTICASE ANALYSES

1. INTRODUCTION

Superelements can be used on detailed FEM to improve the way the boundaries are loaded. They are also widely used on the global FEM to separate the different main sections of the airplanes.

The most usual way to generate the superelement is by generating the condensed stiffness matrix and the condensed load vector as DMIG cards by setting the EXTOUT parameter to DMIGPCH.

Some problems arise when the input file to generate the superelement and/or the input file using the superelement are multicase files.

2. PROBLEMS ON THE FILE GENERATING THE SUPERELEMENT

When the input file that generates the superelement is run a punch file is created (if PARAM EXTOUT is set to DMIGPCH).

The punch file generated this way contains the DMIG cards to be used on the analysis with the superelement.

When the MPC and SPC case control cards reference to the same values on all the subcases the condensed stiffness matrix does not change. But when the SPC or MPC change in any subcase then different condensed stiffness matrices will be generated.

If several subcases are condensed with the same stiffness matrix (same MPC and same SPC) then the condensed load vector is transformed in a matrix with a number of columns equal to the number of subcases. Each column of the condensed load matrix is applied to each subcase on the file that includes the superelement. This is made this way because the P2G must be defined above the subcase level.

When the condensed stiffness matrices of the different subcases are different (because of different SPC or MPC) then several matrices are issued grouping sets of consecutive subcases with the same stiffness matrix. The condensed load vector of this set of grouped subcases will be generated as well. If the set has more than one subcase the load vector will be transformed in a matrix with a number of columns equal to the number of subcases of the group.

The main problem with this punch file is that it cannot be directly used as an input on the

file using the superelement because all the stiffness matrices and load vectors have the same ID (KAAX and PAX). Therefore, the file has to be split in several different files including each one a set of subcases with a common condensed stiffness matrix.

An example is presented to clarify the problem:

$ INPUT FILE	$ PUNCH RESULTING FILE
SOL 101	
…	
SUBCASE 1	DMIG KAAX….
SPC=1	….
MPC=1	DMIG PAX 0 9 2 0 **2**
…	DMIG* PAX 1 0
SUBCASE 2	….
SPC=1	DMIG* PAX 2 0
MPC=1	….
…	DMIG KAAX….
SUBCASE 3	….
SPC=2	DMIG PAX 0 9 2 0 **1**
MPC=2	DMIG* PAX 1 0
…	….
SUBCASE 4	DMIG KAAX….
SPC=3	….
MPC=1	DMIG PAX 0 9 2 0 **3**
…	DMIG* PAX 1 0
SUBCASE 5	….
SPC=3	DMIG* PAX 2 0
MPC=1	….
…	DMIG* PAX 3 0
SUBCASE 6	….
SPC=3	DMIG KAAX….
MPC=1	….
…	DMIG PAX 0 9 2 0 **1**
SUBCASE 7	DMIG* PAX 1 0
SPC=1	….
MPC=1	
…	

3. PROBLEMS ON THE FILE USING THE SUPERELEMENT

The file including superelements can have several subcases defined. The matrix of condensed loads can be used to assign a load vector to each subcase. In fact, it is not possible to use different load vector on the subcase level. If several load cases has to be included then a condensed load matrix has to be called above subcase level.

Theoretically the stiffness matrix can be referenced on the subcase level. Therefore it seems to be possible to implement all the subcases condensed even if these different subcases create different stiffness matrices. The only tasks to be performed are: to separate the stiffness matrices, to rename the different stiffness matrices to be referenced by the subcases K2GG case control command, and to collect and order the condensed load matrix (according to the order of subcases on the file).

However, a problem has been detected if this kind of analysis is run. NASTRAN seems to use the same stiffness matrix (the first one) for all the subcases. This creates erroneous results for all the subcases that must use a different stiffness matrix.

So far, no solution has been found for this problem and the conclusion is that the analysis involving superelements on NASTRAN can only be run if the same condensed stiffness matrix has to be used or all of them.

An example is presented to clarify the problem:

```
SOL 101
…
$ PAX load matrix has three columns
P2G=PAX
SUBCASE 1
        K2GG=KAAX1
        …
SUBCASE 2
        K2GG=KAAX2
        …
SUBCASE 3
        K2GG=KAAX3
        …
```

All the subcases above will use KAAX1 on their calculations.