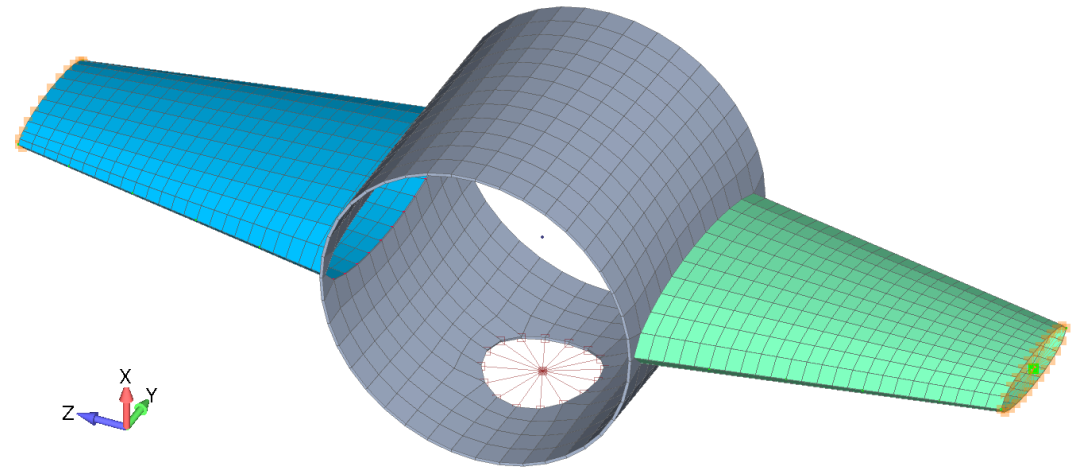


Femap with NX Nastran Tutorial: Working with Multiple Superelements



Thomas Grauer

In aerospace industry a commonly used scenario is:

- a supplier company performs Finite Element Analysis on some component to be developed;
- OEM or customer company incorporates this component into some system and is interested in FE analysis of the assembled system including the FE model of the component.

Now, suppliers are interested in hiding the technical details of their products. On the other hand, the customer is interested in getting assembly FEA done in the most efficient way.

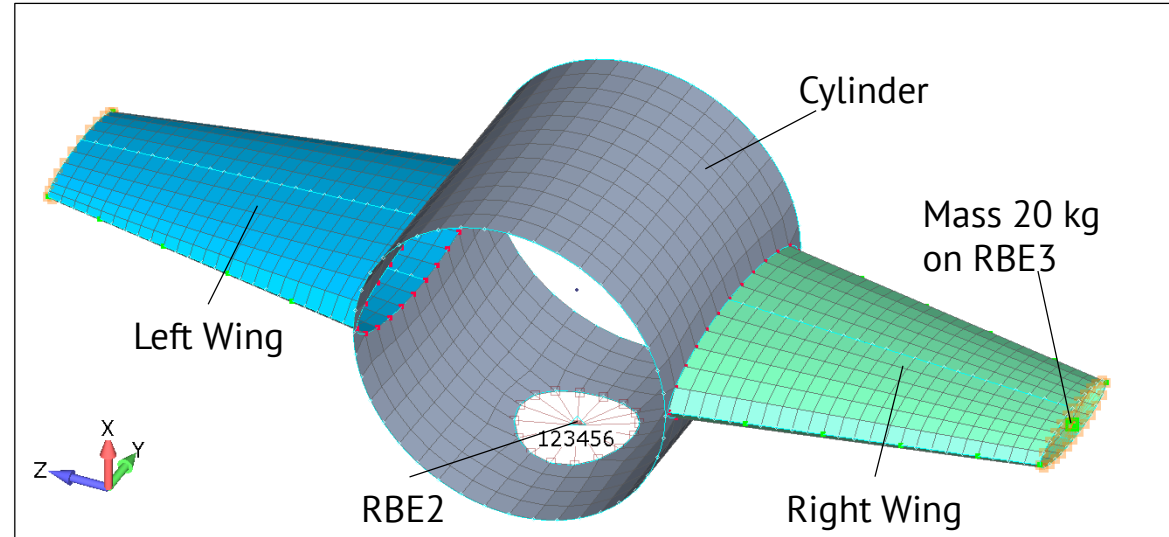
Nastran offers the possibility to generate an **external superelement**, which is a mathematical model of the linear-elastic behavior of the component without any detailed information about its geometry. In fact, stiffness and mass distribution is described by stiffness and mass matrices coupling some predefined boundary nodes of the component. The process of superelement creation is called model reduction. The most prominent reduction method is the **Craig-Bampton method**.

This superelement information is transmitted to the customer and can be assembled with other residual FE data and more superelements for use in an **assembly run**. The number of degrees of freedom of the superelement is much smaller than of the original component FE model, which makes the system run efficient.

Here we describe a step-by-step process for generation of two different superelements, storing their matrix information in OP4 files. Subsequent assembly runs using both superelements and a residual structure are demonstrated.

Example: Cylinder with two Wings

- Aluminum Cylinder with two winglets
- Full model file: FDSE-FullModel.modfem
 - First 15 natural frequencies: 1,6 Hz ... 113,2 Hz
- Superelement generation files:
 - FDSE-RightWingModel.modfem
 - FDSE-LeftWingModel.modfem
- Residual model and assembly run: FDSE-AssemblyModel.modfem



Basic Considerations



- Let us assume: the two wings have to be submitted as reduced FE models (external superelements) to a customer. The reduction process will be performed with Simcenter (NX) Nastran.
- The customer owns the residual model (Cylinder with RBE2 element) and aims to assemble this residual model and the two superelements.
- His goal is to perform modal analysis (0 to 100 Hz) and shock response analysis using the assembled model.
- Furthermore he is interested in the displacements and accelerations of the nodes at the concentrated masses and some more nodes along the wings.
- The customer wishes the mass and stiffness matrices of the superelements in formatted OP4 format. So he can perform assembly runs using MSC Nastran.

What is an External Superelement?



- The nodes of a submodel to be reduced are subdivided in **boundary nodes** and **interior nodes**.
- The reduction process couples the degrees of freedom (DOF) of the boundary nodes in a way, that the stiffness and inertia properties of the submodel are completely described by the boundary degrees of freedom and some extra non-physical degrees of freedom.
- The results are **boundary stiffness and mass matrices** coupling the boundary DOF.
- Submitting the positions of the boundary nodes as well as the reduced stiffness and mass matrices to an assembly model including a residual structure makes it possible to run FE analysis including the submodel without the knowledge of anything about the submodel's internal nodes and elements.
- There are several methods for creating an external superelement:
 - Guyan
 - **Craig-Bampton**
 - Free-Free Dynamic Reduction
- We only recommend the use of the Craig-Bampton Method, which is described in the following.

Boundary Nodes and B-Set

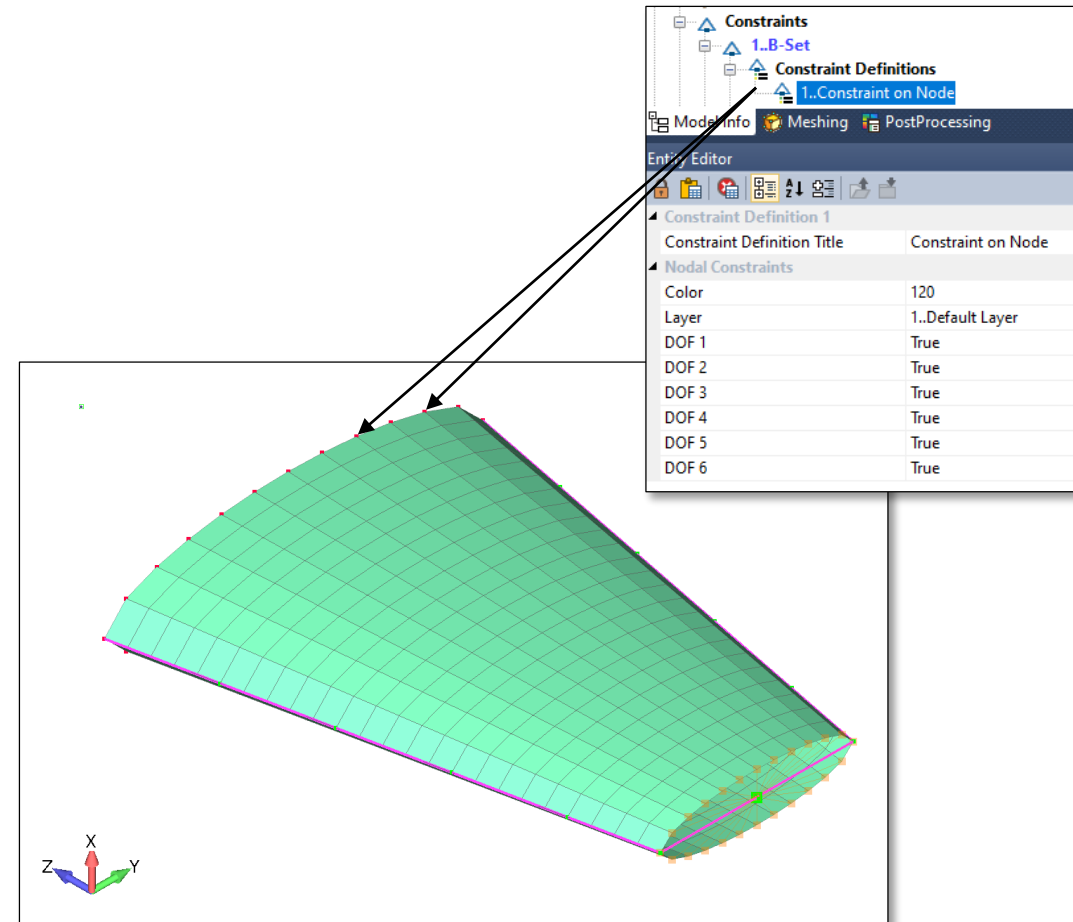


- In the Craig-Bampton procedure the **physical DOFs** of the boundary nodes form the so-called B-Set.
- The B-Set should be chosen such that:
 - Nodes which are physically coupled to the residual structure have to be included.
 - Nodes where loads are defined in the assembly run have to be included.
 - Dependent nodes in RBE2 or RBE3 elements cannot be included.
 - Nodes carrying concentrated mass elements should be avoided.

- In order to get a good approximation of the dynamic behavior of the submodel, we have to introduce further (non-physical) degrees of freedom.
- In the Craig-Bampton procedure these DOFs are constructed according to:
 - A modal analysis of the submodel is performed where all DOFs of the B-set nodes are fixed.
 - This modal analysis should include all modes having natural frequencies up to at least 1.5 times of the interesting frequency range.
 - Amplitudes of these natural vibrations are considered as the extra degrees of freedom, called **modal DOFs**.
- These modal DOFs comprise the so-called **Q-Set**.
- The Nastran A-Set for Craig-Bampton-Reduction comprises the DOFs of the B-Set and the Q-Set.
- The reduced model describes the linear-static behavior of the submodel exactly. The dynamic behavior is approximated. The larger the Q-Set, the better the approximation.

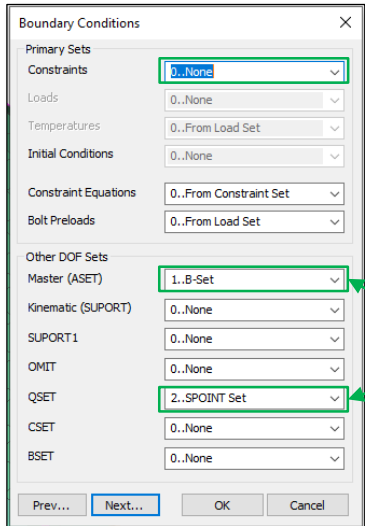
Right-Wing Example: Definition of B-Set in Femap

- In our Right-Wing submodel we create a new Constraint Set (called “B-Set”).
- We define a nodal constraint, selecting all the red connection nodes of the wing to the cylinder. All six DOF fixed.
- The B-Set must be referenced in the **External Superelement Creation** tab of the Analysis Set: see next page, Master (ASET)

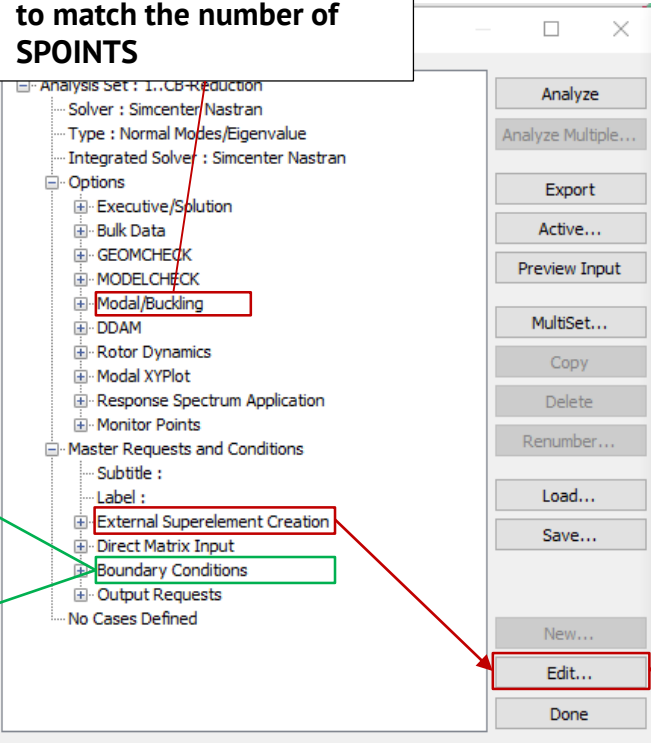


Right-Wing Example: Definition of Q-Set in Femap

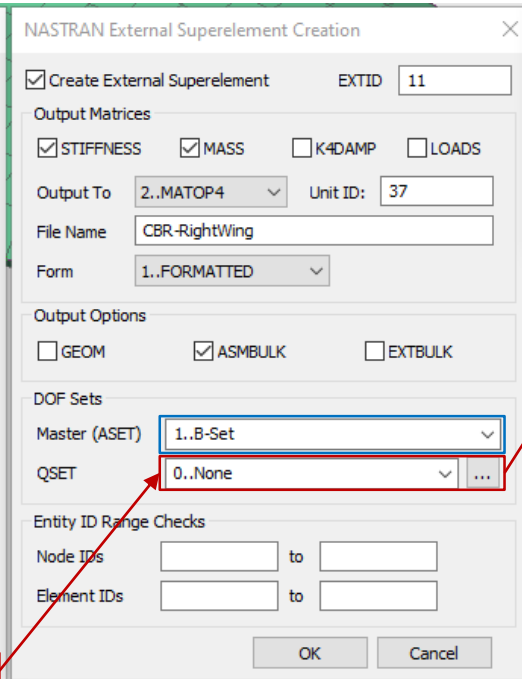
CB Reduction is a modal analysis (SOL 103).
The number of modes have to match the number of SPOINTS



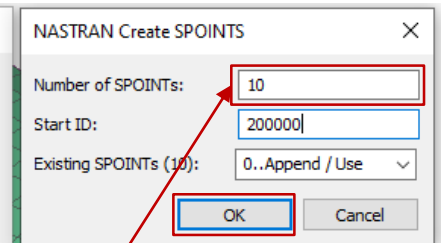
Boundary Conditions dialog box showing various settings. The 'Constraints' dropdown is set to '0..None'. The 'Master (ASET)' dropdown is set to '1..B-Set'. The 'QSET' dropdown is set to '2..SPOINT Set'. The 'Next...' button is highlighted.



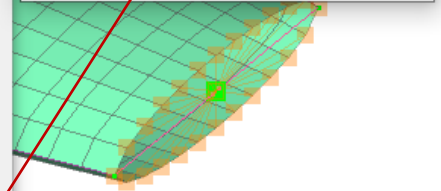
Main Femap interface showing the 'Analysis Set : 1..CB-Reduction' tree. The 'Options' section is expanded, showing 'Modal/Buckling' selected. The 'Master Requests and Conditions' section is also expanded, showing 'External Superelement Creation', 'Direct Matrix Input', and 'Boundary Conditions' selected. The 'Edit...' button is highlighted.



NASTRAN External Superelement Creation dialog box. The 'Create External Superelement' checkbox is checked. The 'EXTID' is set to 11. The 'Output Matrices' section has 'STIFFNESS' and 'MASS' checked. The 'Output To' dropdown is set to '2..MATOP4'. The 'File Name' is 'CBR-RightWing'. The 'Form' dropdown is set to '1..FORMATTED'. The 'Output Options' section has 'ASMBULK' checked. The 'DOF Sets' section has 'Master (ASET)' set to '1..B-Set' and 'QSET' set to '0..None'. The 'Entity ID Range Checks' section has 'Node IDs' and 'Element IDs' fields.



NASTRAN Create SPOINTS dialog box. The 'Number of SPOINTS' is set to 10. The 'Start ID' is set to 200000. The 'Existing SPOINTS (10):' dropdown is set to '0..Append / Use'. The 'OK' button is highlighted.



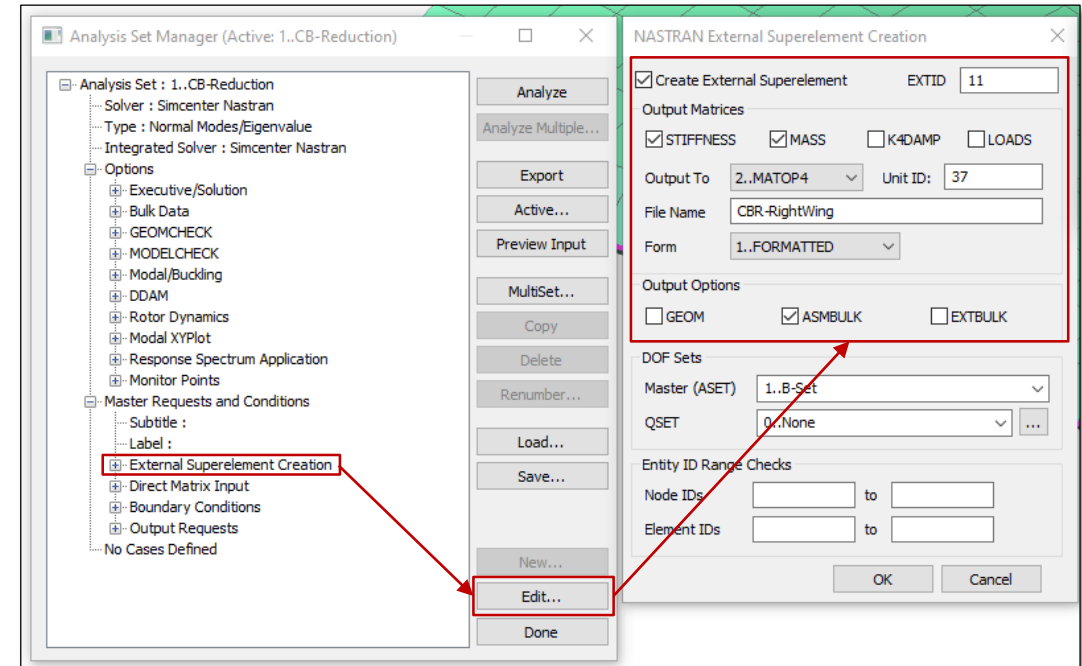
Right-Wing Example: Definition of Q-Set in Femap



- We have to define the modal degrees of freedom in the “External Superelement Creation” tab of the Analysis Set (see previous page). They are defined as a number of SPOINTS in the Nastran input deck.
- In our case: if we include the first 10 eigenvectors then the wing’s dynamic behavior is approximated up to frequencies of 370 Hz.
- This results in:
 - 10 SPOINTS are created in Femap, located in the origin, numbered 200.000 to 200.009.
 - A Constraint Set is created called “SPOINT Set”, where the 10 SPOINTS are fixed.
- Entries in the Boundary Conditions branch of the Analysis Set have to be set according to the previous page. It is important that no primary sets are defined.

Right-Wing Example: Superelement Creation

- There is some more information in NASTRAN External Superelement Creation Dialog of Femap:
 - “EXTID” is the ID of the Superelement used in assembly runs.
 - “Output To MATOP4” means that reduced matrices are stored in a OP4 file.
 - This OP4 file has a FORTRAN output number specified in “Unit ID”.
 - “EXTID” and “Unit ID” have to be unique for every superelement!
 - The OP4 path has to be specified in Field “File Name”.
 - “FORMATTED” has to be specified, if the OP4 file should be in ASCII form.
 - Selection of “ASMBULK” creates an assembly punch file (*.asm) for later use in assembly runs.



Output after Superelement Creation



- A run of the Superelement Creation Analysis Set results in:
 - a superelement punch file (*.pch) with bulk entries used in subsequent assembly runs: B-Set grids and SPOINTS of the SE as well as their boundary connections
 - an assembly punch file (*.asm) with bulk entries used in subsequent assembly runs: connection information between superelement and residual model
 - an OP4 file with the boundary stiffness and mass matrices.
- In a subsequent assembly run the influence of the superelement on the residual structure is more or less perfectly approximated by these data.

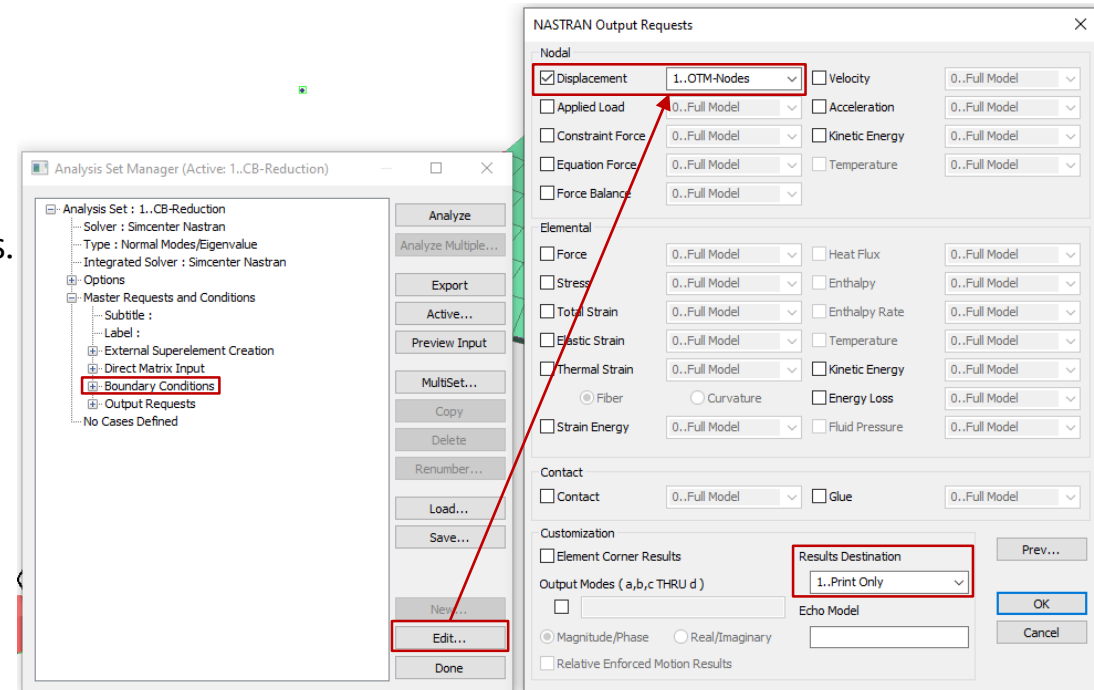
Use of Superelement Interior Node in Assembly Runs



- Up to now, the superelement creation results do not include any information about interior nodes of the superelement. The customer only gets information on the boundary node and their elastic dependence from each other.
- What if the customer wants to output displacement or acceleration information after assembly runs? Then we have to submit more data from superelement creation.
- It is not recommended to include more nodes in the B-Set. In our right wing example an interesting grid for the customer would be the one at the end of the wing (node ID 9.900) carrying the concentrated mass. This grid is a dependent node of the RBE3 element and cannot be used as B-Set node.
- The easiest way to transmit information about such interior nodes to the customer is to define one-dimensional plot-only elements (PLOTTEL) on the interesting interior nodes. On page 8, our right wing example shows the defined PLOTTEL elements in purple color. They connect the interesting interior grids, shown in green.
- If we have PLOTTEL elements in our submodel, the resulting punch file (*.pch) will include these elements and the connected grids.
- Furthermore, we have to transmit so-called **Output Transformation Matrices (OTM)** to the customer. These OTM allow to recover nodal results for the transmitted interior nodes after assembly runs.
- The OTM make it possible to calculate any nodal result of interior nodes from displacements of the boundary nodes.
- In our case, the OTM are then added to the OP4 file in the superelement creation run.
- One should observe not to include OTM for too many nodes. Otherwise the OP4 file would become very large.

Right Wing Example: Output of OTM during Superelement Creation

- First we define a new Femap group called “OTM-Nodes”.
- We include in this group every node on the plot-only elements which are not in the B-Set.
- In the Analysis Set for SE creation this group is referenced in the output requests for displacements and accelerations.
- Never select “Full Model”!
- “Results Destination->Print Only” avoids output of unnecessary OP2 file!



Right-Wing Example: Nastran Input Deck

- Defining the B-Set, the Q-Set and output file information in the Analysis Set will lead to a SOL 103 input deck with two extra lines (red).
- Definition and output of OTM group leads to three more lines (green).
- The CB creation run will result in three files, in our case:
 - cbr-rightwing.op4
 - fdse-rightwingmodel-000.asm
 - fdse-rightwingmodel-000.pch
- All three of them have to be input in the assembly run!

```
1 INIT MASTER(S)
2 ASSIGN,OUTPUT4='CBR-Rightwing.op4',UNIT=37,FORM=FORMATTED
3 NASTRAN SYSTEM(442)=-1,SYSTEM(319)=1
4 ID FDSE,FEMAP
5 SOL SEMODES
6 GEOMCHECK, NONE
7 CEND
8 TITLE = CB-Reduction
9 ECHO = NONE
10 SET 1 = 2006,2018,2027,2032,
11 2037,2042,2327,2332,
12 2337,2342,9900
13 DISPLACEMENT = 1
14 METHOD = 1
15 EXTSEOUT(EXTID=11,STIFFNESS,MASS,MATOP4=37,ASMBULK)
16 BEGIN BULK
```

Left-Wing Example: SE Creation



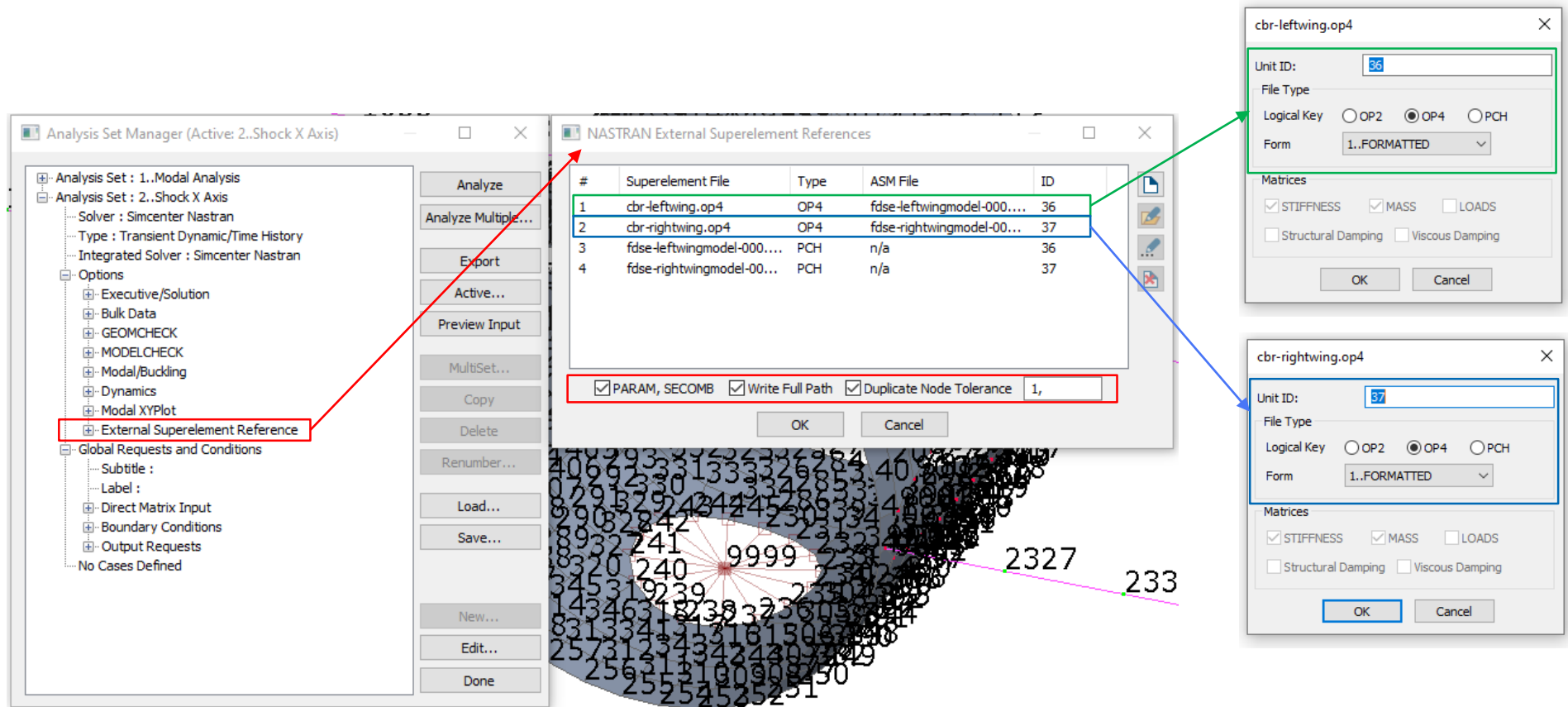
- Now we do exactly the same thing for the left-wing submodel.
- Observe to use different IDs for nodes and SPOINTs in the two submodels. We use 100.000 to 100.009 for SPOINT numbering.
- Superelement ID (“EXTID”) here: 10
- FORTRAN file ID (“Unit ID) here: 36
- The CB creation run will create the following files
 - cbr-leftwing.op4
 - fdse-leftwingmodel-000.asm
 - fdse-leftwingmodel-000.pch

Assembly Run of Two Wings and Cylinder Residual



- Let us assume the customer assembles the left and right wing superelements together with the residual model (cylinder plus RBE2).
- A modal analysis (0 ... 150 Hz) shall be performed.
- In the analysis set, the superelements have to be referenced.
- First we select the *.op4 file and the *.asm file for **every** superelement (see following page). The correct Unit ID and the FORMATTED entry have to be entered.
- Only when every SE was selected this way, the *.pch files for the superelements have to be selected.

Assembly Run of Two Wings and Cylinder Residual



The image displays a software interface for configuring an analysis set. The 'Analysis Set Manager' window shows 'Analysis Set : 2..Shock X Axis' with 'Options' expanded to 'External Superelement Reference'. The 'NASTRAN External Superelement References' dialog box contains the following table:

#	Superelement File	Type	ASM File	ID
1	cbr-leftwing.op4	OP4	fdse-leftwingmodel-000....	36
2	cbr-rightwing.op4	OP4	fdse-rightwingmodel-00...	37
3	fdse-leftwingmodel-000....	PCH	n/a	36
4	fdse-rightwingmodel-00...	PCH	n/a	37

Below the table, the options 'PARAM, SECOMB', 'Write Full Path', and 'Duplicate Node Tolerance' are checked, with the value '1,' entered in the adjacent field.

Two configuration dialog boxes are shown on the right:

- cbr-leftwing.op4**: Unit ID: 36, Logical Key: OP4, Form: 1..FORMATTED. Matrices: STIFFNESS, MASS, LOADS.
- cbr-rightwing.op4**: Unit ID: 37, Logical Key: OP4, Form: 1..FORMATTED. Matrices: STIFFNESS, MASS, LOADS.

A 3D model of a wing assembly is visible at the bottom, with node numbers 9999, 2327, and 233 highlighted.

Assembly Run: Nastran Input Deck

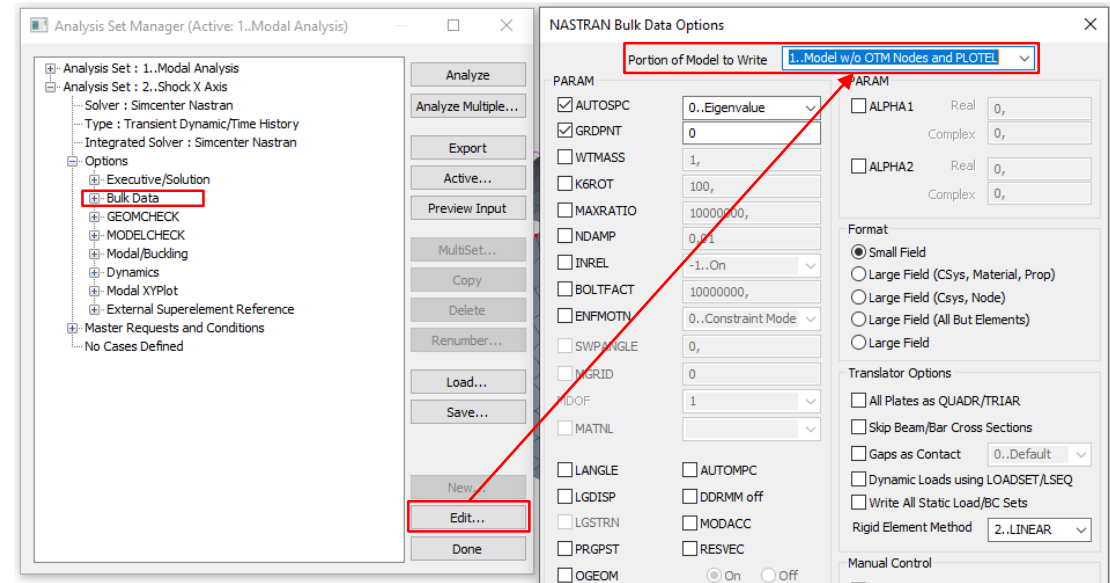


- On top, there are the two ASSIGN statements for referencing the two OP4 files (red).
- Usually there are duplicate Grid IDs in the model (residual and SE boundary nodes). Nastran DPBLKTOL is a tolerance value (blue). Duplicate Grids with coordinates within this value will be removed.
- Parameter SECOMB = YES (green) will lead to combined output of residual model nodes and superelement PLOTEL nodes.
- The INCLUDES for the *.asm files (purple) have to be before any include of a *.pch file (orange). Otherwise connection entries in the assembly files might be ignored.

```
1 INIT MASTER(S)
2 ASSIGN,
3 INPUTT4= [REDACTED],
4 \cbr-leftwing.op4',UNIT=36,FORM=FORMATTED
5 ASSIGN,
6 INPUTT4= [REDACTED],
7 \cbr-rightwing.op4',UNIT=37,FORM=FORMATTED
8 NASTRAN SYSTEM(442)=-1,SYSTEM(319)=1 DPBLKTOL=0.1
9 ID FDSE,FEMAP
10 SOL SEMODES
11 GEOMCHECK, NONE
12 CEND
13 TITLE = Modal Analysis
14 ECHO = NONE
15 WEIGHTCHECK(PRINT,SET=(G,A),
16 GRID=9999,CGI=YES,MASS)=YES
17 DISPLACEMENT(PLOT) = ALL
18 SPCFORCE(PLOT) = ALL
19 METHOD = 1
20 SPC = 1
21 MEFFMASS(PLOT,GRID=9999,FRACSUM) = YES
22 K2GG=K36,K37
23 M2GG=M36,M37
24
25 $----- Bulk Entries:
26
27 BEGIN BULK
28 PARAM,PRGPST,NO
29 PARAM,POST,-1
30 PARAM,OGEOM,NO
31 PARAM,AUTOSPC,YES
32 PARAM,K6ROT,100
33 PARAM,SECOMB,YES
34 ETGR1 1 150. 0 MASS
35 $
36 $
37 $----- Bulk Entries of the residual model
38 $
39 $ .....
40
41
42
43
44 $ .....
45 INCLUDE [REDACTED] \fdse-leftwi
46 ngmode1-000.asm'
47 INCLUDE [REDACTED] \fdse-rightw
48 ngmode1-000.asm'
49 INCLUDE [REDACTED] \fdse-leftwi
50 ngmode1-000.pch'
51 INCLUDE [REDACTED] \fdse-rightw
52 ngmode1-000.pch'
53 ENDDATA
```

Important Notices for Assembly Runs

- Be careful with node numbering:
 - Make sure that the numbering of superelement boundary nodes and respective residual model nodes are equal.
 - If you use multiple superelements: make sure that B-Set nodes, OTM nodes and SPOINTS of every SE are uniquely numbered.
 - Make sure that the residual model does not contain node numbers that equal the superelements OTM nodes.
- If you use PLOTEL elements and OTM nodes from punch files for displaying assembly run results: be careful not to list them twice in the assembly run input file:
 - Run assembly before importing the punch file data.
 - Or restrict assembly runs to a group excluding PLOTEL and related nodes (see right).



Comparison Full vs. Assembly Model Output: Eigenfrequencies

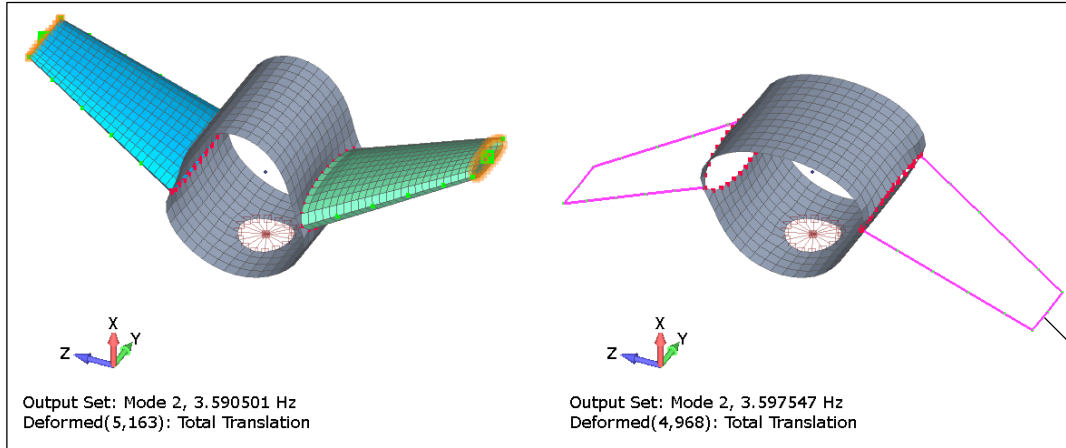
fdse-fullmodel-000.f06 - Notepad2							fdse-assemblymodel-000.f06 - Notepad2																												
316	317	318	319	320	321	322	323	324	325	326	327	328	329	330	331	332	333	1016	1017	1018	1019	1020	1021	1022	1023	1024	1025	1026	1027	1028	1029	1030	1031	1032	1033
MODE NO.	EXTRACTION ORDER	EIGENVALUE	REAL RADIANS	EIGENVALUES CYCLES	GENERALIZED MASS	GENERALI. STIFFNE												MODE NO.	EXTRACTION ORDER	EIGENVALUE	REAL RADIANS	EIGENVALUES CYCLES	GENERALIZED MASS	GENERALI. STIFFNE											
1	1	1.017927E+02	1.008924E+01	1.605752E+00	1.000000E+00	1.017927												1	1	1.019638E+02	1.009771E+01	1.607101E+00	1.000000E+00	1.017927											
2	2	5.089438E+02	2.255978E+01	3.590501E+00	1.000000E+00	5.089438												2	2	5.109433E+02	2.260405E+01	3.597547E+00	1.000000E+00	5.089438											
3	3	1.359615E+03	3.687295E+01	5.868513E+00	1.000000E+00	1.359615												3	3	1.362566E+03	3.691296E+01	5.874880E+00	1.000000E+00	1.359615											
4	4	2.541059E+03	5.040892E+01	8.022828E+00	1.000000E+00	2.541059												4	4	2.547221E+03	5.047000E+01	8.032550E+00	1.000000E+00	2.541059											
5	5	5.407969E+03	7.353890E+01	1.170408E+01	1.000000E+00	5.407969												5	5	5.422150E+03	7.363525E+01	1.171941E+01	1.000000E+00	5.407969											
6	6	6.028588E+03	7.764398E+01	1.235742E+01	1.000000E+00	6.028588												6	6	6.040535E+03	7.772088E+01	1.236966E+01	1.000000E+00	6.028588											
7	7	4.492319E+04	2.119509E+02	3.373304E+01	1.000000E+00	4.492319												7	7	4.497108E+04	2.120639E+02	3.375101E+01	1.000000E+00	4.492319											
8	8	2.326538E+05	4.823420E+02	7.676712E+01	1.000000E+00	2.326538												8	8	2.329866E+05	4.826868E+02	7.682200E+01	1.000000E+00	2.326538											
9	9	2.993565E+05	5.471348E+02	8.707921E+01	1.000000E+00	2.993565												9	9	3.000162E+05	5.477374E+02	8.717511E+01	1.000000E+00	2.993565											
10	10	3.187800E+05	5.646061E+02	8.985985E+01	1.000000E+00	3.187800												10	10	3.190017E+05	5.648023E+02	8.989108E+01	1.000000E+00	3.187800											
11	11	3.280429E+05	5.727503E+02	9.115604E+01	1.000000E+00	3.280429												11	11	3.282349E+05	5.729179E+02	9.118272E+01	1.000000E+00	3.280429											
12	12	3.772621E+05	6.142166E+02	9.775561E+01	1.000000E+00	3.772621												12	12	3.779326E+05	6.147623E+02	9.784245E+01	1.000000E+00	3.772621											
13	13	3.826962E+05	6.186245E+02	9.845714E+01	1.000000E+00	3.826962												13	13	3.835858E+05	6.193430E+02	9.857151E+01	1.000000E+00	3.826962											
14	14	4.315258E+05	6.569063E+02	1.045499E+02	1.000000E+00	4.315258												14	14	4.324628E+05	6.576190E+02	1.046633E+02	1.000000E+00	4.315258											
15	15	5.059592E+05	7.113081E+02	1.132082E+02	1.000000E+00	5.059592												15	15	5.073779E+05	7.123047E+02	1.133668E+02	1.000000E+00	5.059592											

Comparison Full vs. Assembly Model Output: Modal Effective Mass Fractions

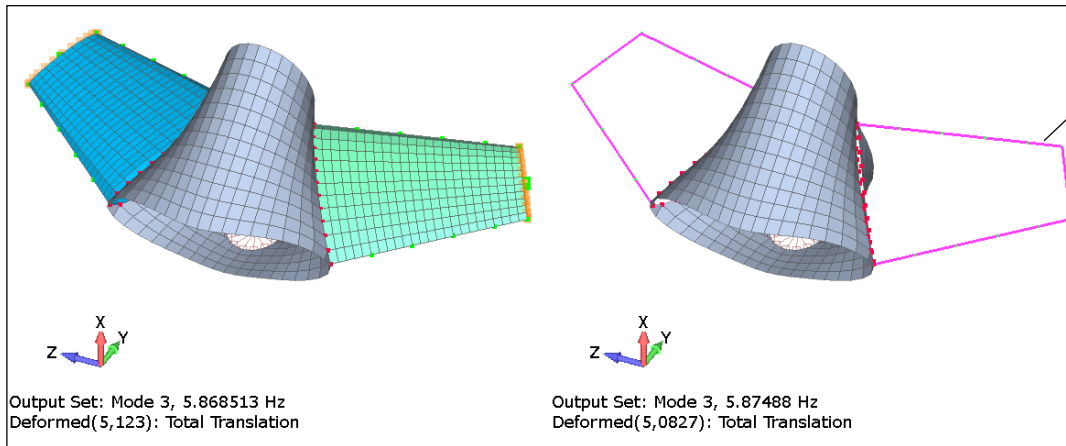


fdse-fullmodel-000.f06 - Notepad2										fdse-assemblymodel-000.f06 - Notepad2									
MODAL EFFECTIVE MASS FRACTION										MODAL EFFECTIVE MASS FRACTION									
MODE NO.	FREQUENCY	FRACTION	T1	SUM	FRACTION	T2	SUM	FRACTION	T3	MODE NO.	FREQUENCY	FRACTION	T1	SUM	FRACTION	T2	SUM	FRACTION	T3
379										1079									
380										1080									
381										1081									
382										1082									
383										1083									
384	1	1.605752E+00	9.963876E-09	9.963876E-09	6.404338E-11	6.404338E-11	6.404829E-02			1084	1	1.607101E+00	1.093392E-08	1.093392E-08	6.334848E-11	6.334848E-11	6.052267E-02		
385	2	3.590501E+00	8.709730E-01	8.709730E-01	3.403740E-09	3.467784E-09	9.058948E-08			1085	2	3.597547E+00	8.709762E-01	8.709762E-01	3.724023E-09	3.787371E-09	8.866315E-08		
386	3	5.868513E+00	3.961489E-09	8.709730E-01	7.799368E-01	7.799368E-01	4.214489E-10			1086	3	5.874880E+00	4.328187E-09	8.709762E-01	7.798386E-01	7.798386E-01	4.426956E-10		
387	4	8.022828E+00	8.031262E-14	8.709730E-01	6.836487E-07	7.799375E-01	2.665105E-09			1087	4	8.032550E+00	1.700151E-12	8.709762E-01	7.249743E-07	7.798393E-01	2.944464E-09		
388	5	1.170408E+01	1.263933E-06	8.709742E-01	3.894122E-10	7.799375E-01	8.673735E-01			1088	5	1.171941E+01	1.275321E-06	8.709775E-01	3.916246E-10	7.798393E-01	8.672256E-01		
389	6	1.235742E+01	9.292975E-04	8.719035E-01	4.897747E-10	7.799375E-01	5.902722E-04			1089	6	1.236966E+01	9.218584E-04	8.718993E-01	5.048879E-10	7.798393E-01	6.048720E-04		
390	7	3.373304E+01	6.829665E-11	8.719035E-01	1.558671E-01	9.358046E-01	9.345626E-14			1090	7	3.375101E+01	7.033120E-11	8.718993E-01	1.559777E-01	9.358170E-01	1.564385E-12		
391	8	7.676712E+01	1.076362E-07	8.719036E-01	2.969032E-11	9.358046E-01	5.996314E-04			1091	8	7.682200E+01	9.963629E-08	8.718994E-01	2.720491E-11	9.358170E-01	5.909690E-04		
392	9	8.707921E+01	2.711623E-10	8.719036E-01	1.388729E-06	9.358060E-01	9.835671E-11			1092	9	8.717511E+01	2.752312E-10	8.718994E-01	1.423526E-06	9.358184E-01	1.028640E-10		
393	10	8.985985E+01	1.087340E-03	8.729910E-01	6.354843E-12	9.358060E-01	1.273496E-06			1093	10	8.989108E+01	1.054688E-03	8.729541E-01	6.351359E-12	9.358184E-01	1.380885E-06		
394	11	9.115604E+01	2.247276E-04	8.732157E-01	8.527519E-12	9.358060E-01	2.731409E-05			1094	11	9.118272E+01	2.217531E-04	8.731759E-01	8.883405E-12	9.358184E-01	2.823969E-05		
395	12	9.775561E+01	1.224140E-10	8.732157E-01	3.694541E-04	9.361755E-01	7.532992E-11			1095	12	9.784245E+01	1.571425E-10	8.731759E-01	3.695222E-04	9.361879E-01	8.020830E-11		
396	13	9.845714E+01	1.247432E-09	8.732157E-01	6.597346E-05	9.362414E-01	7.038795E-13			1096	13	9.857151E+01	1.207989E-09	8.731759E-01	5.650771E-05	9.362444E-01	1.689967E-15		
397	14	1.045499E+02	1.046058E-01	9.778215E-01	2.644766E-10	9.362414E-01	3.553423E-07			1097	14	1.046663E+02	1.045991E-01	9.777749E-01	2.670491E-10	9.362444E-01	1.207885E-07		
398	15	1.132082E+02	1.137926E-06	9.778226E-01	9.337947E-11	9.362414E-01	2.269847E-02			1098	15	1.133668E+02	3.675141E-07	9.777753E-01	9.373113E-11	9.362444E-01	2.264638E-02		

Comparison Full vs. Assembly Model Output: Natural Vibrations



PLOTTEL elements
and nodes imported
from *.pch files



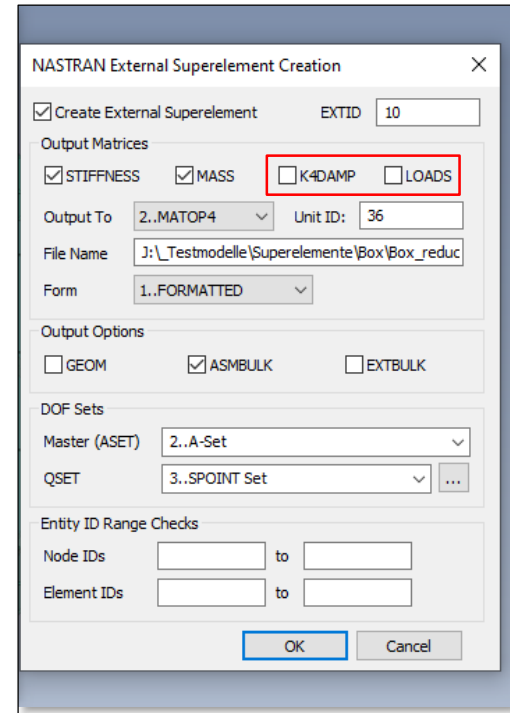
Simcenter Nastran License Consideration



- The superelement creation run is performed by SOL 103. This is included in the NX Nastran Basic Package
- Assembly runs using external superelements need an Advanced Bundle (E520 or E521) license.

Addendum

- If viscous or structural damping is defined in the submodel, damping matrices have to be added to the OP4 file: K4DAMP
- If loads are added to parts of the submodel, which are constant in every assembly run, load matrices have to be added as well: LOADS
- It is possible to calculate OTMs for stresses and forces of submodel interior elements. A Femap group for those elements has to be created and referenced in the output requests of the creation run (analogous to page 14). The element and grid data for these elements has to be transmitted manually to the customer for data recovery in assembly runs.
- In principle, a superelement can be reused several times in an assembly run at different spatial coordinates. In this case the *.asm file of this superelement cannot be used directly. The SECONCT entries there have to be edited manually or a SELOC has to be used for coupling residual nodes and superelement boundary nodes.



NASTRAN External Superelement Creation

Create External Superelement EXTID 10

Output Matrices

STIFFNESS MASS K4DAMP LOADS

Output To 2..MATOP4 Unit ID: 36

File Name J:_Testmodelle\Superelemente\Box\Box_reduc

Form 1..FORMATTED

Output Options

GEOM ASMBULK EXTBULK

DOF Sets

Master (ASET) 2..A-Set

QSET 3..SPOINT Set

Entity ID Range Checks

Node IDs to

Element IDs to

OK Cancel