Obtaining Reduced Order models from FEA

Brumund Philipp, Dehaeze Thomas

October 2020

Contents

Info on file

1   Introduction

A typical design process workflow for a new mechatronic design is shown in Fig. 1.1. The step "first principle modular mathematical modeling" can often be managed with classical kinetic analysis. The *modular* character of the established models is important to easily cycle through different "design optimization" loops as illustrated.

In particular from the "detailed modular mathematical modeling" on, often a numerical simulation-based modeling approach becomes very interesting as many multi body systems become quite complex in terms of mathematical complexity. In mechatronic design today, such models are frequently created using *Simscape*, the multi body module in *Simulink/MATLAB* [4]. This allows to obtain a modular model controlled from a set of parameters. However, the standard *Simscape* blocks are limited to simple geometrical structures and stiffness links.
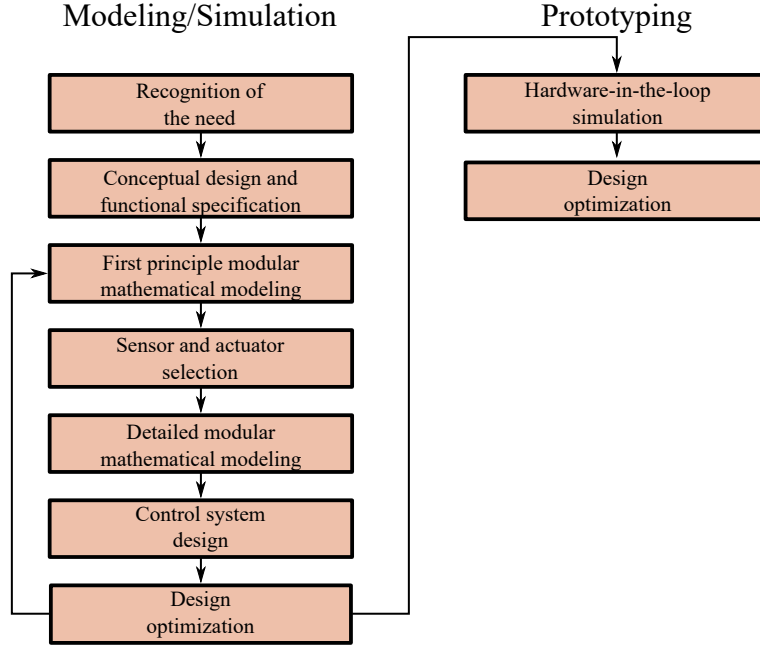
Figure 1.1: Typical mechatronics system design process workflow as presented by [7, p. 7].

To overcome the limitation to simple geometries, *Simscape* includes the option to add reduced order "flexible bodies" and thus model complex structures in terms of stiffness and mass matrices. The term reduced order refers to the reduction of a complex finite-element (FE) model with a huge number of degrees of freedom (DoF) to a reduced system with small number of *retained* degrees of freedom, usually the relevant ones for the treated problem. The reduced set of DoFs is usually referred to as interface DoF (or nodes), boundary node DoFs or master node DoFs, directly referring to the FE-analysis terms and the sense of use: the retention of the DoF that are needed later for application of forces, adding of mass or to extract measured displacements. The reduced system consists then of the reduced order stiffness and mass matrices from the original full matrices of the FE model.

In FEA terms the so-obtained reduced order elements are called super-elements, since after the reduction the behaviour of the whole structure is expressed by reduced order matrices for a chosen remaining subset of master DoFs.

In this work we present the combined workflow to create reduced order systems for complex structures using *ANSYS* which can subsequently be used in *Simscape* or for any other analysis requiring stiffness and mass relations of mechanical structures, e.g. state space models or similar [5].

2 Ansys – Super Element Extraction

This section explains the procedure to follow to create a superelement of a given structure. After this introduction, an overview of the theory behind today's used reduction procedures in structural finite-element-analysis (FEA) is given. They are namely the static Guyan-reduction [6] (which can also be used to generate the for dynamics analysis needed mass matrices) and the component mode synthesis. Different component mode synthesis procedures exist, here we cover the most used Craig-Bampton-method [3].

These methods outputs matrices (mass and stiffness), and thus are representing linear effects. Non-linear effects such as contacts, or special material models canot be modelled using such method.

It is important to note that the explained reduction methods are executed almost automatically by a suitable FEA software after the required user input. The theoretical background is rather given for the proper understanding of typical discrete systems.

Explications are based on the cited original papers as well as the *ANSYS* help [1, 2]. Mathematical notation is orientated on the ANSYS notation, e.g. $\mathbf{K}_{ms}$ referring to a sub-matrix of the stiffness matrix $\mathbf{K}$ with dimensions of m rows (number of chosen retained master DoFs) and s columns (number of not-retained slave DoFs of the FEA model).

2.1 Theory

The typical dynamical matrix equation of motion treated in FEA is

$$\mathbf{M}\,\ddot{\mathbf{u}} + \mathbf{C}\,\dot{\mathbf{u}} + \mathbf{K}\,\mathbf{u} = \mathbf{F} \tag{2.1}$$

where $\mathbf{M}$, $\mathbf{C}$ and $\mathbf{K}$ denote the mass matrix, the damping matrix and the stiffness matrix respectively. The vector $\mathbf{u}$ contains the degrees of freedom of the FEA model (3 displacements per node in 3D models) and $\mathbf{F}$ the applied forces.

The illustration of a super-element in 2 dimensions is shown in Fig. 2.1, where the originally complex shape (the "potato") with full set of DoFs $\mathbf{u}$ is reduced to a small set of "master" DoF $\hat{\mathbf{u}}$ (the ones shown in the figure). The reduced matrices $\hat{\mathbf{K}}$, $\hat{\mathbf{M}}$, $\hat{\mathbf{C}}$ represent the relation between this subset of DoFs. Hence, the goal of the super-element reduction techniques is to reduce the total number of DoFs of a typical FEA model n (usually a couple of ten thousand or hundred
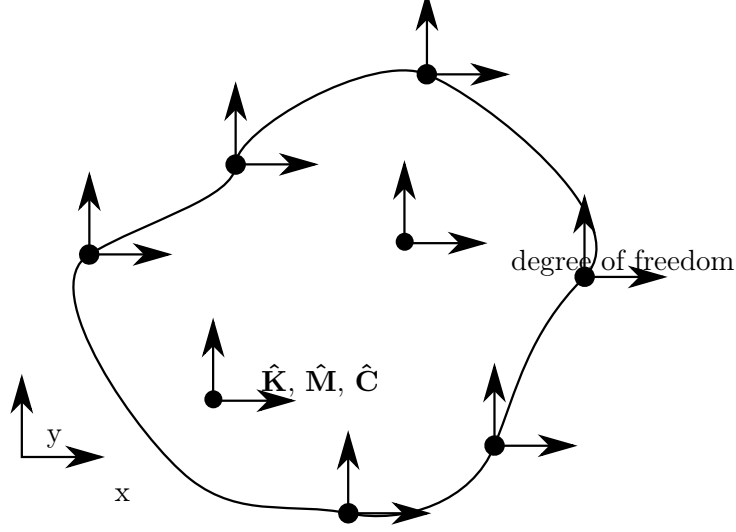
Figure 2.1: Super-element of an arbitrary structure down to a reduced number of degrees of freedoms.

thousand DoFs) to a smaller number of m master DoFs and s slave DoFs, such that:

$$n = m + s \tag{2.2}$$
$$\Rightarrow m = n - s \tag{2.3}$$

2.1.1 Static Guyan reduction

Stiffness matrix reduction   In the static Guyan reduction, named after [6], the static equation of motion ($\ddot{\mathbf{u}} = \dot{\mathbf{u}} = \mathbf{0}$)

$$\mathbf{K}_{n \times n} \, \mathbf{u}_n = \mathbf{F}_n \tag{2.4}$$

with all elements of n dimensions (corresponding to the whole set of n DoFs) is first reorganized such that

$$\begin{bmatrix} \mathbf{K}_{mm} & \mathbf{K}_{ms} \\ \mathbf{K}_{sm} & \mathbf{K}_{ss} \end{bmatrix} \begin{bmatrix} \mathbf{u}_m \\ \mathbf{u}_s \end{bmatrix} = \begin{bmatrix} \mathbf{F}_m \\ \mathbf{F}_s \end{bmatrix} \tag{2.5}$$

which therefore splits the displacement vector $\mathbf{u}$ in master DoFs $\mathbf{u}_m$ and slave degree of freedoms $\mathbf{u}_s$. The master degree of freedoms are selected by the user and contain all necessary DoF to be retained, that can be

- boundary/interface node DoFs for later link the superelement to other model components,

- DoFs at which the displacement needs to be measured in subsequent analysis.

The global stiffness matrix $\mathbf{K}$ is generated by every typical FEA software in the global matrix assembly step of all element matrices of a static analysis.

Solving first the bottom equation in (2.5) for $\mathbf{u}_s$ and inserting in the top equation leads to:

$$\underbrace{\left[\mathbf{K}_{mm} - \mathbf{K}_{ms} \, \mathbf{K}_{ss}^{-1} \, \mathbf{K}_{sm} \right]}_{\hat{\mathbf{K}}} \mathbf{u}_m = \mathbf{F}_m - \mathbf{K}_{ms} \, \mathbf{K}_{ss}^{-1} \, \mathbf{F}_s \tag{2.6}$$

or rewritten shorter

$$\hat{\mathbf{K}}\,\hat{\mathbf{u}} = \hat{\mathbf{F}} \tag{2.7}$$

for the reduced system. $\hat{\mathbf{K}}$ represents the reduced system stiffness matrix.

Constraint modes and coordinate transformation Solving the bottom equation in the algebraic system (2.5) and setting interior forces on slave DoF to zero $\mathbf{F}_s = \mathbf{0}$ yields

$$\mathbf{0} = \mathbf{K}_{sm}\,\mathbf{u}_m + \mathbf{K}_{ss}\,\mathbf{u}_s, \tag{2.8}$$

$$\Rightarrow \mathbf{u}_s = \underbrace{-\mathbf{K}_{ss}^{-1}\,\mathbf{K}_{sm}}_{\mathbf{G}_{sm}}\,\mathbf{u}_m \tag{2.9}$$

The matrix $\mathbf{G}_{sm}$ contains the so called constraint modes, corresponding to *"mode shapes of interior freedoms due to successive unit displacement of boundary nodes, all other boundary nodes being totally constrained"* [3].

With the help of the obtained matrix $\mathbf{G}_{sm}$ a transformation relation between master DoFs $\hat{\mathbf{u}} = \mathbf{u}_m$ and original DoFs $\mathbf{u}$ can be established:

$$\mathbf{u} = \begin{bmatrix} \mathbf{u}_m \\ \mathbf{u}_s \end{bmatrix} = \mathbf{T}\,\hat{\mathbf{u}} \tag{2.10}$$

$$\text{with} \quad \mathbf{T} = \begin{bmatrix} \mathbf{T} \\ \mathbf{G}_{sm} \end{bmatrix} \tag{2.11}$$

which is the typical coordinate transformation for the Guyan reduction.

General static Guyan reduction As shown by Guyan [6] via an approach of the potential and kinetic energy of a discrete system, the more general transformations for both reduced mass and stiffness matrix are

$$\hat{\mathbf{K}} = \mathbf{T}^T\,\mathbf{K}\,\mathbf{T}, \tag{2.12}$$

$$\hat{\mathbf{M}} = \mathbf{T}^T\,\mathbf{M}\,\mathbf{T}, \tag{2.13}$$

with a similar approach for the reduced damping matrix $\hat{\mathbf{C}}$, if required. The reduced mass matrix in equation (2.13) completely expanded is

$$\hat{\mathbf{M}} = \mathbf{M}_{mm} + \mathbf{M}_{ms}\,\mathbf{G}_{sm} + \mathbf{G}_{ms}\,(\mathbf{M}_{sm} + \mathbf{M}_{ss}\,\mathbf{G}_{sm}) \tag{2.14}$$

with $\mathbf{G}_{ms} = \mathbf{G}_{sm}^T$. Thus, the reduced matrix is calculated via stiffness matrix elements contained in $\mathbf{G}_{sm}$.

Important remarks on static Guyan reduction   The stiffness matrix of the reduced system $\hat{\mathbf{K}}$ in equation (2.7) gives exact results for static calculations. The mass matrix of the reduced system $\hat{\mathbf{M}}$, however, is only exact if no generalized inertia is linked to the reduced slave DoFs [5, p. 579]. This is often not the case and leads to small errors in dynamic analysis.

The numerical calculation effort in the Guyan reduction is very light. Only the sub-matrix $\mathbf{K}_{ss}$ needs to be inverted to obtain the reduced stiffness matrix via equation (2.6) or rather to determine $\mathbf{G}_{sm}$ for the transformation matrix (2.11)

2.1.2 Component mode synthesis (modal reduction)

The component mode synthesis is an alternative, numerically heavier method for the reduction of discrete FEA system's matrices. It reduces the global system matrices to a smaller set of slave DoF and "truncated sets of normal mode generalized coordinates" [2, 3].

Again, all global matrices in equation (2.1) are assembled by the FEA software. Then stiffness $\mathbf{K}$, damping $\mathbf{C}$ and mass matrix $\mathbf{M}$ can be re-arranged similarly as in equation (2.5), splitting master DoFs and slave DoFs.

Similarly to equation (2.10) the displacement vector of the global system's DoFs $\mathbf{u}$ can be represented in terms of master DoFs and slave DoFs and linked by a transformation matrix [3]

$$\mathbf{u} = \begin{bmatrix} \mathbf{u}_m \\ \mathbf{u}_s \end{bmatrix} = \mathbf{T}\,\hat{\mathbf{u}} = \mathbf{T} \begin{bmatrix} \mathbf{u}_m \\ \mathbf{y}_\delta \end{bmatrix}. \tag{2.15}$$

As can be seen the difference with the Guyan reduction is the introduction of supplementary DoFs in the reduced model DoF-set $\hat{\mathbf{u}}$ on the right side: the truncated set of generalized modal coordinates $\mathbf{y}_\delta$. Their number is chosen by the user during the modal reduction.

Craig-Bampton method   The Craig-Bampton method, also called *fixed-interface method*, was first presented by Craig and Bampton [3] and is one type of modal reduction. The relevant difference to static reduction is the solution of the Eigenvalue problem (2.16) during the reduction process.

$$\omega^2 \mathbf{M}_{ss}\phi_s = \mathbf{K}_{ss}\phi_s \tag{2.16}$$

The matrices $\mathbf{M}_{ss}$ and $\mathbf{K}_{ss}$ describe the submatrices of the global matrices covering only the slave DoFs (the master DoFs being fully constrained). The determined eigenvectors ϕ_s are the "normal modes of the constrained substructure". This Eigenvalue solution makes the numerical solution of the reduction more complex.

In the Craig-Bampton method the relevant transformation matrix has the form

$$\mathbf{T} = \begin{bmatrix} \mathbf{I} & \mathbf{0} \\ \mathbf{G}_{sm} & \mathbf{\Phi}_s \end{bmatrix} \tag{2.17}$$

where $\mathbf{I}$ denotes the identity matrix of dimension $m \times m$. The sub-matrix $\mathbf{G}_{sm}$ is the same as in the static reduction derived in equation (2.9) and comes from the global stiffness matrix.

The sub-matrix $\mathbf{\Phi}_s$ contains the chosen reduced set of p eigenvectors ϕ_s from the Eigenvalue problem in equation (2.16) of the global system in which all chosen master DoFs are fixed (hence the name "fixed-interface method", as usually the interface nodes of a component are chosen as master DoFs). The number of eigenvalues and eigenvectors to be retained are chosen by the user during the reduction process and the choice is usually based on a desired set of eigenfrequencies to be retained to properly describe the dynamic behavior of the considered element. The total number of retained Eigenmodes in the reduced super-element will be equal to $m + p$ where m is the number of chosen master DoFs m and p the number of additional eigenmodes considered. That becomes also clear from the reduced CMS stiffness and mass matrices which have the same dimensions: $(m + p) \times (m + p)$.

With the Craig-Bampton transformation matrix $\mathbf{T}$ in (2.17) the reduced stiffness, mass and possibly damping matrices are determined as in equation (2.12) and (2.13).

2.2 Procedure in practice

The above described procedure is mostly automatically done by the *ANSYS* solver in a so-called "substructuring generation pass". During this pass the solver generates a superelement matrix file in *ANSYS* binary format (`*.SUB` file). This file can be transformed to a readable textfile format.

The procedure to generate super-element matrices is illustrated in Fig. 2.2. It consists of three sub-steps represented by the "golden blocks": first the "geometry preparation", then the "generation pass" and finally the "matrix writing". Note that during the generation pass, there are two options to generate the .SUB file: one using *ANSYS Mechanical* and one using *Mechanical APDL*. Even tough using Mechanical APDL permits more control over the reduction, all further explanations are based on option 1 (superelement generation with the help of ANSYS Mechanical).

Treated example for procedure explanation   We explain the procedure of a super-element generation on the example of a simple beam with relevant parameters stated in Table 2.1. The FE-model is used in 2D for simplicity, but the whole procedure is equivalent for 3D.
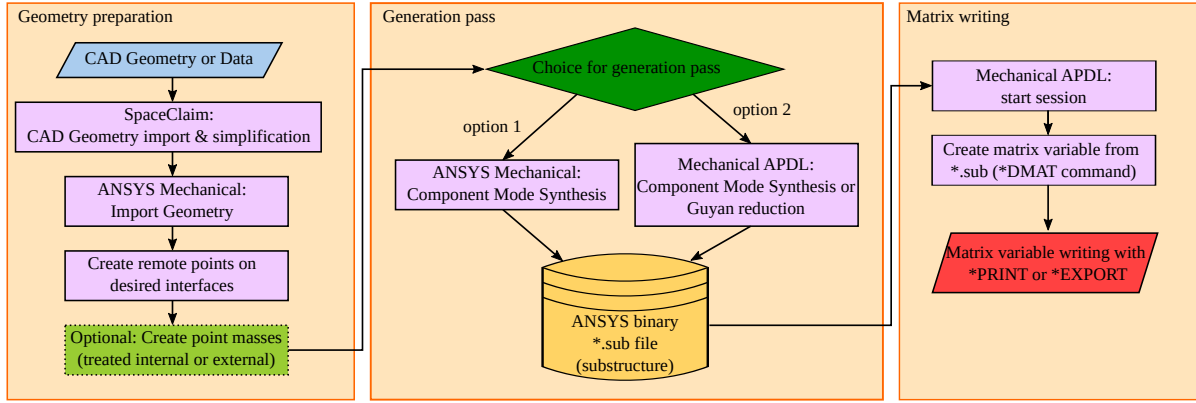
Figure 2.2: Illustration of the superelement reduction process.

Table 2.1: Relevant parameters of treated beam example

Parameter	Value	Unit
h	0.01	m
w	0.01	m
L	0.10	m
E	200	GPa
ρ	7850	$\mathrm{kg\,m^{-3}}$

2.2.1 Geometry preparation

Preparation of the geometry with ANSYS is not covered in-depth at this point, as this document is not meant to be an ANSYS general tutorial. Only the major points are pointed out. The procedure is equivalent to the preparation for a modal analysis.

Any geometry desired can be used to create a super-element: parts or assemblies from an existing CAD file or drawn from scratch. We recommend to use the following tools for the geometry preparation from the *ANSYS* suite:

- Geometry editing: *ANSYS SpaceClaim* (alternatively *DesignModeler* can also be used),

- Further preparation of Model with *ANSYS Mechanical* **in a Modal analysis block**.

The goal of this step is to obtain a model of the original to-be-reduced system in a typical ANSYS Mechanical window as illustrated in Fig. 2.3. The simple given structure of the ANSYS Mechanical tree can be obtained using the following procedure:

1. Import the prepared geometry,

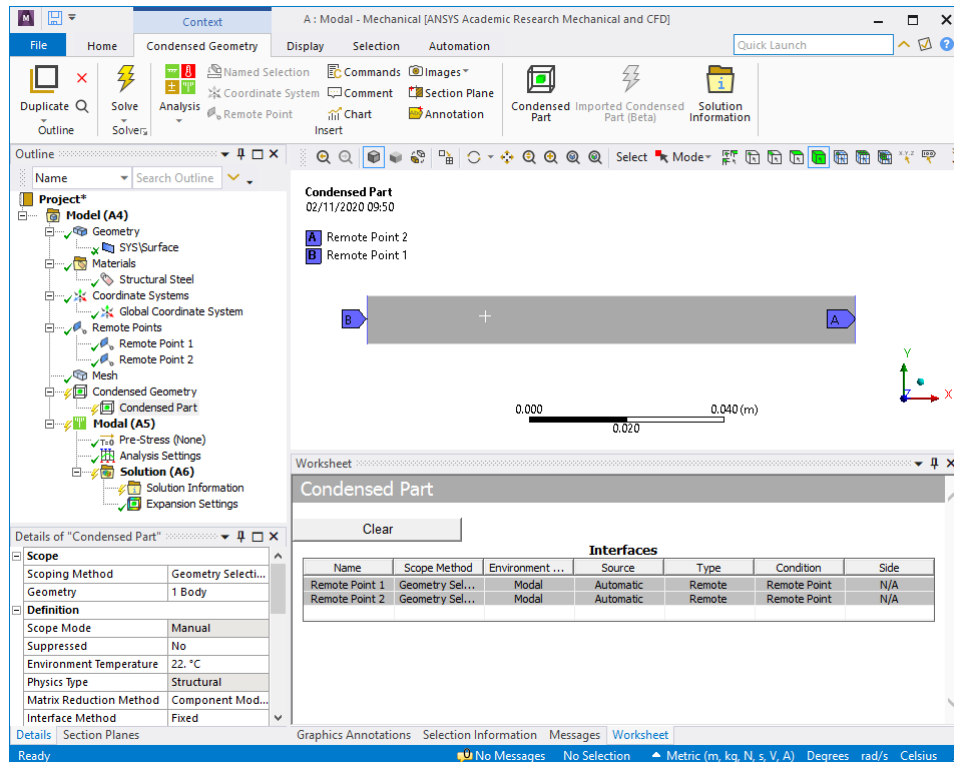2. Select the materials used for one or more parts,

Figure 2.3: Super-element generation settings view in the *ANSYS Mechanical* GUI structure tree with help of the "condensed geometry" functionality.

3. Define remote points if some model geometry (face, edge) with multiple nodes should be handled as one point in the reduced super-element (equivalent to a multi-point constraint of the base nodes)

4. Add point masses if additional weight is to be considered. One has to choose here if the mass should be handled internally (in the reduced geometry) or in the subsequent (Simscape) model. If it is handled in the Simscape model, an additional remote point should be added at the location of the wanted mass.

5. Optionally **in case of assemblies**: Connections or joints can be added to link parts in assemblies as long as they are linear ("bonded" or "frictionless" contacts). Otherwise one can also use shared topology.

After the preparation of the model, the **condensed geometry** is added to the modal analysis tree (see Fig. 2.3). This is how the ANSYS Mechanical GUI calls the generated super-elements (added in ANSYS 2020R1). This works as follows:

1. Right-click on *model-insert-condensed geometry*,

2. Right-click on condensed *geometry-insert-condensed part*,

3. Select all bodies to be condensed (here only the beam)-*apply* (at this point there is still a question-mark indicating missing info),

4. Interfaces need to be defined to specify which nodes should be kept of the reduced super-element: *right-click on condensed part-detect condensed part interface.*

5. Only the fixed-interface component mode synthesis is currently supported by the GUI (see Section 2.1.2), but we can select how many modes should be considered additionally for the reduced model. These are the eigenvector columns in $\mathbf{\Phi}_s$. It is therefore important to consider sufficient modes to correctly represent the dynamic behavior of the reduced component, dependant on the later use.

After the above procedure, ANSYS should create a list of all relevant points to be retained after the reduction, as shown in screenshot Fig. 2.3.

Some more notes concerning "remote points"   They are very useful to limit the number of master DoFs. If the remote point menu is not shown: right-click on the model item and selecting the remote point. In the example in Fig. 2.3 both left and right edges were taken as remote points, linked kinematically to the edge nodes and hence leading to typical beam matrices. ANSYS remote points are equivalent to *NASTRAN* RBE2 or RBE3 (rigid body elements) in which the displacements of one pilot node controls the movements of all sub-nodes. Due to the usefulness for super-elements, these points are automatically added as part interfaces.

Note for use of Mechanical APDL for the generation pass   Using Mechanical APDL for the reduction (option 2 in Fig. 2.2) enables the user to define more inputs for the modal reduction, such as:

- Type of reduction: static or component mode synthesis,

- Detailed control of to-be-retained master DoFs (any node and corresponding degree of freedoms can be retained).

The use of the "old-school" APDL interface requires detailed knowledge of this and is not futher covered here. However, one can prepare the geometry as described above and export the model definition as an input file (with button shown in Fig. 2.4(a)). This can be loaded in an APDL session. This step simplifies the model definition significantly.

Important: This has to be done using a static structural block, as otherwise the input file covers only the use pass of the already-generated super-element.
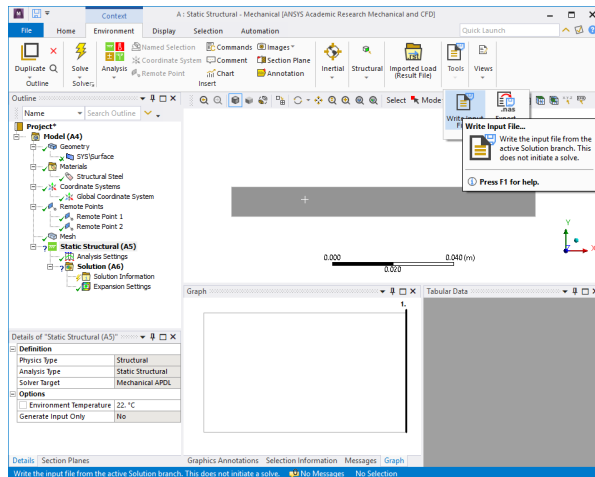
(a) Input file export (for use in *Mechanical APDL*) from previously defined model in *Mechanical*.
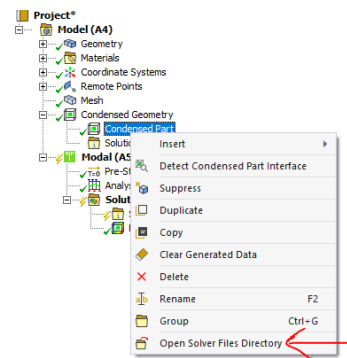
(b) Finding the solver directory which contains the generated files

Figure 2.4: Different useful buttons in ANSYS Mechanical GUI.

2.2.2 Generation pass

Before the actual generation pass we have to create an APDL code to store the info on the used master DoFs and their coordinates for later use in a multi-body model. The code shown in listing 2.1 should be added to the *condensed part* object in the tree (*right click — insert — commands*). Adding this code snippet will create a file `out_nodes_3D.txt` in `c:/ANSYS_temp` containing that info.

The generation pass is relatively simple:

1. right-click on the defined "condensed part" item (see Fig. 2.3) and select *generate condensed part.*

2. copy the generated `*.SUB` file from the *solver directory* to a temporary location for further processing.

The first step generates the super-element `.SUB` file (see Fig. 2.2) in the solver files directory. In the generated solver file (`solve.out`) additional info on the reduction can be found, such as frequency of additionally calculated modes.

To locate the solver directory: right click on *condensed part-open solver files directory* (see Fig. 2.4(b)).

Remarks on `*.SUB` filename   If the described "condensed part" procedure is followed, *ANSYS* Mechanical generates a condensed part folder with generic number for the `*.SUB` file. If the

```APDL
! Write Output to txt file
/OUTPUT,out_nodes_3D,txt,c:/ANSYS_temp

! Get maximum node number in model
*GET, max_node, NODE, 0, NUM, MAX,

! Select only master nodes
NSEL, S, M, , 1, max_node

! List node coordinates for output, then reselect original set
NLIST
NSEL, ALL

! List master nodes and corresponding selected DoFs
MLIST
! Re-direct output to terminal
/OUTPUT,TERM
```

Listing 2.1: APDL code to be added as a snipped to the Condensed Part object to write the master DoF node locations and selected DoFs. Also provided as file: `APDL-codes/write_MDoFs.mac`

APDL GUI is used, the `*.SUB` file carries the name of the current jobname (`jobname.SUB`).

2.2.3 Matrix writing step

The ANSYS binary substructure (or super-element) matrix `*.SUB` file needs to be converted into readable data by other programs. This has to be done using three different APDL functions from within an APDL session (we could also write some program running APDL commands automatically, but here the "manual" way is presented):

- `*DMAT` : to create a matrix variable from a binary *ANSYS* file

- `*PRINT` : to write a matrix to a file

- `*EXPORT` : also to write the matrix to a file, but possibility to select `*.CSV` with user desired number of decimals (sometimes `*PRINT` does not yield sufficiently precise numbers)

The following steps have to be followed to write the files in textformat (Assuming the `*.SUB` file is copied to the path PATH):

1. Start an APDL session,

2. **Important:** change the working directory to PATH (where the previously generated sub file is located)

3. In the command line type the code from code listing 2.2 line-by-line or save and run as

13

a macro using the `*USE` command (`ARG1` representing the generated condensed `*.SUB` file, must be specified as non-case-senstive APDL string, e.g. `'cp123.sub'`).

```apdl
! Delete old files
/DELETE, mat_K.matrix
/DELETE, mat_M.matrix
/DELETE, mat_K.CSV
/DELETE, mat_M.CSV

! Create Matrices K (stiffness) and M (mass) from file specified as 'ARG1'
*DMAT, mat_K, , IMPORT, SUB, ARG1, STIFF
*DMAT, mat_M, , IMPORT, SUB, ARG1, MASS

! Print matrices to a matrix file (3 digits precision - maybe limited)
*PRINT, mat_K, mat_K.matrix
*PRINT, mat_M, mat_M.matrix

! Print matrices to a csv file with specified precision (exponential format)
*EXPORT, mat_K, CSV , mat_K.CSV , 16
*EXPORT, mat_M, CSV , mat_M.CSV , 16
```

Listing 2.2: APDL code to be executed line-by-line or run as a script to export substructure matrices as readable text files. Also provided as file: `APDL-codes/write_mats.mac`

After running these commands, the working directory should contain the mass and stiffness matrix in `*.matrix` and `*.csv` format (with csv having higher precision).

Results from example

The resulting reduced matrices for the described beam model at the beginning of Section 2.2 with data from Table 2.1 *in SI-units* are

$$
\hat{\mathbf{K}} = \begin{bmatrix}
1.94E+06 & 9.71E+04 & -1.94E+06 & 9.71E+04 & 0 & 0 \\
9.71E+04 & 6.52E+03 & -9.71E+04 & 3.19E+03 & 0 & 0 \\
-1.94E+06 & -9.71E+04 & 1.94E+06 & -9.71E+04 & 0 & 0 \\
9.71E+04 & 3.19E+03 & -9.71E+04 & 6.52E+03 & 0 & 0 \\
0 & 0 & 0 & 0 & 9.37E+08 & 0 \\
0 & 0 & 0 & 0 & 0 & 6.17E+09
\end{bmatrix}
$$

$$
\hat{\mathbf{M}} = \begin{bmatrix}
2.91E-02 & 4.09E-04 & 1.01E-02 & -2.46E-04 & 1.17E-01 & 7.32E-02 \\
4.09E-04 & 7.51E-06 & 2.46E-04 & -5.69E-06 & 2.52E-03 & 9.06E-04 \\
1.01E-02 & 2.46E-04 & 2.91E-02 & -4.09E-04 & 1.17E-01 & -7.32E-02 \\
-2.46E-04 & -5.69E-06 & -4.09E-04 & 7.51E-06 & -2.52E-03 & 9.06E-04 \\
1.17E-01 & 2.52E-03 & 1.17E-01 & -2.52E-03 & 1.00E+00 & 0 \\
7.32E-02 & 9.06E-04 & -7.32E-02 & 9.06E-04 & 0 & 1.00E+00
\end{bmatrix}
$$

and are directly read from the generated `*.csv` files. For simplicity of representation these matrices were generated with the APDL GUI, enabling us to select the desired retained Master DoFs on each remote point, where we selected only the vertical UY and rotational ROTZ degree of freedom of each remote point pilot node (compare Fig. 2.3). This leads to the reduced master

DoFs:

$$\hat{\mathbf{u}} = \begin{bmatrix} UY_1 \\ ROTZ_1 \\ UY_2 \\ ROTZ_2 \\ y_{\delta 1} \\ y_{\delta 2} \end{bmatrix} \tag{2.18}$$

with the typical FEA convection of Ui_j describing a translational DoF in i direction of node j and $ROTi_j$ a rotation around i of node j. Generalized coordinates $y_{\delta i}$ are introduced and are due to the selected additional reduced modes in the CMS (see Section 2.1.2). Here we selected 2 additional modes.

The analytical stiffness (Euler theory) matrix for a 2D beam element is

$$\mathbf{f} = \mathbf{K}\,\mathbf{u} \tag{2.19}$$

$$\begin{bmatrix} f_{1y} \\ m_1 \\ f_{2y} \\ m_2 \end{bmatrix} = \frac{EI}{L^3} \underbrace{\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}}_{\mathbf{K}} \begin{bmatrix} y_1 \\ \varphi_1 \\ y_2 \\ \varphi_2 \end{bmatrix} \tag{2.20}$$

and the analytical mass matrix (lumped mass matrix)

$$\mathbf{f} = \mathbf{M}\,\ddot{\mathbf{u}} \tag{2.21}$$

$$\begin{bmatrix} f_{1y} \\ m_1 \\ f_{2y} \\ m_2 \end{bmatrix} = \frac{\rho A L}{420} \underbrace{\begin{bmatrix} 156 & 22L & 54 & -13L \\ 22L & 4L^2 & 13L & -3L^2 \\ 54 & 13L & 156 & -22L \\ -13L & -3L^2 & -22L & 4L^2 \end{bmatrix}}_{\mathbf{M}} \begin{bmatrix} \ddot{y}_1 \\ \ddot{\varphi}_1 \\ \ddot{y}_2 \\ \ddot{\varphi}_2 \end{bmatrix} \tag{2.22}$$

Comparing the analytical values with the FE-values from the reduced model reveals a very small difference. This difference can be due to shear stress negligence of the Euler beam theory for slender beams (for the stiffness matrix). Generally the comparison confirms the method.

As described by Craig and Bampton [3], the additional entries due to generalized modal coordinates (of the last 2 rows in the example) in $\hat{\mathbf{K}}$ and $\hat{\mathbf{M}}$ are linked to the natural frequencies ω_j of the substructure normal (fixed-interface) modes by:

$$\hat{k}_j = \omega_j^2 \hat{m}_j \tag{2.23}$$

where the jth index describes the additional entry index, e.g. in the example $j = 1$ corresponds to

$$\hat{k}_1 = \omega_1^2 \hat{m}_1 \tag{2.24}$$

$$9.37 \times 10^8\,\mathrm{N\,m^{-1}} = \omega_1^2 \cdot 1\,\mathrm{kg} \tag{2.25}$$

$$\Rightarrow \omega_1 = 30\,614\,\mathrm{rad\,s^{-1}} = 4872\,\mathrm{Hz} \tag{2.26}$$

which is exactly the first mode of the described beam with both ends fixed.

Bibliography

[1] *Ansys® Academic Research Mechanical APDL, Release 2020 R2, Help System. Substructuring Analysis Guide*. 2020. ANSYS Inc.

[2] *Ansys® Academic Research Mechanical APDL, Release 2020 R2, Help System. Theory Reference*. 15.6. Substructuring Analysis. 2020. ANSYS Inc.

[3] Roy R. Craig and Mervyn C. C. Bampton. "Coupling of Substructures for Dynamic Analyses". en. In: *AIAA Journal* 6.7 (July 1968), pp. 1313–1319. ISSN: 0001-1452, 1533-385X. DOI: 10.2514/3.4741.

[4] Simulink Documentation. *Simulation and Model-Based Design*. 2020. URL: https://www.mathworks.com/products/simulink.html.

[5] G. Genta. *Introduction To The Mechanics Of Space Robots*. Space Technology Library 26. Dordrecht [Netherlands] ; New York: Springer, 2012. ISBN: 978-94-007-1795-4.

[6] Robert J. Guyan. "Reduction of Stiffness and Mass Matrices". en. In: *AIAA Journal* 3.2 (Feb. 1965), pp. 380–380. ISSN: 0001-1452, 1533-385X. DOI: 10.2514/3.2874.

[7] D. Shetty and R.A. Kolk. *Mechatronics System Design*. Cengage Learning, 2010. ISBN: 9781133169482. URL: https://books.google.fr/books?id=jVcIAAAAQBAJ.