

## BUCKLING WITH SUPERELEMENTS

### 1. INTRODUCTION

In some kind of structures it is desired to analyze the buckling behavior of specific zones. It is important to notice that this analysis only makes sense when the rest of the structure does not have relevant influence in the local buckling.

The first buckling mode associated with the desired zone is not usually one of the first buckling modes of the global structure. Therefore, obtaining the desired buckling mode could be a laborious task. In order to reduce the necessary effort it is possible to isolate and analyze the desired zone and its first buckling mode.

To do this document more straightforward, an example is going to be used along the document. For example, it is desired to analyze the vertical tail plane (VTP) of an aircraft, as the one shown in Figure 1. To achieve the airworthiness certification, it is compulsory to prove the non-buckling for the leading edge panels under different load cases. Therefore, a buckling analysis has to be carried out to achieve the previous condition. If a buckling analysis is performed in the whole structure, probably buckling modes on the skin will appear before the leading edge buckling modes. Therefore, as it has mentioned previously, the desired zone (in this case the leading edge) is going to be isolated and analyzed.

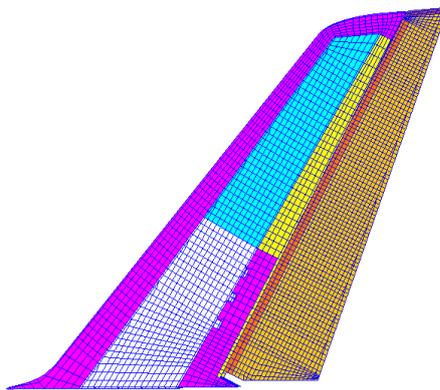


Figure 1

Although there are different ways to perform the analysis of the desired zone, the most accurate one is to perform an analysis with superelements. Despite the fact this is the most accurate solution, it is not as exact as the global buckling analysis. The error for the superelements analysis is a consequence of non-considering the differential stiffness of

		<b>AERSYS KNOWLEDGE UNIT</b>				AERSYS-7007											
Author:		Antonino Vicente Rico				Date:		19/06/2013									
FEM	X	HAND		LIN		NOLIN		BUCK	X	FAT		STATIC		COMP	X	MET	X
<b>BUCKLING WITH SUPERELEMENTS</b>																	

the condensed structure. This topic will be explained in the following chapters.

## 2. SUPERELEMENTS CREATION AND ANALYSIS

There are different ways to isolate a desired zone:

- **Enforced displacements:** To carry out this method, the whole structure is analyzed with a static linear solution (SOL 101). Displacements are obtained on the interface nodes of the desired zone. Once the displacements are obtained, it is possible to carry out the analysis of the desired zone. To perform the isolated analysis, the previous displacements are imposed as enforced displacements in the isolated structure.
- **Force load:** In this method instead of requesting displacements on the interface nodes, forces on these nodes are requested. These forces are imposed in the isolate structure to perform the buckling analysis. An isostatic boundary condition should be imposed in order to avoid mechanism.
- **Superelements:** On this method the influence of the whole structure in the desired zone is condensed in a stiffness matrix (KAAX) and a load vector (PAX) in the interface nodes. The stiffness matrix and the load vector represent the behavior of the structure without the desired zone in order to transfer it to the isolated structure. Once the vector and the matrix are obtained, the isolated analysis can be performed including the KAAX and the PAX in the interface nodes of the isolated analysis.

Among the three different ways explained above, the most accurate one is the use of superelements. The reason for this is that using enforced displacements or force load take into account the properties of the desired zone. This is because the displacements or forces are obtained on the interface nodes when the whole structure (including the desired zone) is analyzed. Therefore, if stiffness changes appear between the desired zone in the global structure and the analysis of the isolated zone (because the isolated zone is analyzed by means of a DFEM instead of the GFEM mesh) errors arise. Further explanation is provided on AERSYS Knowledge Unit 7015.

This kind of solution condenses the influence in the desired zone of the rest of the structure. The influence is given by a load matrix (PAX) and a stiffness matrix (KAAX). Matrix PAX represent the reduction of the applied loads on the boundary (interface). And matrix KAAX represent the stiffness of the rest of the structure on the boundary (interface).

 <a href="http://www.aersys.com">www.aersys.com</a>		<b>AERSYS KNOWLEDGE UNIT</b>				AERSYS-7007											
Author:		Antonino Vicente Rico				Date:		19/06/2013									
FEM	X	HAND		LIN		NOLIN		BUCK	X	FAT		STATIC		COMP	X	MET	X
<b>BUCKLING WITH SUPERELEMENTS</b>																	

For the analysis with superelements, two different task have to be performed:

- Obtain the superelement from the static case. That is, obtain the matrix KAAx and the vector PAX.
- Impose the stiffness matrix and the load vector in the analysis of the isolated zone.

The first step to obtain the influence of the structure is to select the boundary degrees of freedom. To do that, ASET1 cards must be introduced in the bulk data section. Once DOFs are selected, the output file must be specified with the PARAM EXTOUT. An example NASTRAN code is shown below. In this example punch file exit has been selected.

```

SOL 101
CEND
...
...
BEGIN BULK
$111111122222223333333344444444555555556666666677777777
...
PARAM,EXTOUT,DMIGPCH
...
ASET1 123456 111
ASET1 123456 112
...

```

It is important to notice that in order to obtain the superelement matrix and vector, the zone that has to be isolated must be removed from the FEM.

This first task can be extrapolated to the VTP example. In this case the VTP without the leading edge (zone to be removed) must be analyzed, as it is shown in Figure 2. Also interface DOFs between the leading edge and the whole structure must be placed in ASET1 cards. Interface nodes are plotted in Figure 2 in red.

## BUCKLING WITH SUPERELEMENTS

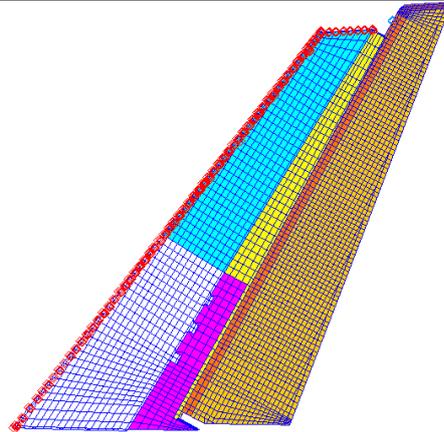


Figure 2

Once KAAX and PAX are obtained, the second task can be carried out. Now it is possible to analyze the buckling of the desired zone. To perform this analysis, a new NASTRAN code has to be created with only the desired zone of the structure. Matrix KAAX and vector PAX must be taken into account to include the influence of the rest of the structure. To introduce the matrix and the vector in the analysis, they must be called as stiffness matrix and force vector. This is performed introducing K2GG=KAAX and P2G=PAX into the case control, and including KAAX and PAX matrix into the bulk data section.

An example of NASTRAN code to analyze the buckling of the desired zone is shown below.

```
SOL 105
CEND
...
K2GG=KAAX
P2G=PAX
...
SUBCASE 1
...
SUBCASE 2
METHOD = 1
...
BEGIN BULK
$111111122222223333333344444444555555556666666677777777
```

## BUCKLING WITH SUPERELEMENTS

```

...
DMIG KAAX      0  6  2  0          2160
DMIG* KAAX           111          1
*           111      1 5.8E+01
...
DMIG PAX       0  9  2  0          1
DMIG* PAX           1          0
*           111      1-1.8E+01
...

```

The second task can be extrapolated to the VTP example. In this case, the analysis must be performed in the leading edge, as is shown on Figure 3. KAAX matrix and PAX vector must be imposed on the interface nodes (plotted in blue on Figure 3).

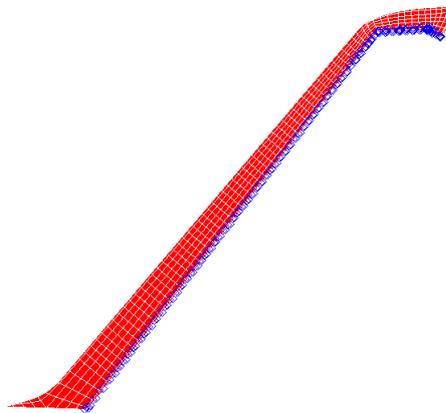


Figure 3

This method is more accurate not only because the static subcase is better loaded, it is also more accurate because the interface clamping degree for buckling is better matched than on the other two approaches. At the end, using the enforced displacements or the forces will imply the need of imposing a boundary condition for the buckling subcase. This boundary condition is typically a clamping or a simple support. The accuracy of the boundary condition for the buckling subcase can be improved by adding spring with a value of the stiffness calculated by analytical approaches, but it will never be very accurate. The superelements method estimates the stiffness of the interface by using the FEM stiffness and it provides the stiffness as a clamping degree on the buckling subcase.

		<b>AERSYS KNOWLEDGE UNIT</b>				AERSYS-7007											
Author:		Antonino Vicente Rico				Date:		19/06/2013									
FEM	X	HAND		LIN		NOLIN		BUCK	X	FAT		STATIC		COMP	X	MET	X
<b>BUCKLING WITH SUPERELEMENTS</b>																	

### 3. ERRORS ON BUCKLING

It is important to notice that when a local buckling is performed, errors appear in comparison with the global buckling. Errors exist in the three different ways of isolating a zone of a structure although they are different.

In case of enforced displacements and force load there are errors due to possible stiffness changes, as it has been mentioned in the beginning of the previous chapter. Typically the buckling is calculated with models more detailed than the ones used to get the KAAX and PAX.

But there is one common error for the three ways which is due to the condensation of the structure. When the structure is condensed the stiffness matrix of the rest of the structure is obtained, but the differential stiffness matrix is not computed.

The differential stiffness matrix is required for buckling analysis. This matrix represents the variation of the stiffness in a structure as the load increases. For example, imagine you have a ruler. If you apply on it a tension load, the transversal stiffness increases regarding null load. On the other hand, if you apply a compression load, the transversal stiffness decreases, being possible to reach the buckling (null transversal stiffness).

In the global structure, the stiffness modification occurs before the desired local buckling is achieved. These stiffness variations are not taken into account with an isolated study as it has been mentioned above, therefore errors appear. Notice that this error can be either conservative or not, depending of the stiffness variation regarding the initial stiffness.

There are two different ways of taking into account the previous error. The first one is to consider a minimum required RF. That is, instead of considering  $RF=1.0$  as threshold,  $RF=1.1$ , for example, would be the new threshold. Therefore obtaining  $RF=1.09$  would mean the failure of the structure.

The other way to take into account the error, is to perform a conservative analysis changing the boundary conditions as explained in the next chapter.



# AERSYS KNOWLEDGE UNIT

AERSYS-7007

Author: Antonino Vicente Rico

Date: 19/06/2013

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

## BUCKLING WITH SUPERELEMENTS

### 4. CHANGES ON BOUNDARY CONDITIONS

When a buckling analysis is performed with superelements, the boundary conditions for the buckling subcase are imposed by constraints existing in the studied zone and in the rest of the structure.

The rest of the structure applies constraints to the desired zone thanks to the stiffness matrix. It is important to remark that the stiffness matrix KAAX only provides stiffness to the interface DOFs corresponding to the a-set nodes. These DOFs have been selected in the NASTRAN code corresponding to the superelement creation.

If it is desired to change the boundary conditions in order, for example to obtain more conservative results, the following process can be performed. Coincident nodes should be placed in the interface nodes. MPC should join the coincident nodes rotational DOF whilst the translational DOF will be transferred by RBE2. These MPC would connect the rotational DOFs connecting or disconnecting them depending on the subcase selected.

Usually, to obtain conservative results, a simple supported condition is applied on the interface nodes (translational DOF are fully constrained). Therefore, rotational stiffness would be null. In the static subcase rotational DOF can be transferred with the corresponding MPC, and this way the structure is correctly loaded. But in the buckling subcase the MPC corresponding to the rotational DOFs is not called in order to obtain simple supported conditions (or the translational stiffness that the KAAX provides at least).

A NASTRAN code with the boundary conditions changes is shown below.

```
SOL 105  
CEND  
...  
K2GG=KAAX  
P2G=PAX  
...  
SUBCASE 1  
MPC=1  
...
```



# AERSYS KNOWLEDGE UNIT

AERSYS-7007

Author: Antonino Vicente Rico

Date: 19/06/2013

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

## BUCKLING WITH SUPERELEMENTS

```

SUBCASE 2
METHOD = 1
...
BEGIN BULK
$111111122222223333333344444444555555556666666677777777
...
RBE2  1111  1111  123  111
...
MPC   1  1111  456  1.  111  456  -1.
...
DMIG  KAAX    0  6  2  0          2160
DMIG* KAAX          111    1
*      1111      1 5.8E+01
...
DMIG  PAX    0  9  2  0          1
DMIG* PAX          1    0
*      1111      1-1.8E+01
...

```

It is important to take special care when this change is implemented because the stiffness matrix and the load vector have been obtained in the interface nodes. Due to the fact that new nodes are introduced as interface nodes, nodes ID of the KAAX and PAX should be replaced with the new ID interface nodes.