Structural dynamics simulation of a swinging fairground attraction using a novel implementation of reduced order modeling

J.B. Blaauw

University of Twente, Drienerlolaan 5, Enschede

Abstract

This research combines the strengths of modal order reduction, Simscape Multibody and Ansys to come up with a novel method to determine transient stresses in a fairground attraction. Currently, many of these structures are designed using static load cases. This study shows that these load cases can overestimate the stresses greatly in highly dynamic structures. Also, dynamic effects like vibrations within the structure are not considered, possibly resulting in underestimates of the stresses in other places. This method is therefore not suitable for structures like fairground rides. On the other hand, full transient structural analyses are computationally expensive to run. The method outlined in this paper obtains a similar accuracy compared to a full transient study, but by using only a fraction of the computational time. This also enables the model to be used in real time applications of determining the dynamic behavior of structures. Furthermore, as this model is built up within Simulink, the powerful tools within the electrical and control domain provided by Simulink are easy to apply on this structural model. A fairground attraction is used as a practical scenario to apply this method on. The paper concludes by extensively comparing the results of the new method with the load cases and full transient structural analysis.

Keywords: Modal order reduction, Simscape Multibody, Finite-element import method, Transient structural dynamics, Load cases, Fairground attraction, Flexible multibody dynamics

1. Introduction

Accurate and easy modeling of dynamic structures enables designers to effectively optimize the geometry and weight of a design. These aspects are of great importance considering fairground attractions, since large swinging motions and rotating masses induce high dynamic loadings. These motions, in combination with possible vibrations occurring within the structure, means material fatigue is an important consideration. On the other hand, these attractions must be easy to transport and assemble to travel from fairground to fairground. Therefore, it is essential for these attractions to be light weight. The assessment between strength and weight calls for an accurate modeling technique to get to a design both lightweight and strong.

Classical linear FEM is often used to determine the deformation and stresses of such attractions. For example, one could model the structural components of moving parts as a rigid multibody system. Constraint forces over time can be determined at the joints between bodies from which load cases can be derived. Although giving a general idea about the dynamic response and internal loads in this structure's bodies, more complex dynamic behavior like the intricate influence on deformation of one body on other bodies cannot be derived with this quasi-static approach.

Conversely, looking at moving bodies that are not rigidly connected to the fixed world, for example the seating structure of the ride, the movements are large and geometrically nonlinear. They therefore cannot be simulated using linear FEM. Also, as will be shown, the current state of the art using quasi static load cases is not appropriate to accurately describe the stresses in these bodies, due to them not being connected to the rigid world. Besides this, these load cases are also unable to predict vibrations and resulting metal fatigue within these bodies.

A different approach would be to perform a full flexible analysis using nonlinear FEM. Here, each body gets meshed and solved using nonlinear FEM at the same time. This enables to see how bending and dynamic behavior of bodies influence the structure and constraint forces. The main disadvantage of this method is that this nonlinear model is computationally expensive to run. Also, solving for a full mesh for each time step results in a lot of degrees of freedom to solve for and a nonlinear FEM model is hard to reduce.

In an effort to solve aforementioned shortcomings, this paper will describe a novel method capable of solving these geometrically nonlinear structures whilst keeping computational costs low. This is done by combining the strengths of the existing software packages Simscape Multibody, Matlab and Ansys. Using this method, one can quickly set up a study evaluating the dynamic behavior of these

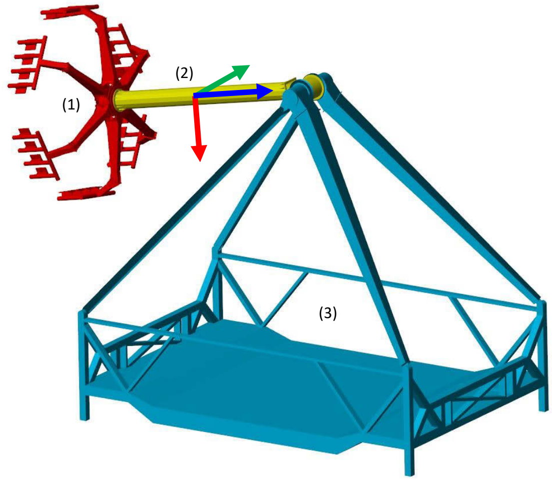

Figure 1: Layout of a KGM Afterburner. The main beam (2) swings backwards and forwards while the seats (1), rotates simultaneously along the axis of the main beam. The frame (3) always remains stationary. The red, green, and blue arrows indicate the location of the local x, y, and z axis, respectively

complex moving structures and quickly analyze transient stresses in the body. This method extends on the method described in [1], which uses modal order reduction to evaluate flexible behavior of bodies within Simscape Multibody. Although the rigid body motions are nonlinear, this method uses a local frame fixed to each body to determine the flexible behavior using linear FEM. With the new approach provided in this paper, tools have been set up to quickly import an existing CAD design into Simscape Multibody. Modal superposition is implemented by combining Ansys and Matlab to determine the mode shapes. This enables the transient stresses within a body to be determined quickly. The complete procedure to set up a model like this, starting with a CAD-assembly to simulating flexible behavior to getting deformation and stress data, will be discussed extensively. Results will be compared to other methods in order to validate its accuracy and shortcomings.

As a practical load case, a fairground attraction called the Afterburner manufactured by KMG will be used. This attraction consists out of a swinging pendulum driven from a hinge at the top with a rotating frame with seats connected to the bottom end, see Figure 1. This attraction is also representative for many other rides that are pendulum like and consists out of beams. The main focus of this research is to get a fast and accurate flexible model for the main beam of this structure, as this body has a complex but repetitive dynamic loading in multiple directions which are hard to determine using traditional static methods or hand calculations.

The outline of this paper is as follows: Before explaining the new method, the currently used method using load cases will be discussed in section 2. The new method based on [1] will be shortly explained in section 3. Next, the modal order reduction strategy used will be discussed in section 4. After this, the newly developed method will be extensively discussed in section 5. This method will be optimized by validating the results to a full transient study using a cantilever beam in section 6. Applying this method on the Afterburner, the results will be shown and discussed in section 7. The paper will end by recapturing the conclusions in section 8.

2. Industrial state of the art

The current state of the art uses load cases to determine the strength of individual components within such a construction. A commonly used method to set up these load cases starts with a rigid multibody analysis. The constraint forces and moments over time between bodies are recorded. For each time step, loads are combined into data points. For a 3D model, these data points consist out of 3 forces and 3 moments for each interface. The data points can be plotted for each time step in a 6-dimensional graph. Now, a 6-dimensional box can be drawn enveloping each data point. The corners of this box, 64 of them, will contain combinations of the minimum and maximum forces and moments that occurred during the simulation. These combinations, multiplied with the required safety factors enforced by the regulations, result in load cases which in turn are solved for within a flexible FEM modal in a static way. Note that these combinations are not occurring in real live but are commonly (much) fiercer. The thought is that if a structure can withstand these corner points of the envelope, it can withstand all combinations inside this envelope. As will be shown later in this paper, this leads to rather conservative designs. Using methods that more closely resemble the occurring stresses within a structure will often lead to large material and weight savings compared to using aforementioned load cases.

A package that can be used to perform such a rigid multibody simulation is Simscape Multibody. This is an extension package on Matlab Simulink. Rigid multibody structures can easily be set up in a method comparable to how Simulink works. The main differences are that Simulink blocks here correspond to bodies or joints instead of (transfer) functions, and connections between blocks do not correspond to numerical data, but rigid connections between bodies of the structure. Tools exist that can import assemblies from CAD programs directly into Simscape as rigid multibody models. A strong advantage of Simscape Multibody is that these models can use the powerful tools of Simulink to be applied on rigid multibody structures. It is, for example, easy to set up multidisciplinary studies, as control systems and electrical components can be included without much effort. Also, the user interface will be easy to understand when one has some experience with Simulink.

Modeling the Afterburner, and many similar (fairground) attractions, is straightforward. Looking at Figure 1, only 2 actuated joints are present: one enabling the pendulum motion located between the top of the main beam (2) and the rigid frame (3), and one enabling the rotation of the seat located between the main beam (2) and seat structure (1). A simplified view of this model in Simulink can be seen in Figure 2.

The sweeping motion of the beam is controlled by a flip-flop controller, whilst the rotating motion of the seat is kinetically driven to ramp up to a constant speed. The study starts from a static situation, and slowly ramps up to a sweep angle of 120°. The seats speed up to a speed of 15 rpm.

The constraint forces and moments acting on both the top and bottom of the beam are logged. These points can be processed into load cases as described in the beginning of this section. However, two sets of load cases can be extracted from this data: one where the load cases are applied on the top of the beam using the constraint forces and moments from this top frame, and one with the load cases applied on the bottom of the beam using the constraint forces and moments of the bottom frame. As both sets represent the same structural problem, both sets should result in similar stresses. However, as will be shown in subsection 7.1, this is not the case, again indicating that using load cases is not an accurate method of determining the stresses within a dynamic structure.

3. New method: Flexible bodies in Simscape Multibody

As discussed in the previous section, Simscape Multibody is initially only intended for rigid multibody structures. Recently, however, a new method has been proposed by (S. Miller et al. 2017) enabling this software to also be used for flexible multibody structures. This is done by superimposing flexible behavior up on the previously defined rigid bodies. The most promising method discussed is the 'finite-element import method'. Following this method, the previously rigid bodies are replaced by bodies including 'Deflection Joints' at each interface, see Figure 3. The deflection joints measure the motion (displacement, velocity, and acceleration) between the interface of a body and the body itself and calculates the reaction force on these joints according to the linear equation of motion:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{D}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{F} \tag{1}$$

Here $\mathbf{M}$, $\mathbf{D}$ and $\mathbf{K}$ are the local mass, damping and stiffness matrices for the flexible behavior of all the deflection joints of the body, respectively. These matrices represent the mass, damping and stiffness properties that each joint feel when moved relative to its undeformed position. $\mathbf{u}$ is the vector containing displacements and rotation of each frame relative to its undeformed position. The direction of these displacements is expressed in a local frame rigidly connected to the body, from here on called body fixed frame. The flexible behavior of a body is described in the local frame. This means that the displacements of this equation of motion remains small for stiff parts, which means that the assumed linear nature of this system does not introduce large errors due to non-linear flexible behavior. For a 3-dimensional body with 6 degrees of freedom per frame and 3 frames, $\mathbf{u}$ has 18 entries. This also implies that the matrices of Equation 1 in this case must be of size 18×18. $\mathbf{F}$ is the vector with forces and moments applied to the joints that occur because of the inertia, damping and stiffness of the body.

When the matrices are derived, these can be imported into the flexible body model within Simscape. If this is done, the original rigid body block can simply be replaced with the suitable flexible body block. For the Afterburner, this implies replacing the 'Beam Rigid (2)' block from Figure 2 with a block built up like Figure 3. The top and bottom boundary frames of the beam are connected to 'F1' and 'F3' from Figure 3. Frame 'F2' is the body fixed frame which is placed in the center of mass; it has no external connections. No further changes must be made to the Simscape Multibody study, making the process easy if a rigid body model of the structure is already on hand.

4. Modal order reduction

As discussed in the previous section, the mass, damping, and stiffness matrices needs to be reduced to the right format before they can be implemented in the flexible Simscape model. This section will first explain how to derive these matrices for the boundary frames using Craig-Bampton modal order reduction. After this, these matrices will be extended to also include internal modes. Section 5 will continue with explaining how implement these matrices into the flexible multibody model.

4.1. Craig-Bampton boundary modes

The mass, damping and stiffness matrices should be derived for a unit displacement in each degree of freedom (3 translations and 3 rotations) for each boundary node. To achieve this, Craig-Bampton model order reduction is used [2]. To do so, the stiffness matrix is first sorted such that boundary and internal nodes are grouped in the following partitioned form:

$$\bar{\mathbf{K}}_{\mathrm{FEM}} = \begin{bmatrix} \mathbf{K}_{bb} & \mathbf{K}_{ib} \\ \mathbf{K}_{bi} & \mathbf{K}_{ii} \end{bmatrix} \tag{2}$$

The indices i and b correspond the internal and boundary nodes, respectively. The boundary nodes consist out of the previously defined boundary frames and the body fixed frame. $\mathbf{M}_{\mathrm{FEM}}$ is partitioned in the same way. The Craig-Bampton boundary modes can be set up:

$$\mathbf{\Phi}_{\mathrm{CB}} = \begin{bmatrix} \mathbf{I} \\ \mathbf{K}_{ii}^{-1}\mathbf{K}_{ib} \end{bmatrix} \tag{3}$$

Here, $\mathbf{I}$ is an identity matrix. $\mathbf{K}_{ii}$ and $\mathbf{K}_{ib}$ are the quadrants of $\bar{\mathbf{K}}_{\mathrm{FEM}}$. Pre and post multiplying $\bar{\mathbf{M}}_{\mathrm{FEM}}$ and

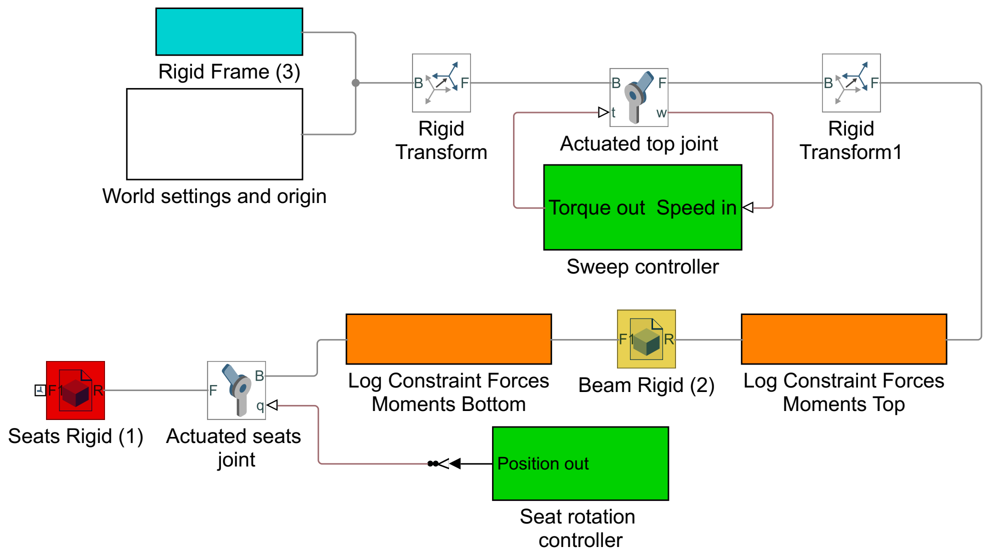

Figure 2: Simplified view of the rigid Simscape Multibody model of the Afterburner. The gray lines are rigid connections between bodies and joints. The body of interest is "Beam Rigid (2)", which has blocks placed on either end to log the constraint forces and moments acting on this body. The numbering of each body corresponds to the numbering in Figure 1

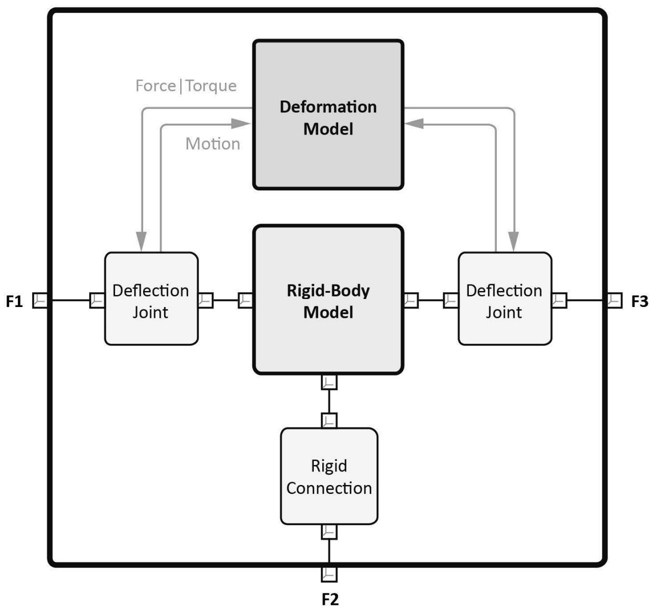

Figure 3: Overview of a flexible body using the finite-import method. Image from [1]

$\bar{\mathbf{K}}_{\text{FEM}}$ give the matrices in the correct form to be used in the Simscape model:

$$\begin{aligned}\mathbf{K}_{\text{CB}} &= \mathbf{\Phi}_{\text{CB}}^T \bar{\mathbf{K}}_{\text{FEM}} \mathbf{\Phi}_{\text{CB}} \\ \mathbf{M}_{\text{CB}} &= \mathbf{\Phi}_{\text{CB}}^T \bar{\mathbf{M}}_{\text{FEM}} \mathbf{\Phi}_{\text{CB}}\end{aligned} \tag{4}$$

The $[\ ]^T$ indicates the transposed of the matrix. The resulting matrices are square and symmetric and have a size of 6 times the number of boundary frames, that is, 3 translations and 3 rotations for each frame. The order in which these frames are sorted corresponds with the order defined in $\mathbf{K}_{bb}$. In the case of the Afterburner, 3 frames are defined, resulting in 18 by 18 matrices. This also means that the number of degrees of freedom has reduced significantly, enabling faster simulation times. The Craig-Bampton modes corresponding to the top and bottom can be seen in Figure 4 and Figure 5 as an example.

4.2. Internal modes

Craig-Bampton modes only define displacement of the boundary frames. In practice however, when these points are fully constrained, parts of the beam can still vibrate. Examples of such internal modes for the beam can be found in Figure 6 and Figure 7. Depending on the geometry and loads on the body, these modes can form a significant part of the total response of the structure. Not taking these modes in to account, may therefore reduce the accuracy of the model. To solve this, the Craig-Bampton matrices can be extended to also include these internal modes. To do this, first the internal modes must be calculated using the standard equation for eigen modes, but with the boundary frames fixed by removing its corresponding rows and columns from the mass and stiffness

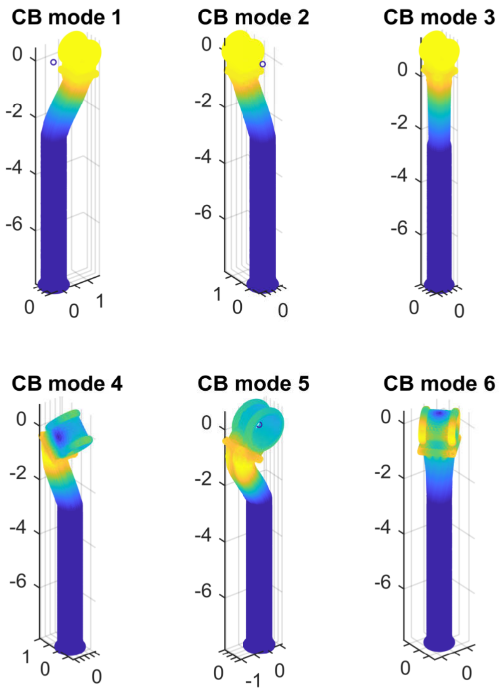

Figure 4: 6 Craig-Bampton modes corresponding to the top boundary frame. The color indicates the displacement relative to the undeformed model. Axis units are in meters

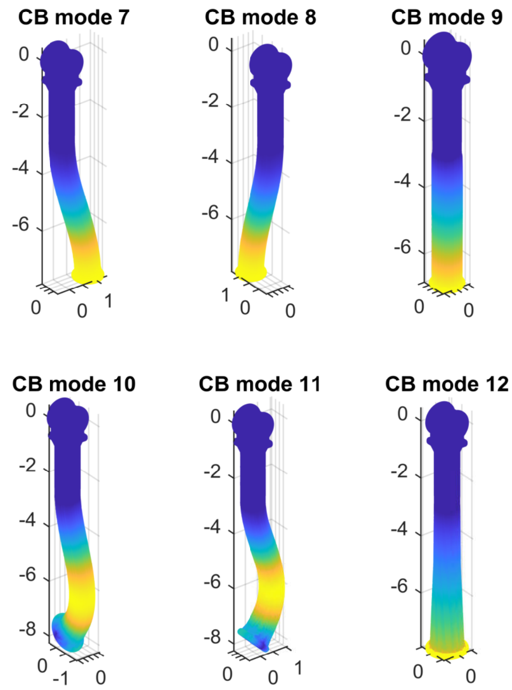

Figure 5: 6 Craig-Bampton modes corresponding to the bottom boundary frame. The color indicates the displacement relative to the undeformed model. Axis units are in meters

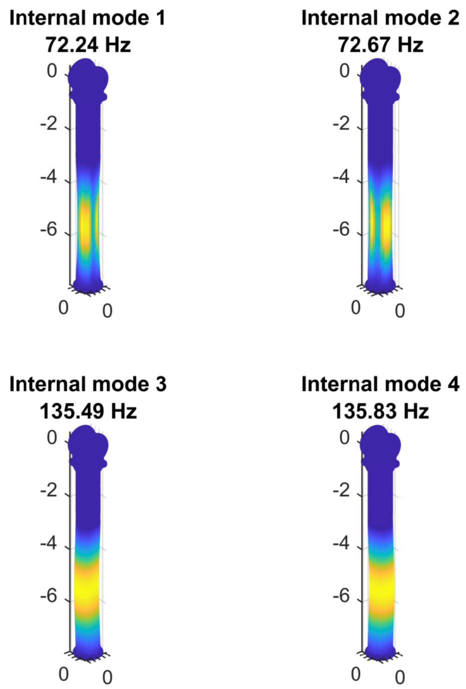

Figure 6: First 4 internal modes of the part of the beam below the center of mass. The color indicates the displacement relative to the undeformed model. Axis units are in meters

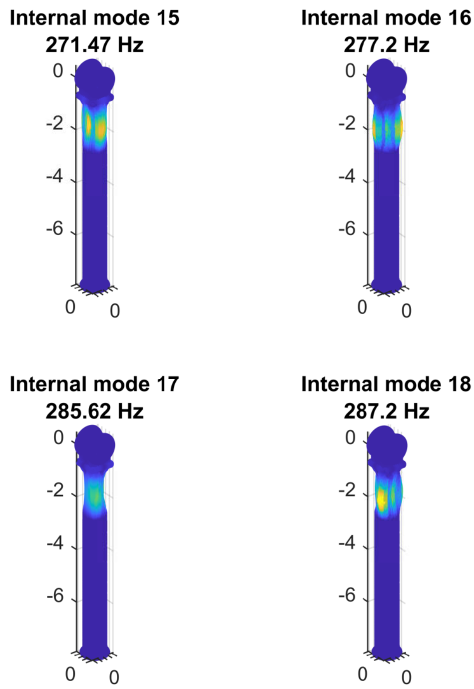

Figure 7: First 4 internal modes of the part of the beam above the center of mass. The color indicates the displacement relative to the undeformed model. Axis units are in meters

matrix:

$$\left(\mathbf{M}_{\text{FEM,fixed}} - \omega_{\text{k}}^2 \mathbf{K}_{\text{FEM,fixed}}\right) \mathbf{\Phi}_{\text{int,k}} = \mathbf{0} \tag{5}$$

where k corresponds to internal mode k.

Although it is possible to consider all internal modes, it is generally not needed for accuracy. Normally, only the few smallest internal modes cause for most of the flexible behavior. For example, according to [3], only internal modes with frequencies up to 4 times the highest frequency present in the constraint forces have to be taken into account; internal modes with frequencies higher than this do not have a significant effect on the total flexible behavior. Therefore, only the eigenvectors in $\mathbf{\Phi}_{\text{int}}$ corresponding to these few modes are taken, here called $\mathbf{\Phi}_n$. This can be appended to Equation 3 like:

$$\mathbf{\Phi} = \begin{bmatrix} \mathbf{I} & \mathbf{0} \\ \mathbf{K}_{ii}^{-1}\mathbf{K}_{ib} & \mathbf{\Phi}_n \end{bmatrix} \tag{6}$$

Pre- and post-multiplying in accordance with Equation 4 results in the mass and stiffness matrix including the internal modes. For each internal mode, 1 extra row and column is added to the matrices. For example, a 3D body containing 2 boundary frames, 1 body fixed frame and 2 internal modes will have $2 \times 6 + 1 \times 6 + 2 = 20$ rows and columns in its matrices.

5. From CAD design to transient stresses

To perform a flexible multibody simulation in Simscape, many properties of the bodies must be processed and imported, like the matrices discussed before and the dimensions of the body. This process is quite involved, as it requires multiple different programs to work together. This study automates this process to a large extend. For this, Matlab is used as the programming environment because of its close integration with Simulink and Simscape. The resulting process is graphically represented in Figure 8. Upcoming part of the paper will discuss each process block of this figure extensively. It is assumed that a CAD design and rigid multibody Simscape study are already available.

5.1. Ansys Workbench

To start, a mesh should be created from the CAD design of each body that needs to be modeled flexibly. From this the mass and stiffness matrices $\mathbf{M}_{\text{FEM}}$ and $\mathbf{K}_{\text{FEM}}$ can be derived for these bodies. Also, boundary frames must be defined to connect other bodies to in the Simscape model. These frames must be rigidly connected to the nearby nodes in the mesh to prevent rigid body modes of these frames. It should be ensured that the nodes corresponding to these frames have both translational as rotational degrees of freedom, as commonly nodes within a FEM mesh do not have rotational degrees of freedom. Furthermore, because the flexible behavior is expressed in a local, body fixed frame, the position of this frame must be chosen. For accuracy, it is best practice to place this frame in the center of mass and connect this frame rigidly to the FEM mesh [4]. Also, this frame should have both translational and rotational degrees of freedom.

For the main beam of the Afterburner, the first frame is placed at the joint near the top of the beam where it connects to the static frame as can be seen in Figure 1. The second frame is placed at the bottom of the beam, where it connects to the seat structure. Furthermore, a frame is placed in the center of mass, connected to the perimeter of the hollow beam.

For this study, Ansys Workbench is used because of the easy to use user interface, making the process more accessible to less experienced people. However, this method is not limited to Ansys Workbench as other FEM programs can also be used, as long as it is possible to obtain the mass and stiffness matrices and the nodal locations. The boundary frames are placed at so-called 'remote points' that are rigidly connected to nearby nodes. The main disadvantage of using Ansys Workbench is the limited options to select only a few nodes to connect these remote points to, which can result in large parts of the body becoming rigid. Other FEM programs might offer more options to prevent this. Next, using an APDL command snippet, $\mathbf{M}_{\text{FEM}}$, $\mathbf{K}_{\text{FEM}}$ and the locations of each node is exported into text files. Depending on the size of the matrices, these files can become large and take long to process. A trade off must be made between the size of these matrices and accuracy of the mesh.

5.2. Data preparation

Now the full mass and stiffness matrices are known, they can be reduced in the same method as discussed in section 4. This makes the matrices much easier to handle, as their size is greatly reduced. Now, a few more steps are required to prepare the data to be imported into Simscape, which will be explained in upcoming sections. First, a damping matrix must be derived from the reduced mass and stiffness matrices. Then, these matrices must be converted into a state space formulation which can then be imported into Simscape. Furthermore, the mass of the body must be divided over its boundary and body fixed frames. Multiple methods of dividing the mass will be discussed in the final subsection.

5.2.1. Damping

Damping is added for this model to mimic the damping properties of the real-live construction. This has the added benefit to prevent energy build op in the flexible bodies, causing the system to become unstable over time.

In the current model, Equation 1, viscous damping can be added straightforwardly. Two common damping models are often used: proportional damping and modal damping [3]. Choosing the damping model and ratios is not straightforward, as this not only depends on material properties, but also on the geometry of the body. In

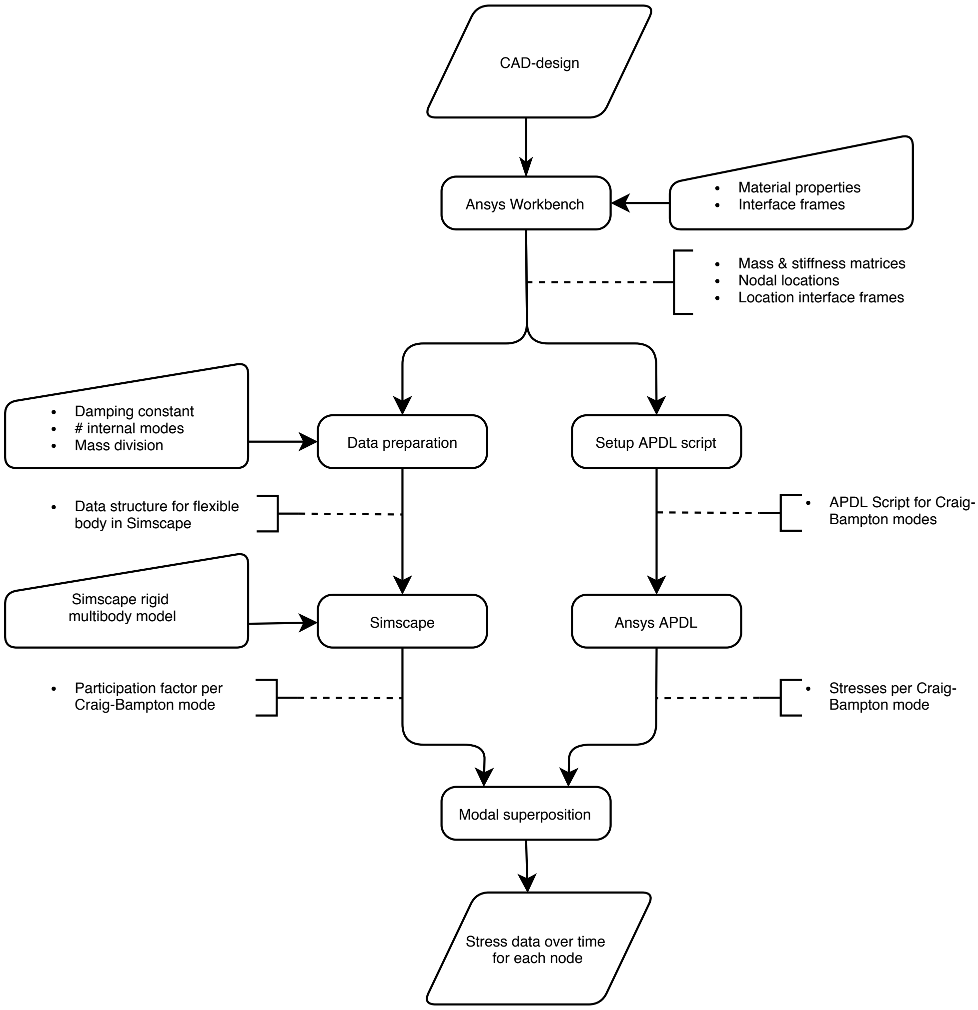

Figure 8: Schematic representation of the workflow

this case, modal damping with a constant damping ratio of 0.05 is chosen to provide predictable and realistic damping behavior for this study. When performing another study, other models and damping ratios may be needed to accurately model the structure.

5.2.2. State Space

The flexible model within Simscape Multibody requires the mass, damping and stiffness matrices to be converted into a state-space system. In accordance with [1], this is executed as follows:

$$\dot{\mathbf{x}} = \mathbf{A}\mathbf{x} + \mathbf{B}\mathbf{u} \tag{7}$$

$$\mathbf{y} = \mathbf{C}\mathbf{x} + \mathbf{D}\mathbf{u} \tag{8}$$

where

$$\mathbf{A} = \begin{bmatrix} \mathbf{0} & \mathbf{I} \\ -\frac{\mathbf{K}_{\mathrm{CB},ii}}{\mathbf{M}_{\mathrm{CB},ii}} & -\frac{\mathbf{L}_{\mathrm{CB},ii}}{\mathbf{M}_{\mathrm{CB},ii}} \end{bmatrix}$$

$$\mathbf{B} = \begin{bmatrix} \mathbf{0} & \mathbf{0} & \mathbf{0} \\ -\frac{\mathbf{K}_{\mathrm{CB},ib}}{\mathbf{M}_{\mathrm{CB},ii}} & -\frac{\mathbf{L}_{\mathrm{CB},ib}}{\mathbf{M}_{\mathrm{CB},ii}} & -\frac{\mathbf{M}_{\mathrm{CB},ib}}{\mathbf{M}_{\mathrm{CB},ii}} \end{bmatrix}$$

$$\mathbf{C} = \begin{bmatrix} -\mathbf{K}_{\mathrm{CB},bi} - \mathbf{M}_{\mathrm{CB},bi}\frac{\mathbf{K}_{\mathrm{CB},ii}}{\mathbf{M}_{\mathrm{CB},ii}} \\ -\mathbf{L}_{\mathrm{CB},bi} - \mathbf{M}_{\mathrm{CB},bi}\frac{\mathbf{L}_{\mathrm{CB},ii}}{\mathbf{M}_{\mathrm{CB},ii}} \\ \mathbf{I} \end{bmatrix}$$

$$\mathbf{D} = \begin{bmatrix} a & b & c \\ \mathbf{0} & \mathbf{0} & \mathbf{0} \end{bmatrix}$$

where

$$a = -\mathbf{K}_{\mathrm{CB},bb} - \mathbf{M}_{\mathrm{CB},bi}\frac{\mathbf{K}_{\mathrm{CB},ib}}{\mathbf{M}_{\mathrm{CB},ii}}$$
$$b = -\mathbf{L}_{\mathrm{CB},bb} - \mathbf{M}_{\mathrm{CB},bi}\frac{\mathbf{L}_{\mathrm{CB},ib}}{\mathbf{M}_{\mathrm{CB},ii}}$$
$$c = -\mathbf{M}_{\mathrm{CB},bb} - \mathbf{M}_{\mathrm{CB},bi}\frac{\mathbf{M}_{\mathrm{CB},ib}}{\mathbf{M}_{\mathrm{CB},ii}}$$

This system uses $\ddot{\mathbf{u}}$, $\dot{\mathbf{u}}$ and $\mathbf{u}$ combined into a vector as input (u) and outputs the reaction forces in each degree of freedom (y). Internal modes are handled inside the state-space system and are part of the state vector (x). As Simulink does not output the state vector in its State-space block, the $\mathbf{C}$ matrix is appended with the identity matrix and the $\mathbf{D}$ matrix is appended with zeros to match the number of rows of the $\mathbf{C}$ matrix. In this way, the excitation of internal modes and its derivatives are added to the output vector y and can be used for verification.

5.2.3. Mass division

In the Simscape model, the boundary frames are not rigidly connected to the rest of the body and do not have any inertia, as is shown in Figure 3. In Simulink, this results in unstable system behavior because a force is put on a massless object resulting in high accelerations. To solve this problem, some inertia must be added to the external nodes. Two methods are compared:

First, as common in classical flexible multibody dynamics, only mass is put on the body fixed frame. To prevent unstable behavior, in this application it means the mass on the external frames should be as low as possible without causing high simulation times or instability. To do so, a small portion of the total inertia of the body is subtracted from the body fixed frame and added to each boundary frame. An assessment must be made to find the optimum portion of mass put on the boundary frames.

Second, the inertia of the body can be equally divided over each body such that each boundary frame reflects its adjoining part of the body. This process has many similarities with mass lumping when using a lumped system approach. There is, however, no consensus on how the total inertia of the system must be divided over the three frames [3, 5, 6]. An advantage is that the inertia properties of the deformed beam are more accurately reflected in the total multibody system [1].

Multiple methods of mass lumping have been compared. However, many either result in negative inertias which are not supported by Simscape or require shape functions of the body which, due to the complex geometry, are not available in this case. A commonly used method is the HRZ lumping method, because of its simplicity and relatively good accuracy [7]. Here, the translational mass on the main diagonal of the Craig-Bampton matrix is summed. Because cross terms are neglected, this sum does not correspond exactly to the real mass of the body. This difference in mass has been compensated for by multiplying the mass on each frame with the same constant in order to keep the total mass correct. The rotational inertias on the main diagonal are also multiplied by the same constant. All off-diagonal terms are set to zero. This method normally results in a good mass division, but rotational inertia is less accurate. Generally, this does not impose large errors because the rotational inertias only have a minor influence on the flexible behavior of a body [3].

Both methods are being compared to each other in subsection 7.2.

5.3. Simscape

The only change that needs to be made in Simscape is replacing the rigid body block with a flexible body block. This block builds upon the work done by MathWorks [1] to automatically import the right data setup in the previous step from the Matlab Workspace. What is new is that this block also stores its deformation data, consisting out of participation factors of each Craig-Bampton mode and each internal mode for every time step, in the Matlab Workspace so it can later be used for further processing.

For this study, the 'daessc' solver provided with Simscape Multibody is used in its default settings, as this proved to be a fast solver with accurate results. Optimizing the solver settings might improve simulation speed or accuracy and stability, but the default settings seem to work quick and reliable with this system.

5.4. Setup APDL script

This model uses modal superposition to determine the stresses inside a body. This process will be discussed more extensively in subsection 5.6. The problem is that for each Craig-Bampton and internal mode a static structural analysis must be performed to determine the stresses occurring during each mode. As setting up a study manually for each mode is laborious, this Matlab script automates this process by setting up an Ansys APDL file containing the deformations for each boundary node during each mode. This file can simply be imported into Ansys APDL where all studies automatically run successively, and all data is stored in text files.

A possible disadvantage of this method is that, based on the FEM software used, this method is unable to work with internal modes. The culprit here is that some FEM programs do not determine stresses when performing a modal study, making modal superposition impossible. Switching to other FEM programs might offer a solution. In this research, it is assumed only the Craig-Bampton modes result in stresses; stresses due to internal modes are neglected. As the natural frequency of the internal modes is much higher than the occurring frequencies in the construction, this is not a large problem for this case. However, depending on the geometry of the body, this assumption might have a large effect on the results.

5.5. Ansys APDL

The APDL script set up at the previous section is imported and run in Ansys APDL. This will perform the static structural study for each Craig-Bampton mode successively for each boundary frame defined in subsection 5.1.

For example, the first mode consists of all frames being fixed at their original position, only with the first frame displaced 1 meter in the x direction. The deformations, normal stresses and shear stresses at each node are calculated and exported into a text file. Then, this process is repeated until each Craig-Bampton mode is calculated. The body fixed frame is fixed during each study, as the local frame used during the study in Simscape multibody is rigidly connected to this frame. A total of 6 studies for each boundary frame are performed. In case of the Afterburner, having 2 boundary frames, this thus results in 12 studies.

5.6. Combining data: Modal superposition

Commonly, the deformation data derived in the previous section is used to perform a transient structural analysis. This process will be explained in detail in section 6. Performing a transient analysis, however, means performing the time integration and solution convergence twice; once during the simulation in Simscape and once in the FEM package. Also, determining the stress for a complete mesh is computationally expensive. In Simscape, the possible deformation shapes are limited by the number of Craig-Bampton and internal modes considered. Having to solve for the full FEM mesh at each time step is therefore of no use, as the accuracy is determined by the number of deformation modes considered, not by the number of nodes in the FEM mesh.

A more effective way to simplify the stress computations is using modal superposition of the stresses. From the Simscape study, it is already known how much each mode is excited at each time step. From the previous subsection, the stresses occurring with each Craig-Bampton or internal mode are known. Now, multiplying the stresses per mode by its flexible coordinate η and adding the results of the different modes together results in a good approximation of the total stresses at each point in the mesh:

$$\boldsymbol{\sigma} = \sum_{i=1}^{n} \eta_i \boldsymbol{\sigma}_i \tag{9}$$

Here, $\boldsymbol{\sigma}$ being the tensor with normal and shear stresses in each direction for each point in the mesh, n the number of modes taken into account, η_i the participation factor of mode i and $\boldsymbol{\sigma}_i$ the normal and shear stresses at each node for mode i. This function can be repeated for each time step to obtain the stresses over time.

The modal superposition is executed within Matlab, as it makes it easy to use the data gathered previously from Simscape. To start, Matlab imports the displacement data from Simscape and the stress per Craig-Bampton-mode from the text files. For each timestep, the normal and shear stresses are multiplied by the flexible coordinate of each Craig-Bampton mode and added to the result of the other Craig-Bampton mode shapes, like Equation 9. This results in the normal and shear stresses and displacements in each local direction at each time step.

With this information, it is possible to retrieve the stresses and displacement of each node at each time step. This data can for example be used for constructive evaluation or fatigue studies. The Von Mises equivalent stress is also calculated from the individual stress directions. This is later used in this study for comparison with Ansys Workbench.

6. Validation with Ansys Workbench

Using modal superposition to describe the stresses and deformation of the body requires some assumptions to be made. The main assumption is that the deflection of the body can be fully described by a combination of its Craig-Bampton modes of the boundary frames. To validate whether these assumptions and simplifications are accurate, the results are compared to a second method for determining the stresses.

This method starts with the same deformation data of the boundary frames as used with the method explained in the previous section. The deformation of the boundary frames with respect to the body fixed frame is imposed on the same boundary frames in Ansys Workbench. The body fixed frame is kept stationary, as Ansys Workbench does not support rigid body movements. It is assumed that effects of the rigid body motion, like the shifting direction of gravity, is already present in the deformation data of the boundary frames and should therefore not be compensated for.

Analytically, the main difference of this method compared to Simscape, is that the position and stresses of each node are not prescribed based on the Craig-Bampton modes as the model is solved for a full mesh. This means that the deflection of each node within the body is not limited to the deflection described by a limited combination of modes but is free to take on any position. From this view, this method offers a more accurate representation of the real world. On the contrary, this method is computationally more expensive than modal superposition.

To import the data from Simscape into Ansys, it should be converted to a data structure compatible with Ansys Workbench. For this reason, two important assumptions must be made:

First, Ansys Workbench is unable to work with large data structures. As Simscape Multibody uses small time steps to come to an accurate and stable result, the amount of time steps is rather large. To decrease the number of steps, the deformation data is resampled using a lower sample frequency. For this, first a fast Fourier transform of the deformation has been made to analyze the frequency content of the deformation data. From this, it can be seen that there are barely any vibrations present in the Afterburner above 10 Hz. The data should therefore be resampled at at least the Nyquist frequency of 20 Hz. To account for some margin of error, the data is resampled at 40 Hz.

Secondly, the rotation convention between Simscape Multibody and Ansys Workbench is different. Simscape Multibody can export rotations as either axis-angle, quaternions or 3d rotation matrices. It is also possible to get the rotation of the boundary frames by requesting the participation factors of each rotational Craig-Bampton mode. The rotation order of this method is the extrinsic x-y-z Tait-Bryan angles. Ansys Workbench on the other hand uses the intrinsic Z-X-Y Euler Angle sequence. Transforming between those conventions can be complex as the solution is not always unique.

Generally, however, rotations due to deformation are small. In the case of the Afterburner, the rotation in any direction is at most 0.013 radians. Therefore, it is assumed rotations are linear, making rotation order unimportant. This removes the need for converting one convention into the other as the rotational data can directly be taken from the Craig-Bampton participation factors and applied in Ansys Workbench.

To import, the translation and rotation data is converted into a xml-file used by Ansys Workbench to import displacement data for a transient analysis. This results in 6 separate files per boundary frame, which can be assigned to each degree of freedom of each boundary frame. After this, the simulation can be run. The resulting data will be compared in the next section.

7. Results and discussion

This section is divided into three subsections. First, the Afterburner will be simulated using the current state of the art as discussed in section 2. This method uses the rigid multibody Simscape model to determine the constraint forces and moments on the main beam. Static load cases are determined from these constraint forces and moments. The results between the two sets of load cases, with the forces either applied at the top or bottom boundary, will be compared and discussed.

Now a baseline is set using the rigid multibody model, the results can be compared to the flexible multibody model of the Afterburner. First, however, some model settings and the model accuracy will be tested on the beam when rigidly fixed at its top. Using this setup, the influence of the mass division and internal modes on the accuracy and simulation time is compared to a transient structural analysis. The optimal settings for simulating the Afterburner will be derived using the results.

In the final subsection, the flexible response of the beam of the Afterburner will be simulated following the method described in section 5, using the optimal settings and the flexible multibody model. The constraint forces on the beam will be compared for the rigid and flexible multibody model. After this, transient studies using Ansys and modal superposition are compared.

During this section, all directions mentioned are in the local body fixed frame of the beam, see Figure 1. In graphs, the x-direction is marked red, the y-direction is marked green and the z-direction is marked blue.

7.1. Industrial state of the art: Load cases

The constraint forces and moments on the beam are recorded for both the top and bottom boundary frame using a rigid multibody simulation. For one period of the sweeping motion, the results of the rigid multibody study can be found in Figure 9 and Figure 10. Note how the moments in transverse direction due to the gyroscopic effects of the rotating seats are larger than the actuating torque.

As noted in section 2, the current state of the art is to use the maximum and minimum values of these forces over time to set up load cases. In this case there are 6 loadings (3 forces and 3 moments) making up for 64 load cases. To evaluate these load cases, the beam is rigidly fixed to one boundary frame whilst a load case is put on the free boundary frame. This is done for both the top and the bottom boundary. For each node, the maximum occurring Von Mises stress during all 64 load cases is recorded. These results are displayed in Figure 11 and Figure 12.

Examining these figures, it is clear that the location of where the beam is clamped has a great effect on the results. The maximum stress is different for both cases, with the load case applied at the bottom boundary frame resulting in lower stresses. Furthermore, the location where the maximum stress occurs is different for both cases, as it is located near the clamped end as expected for a cantilever beam. All in all, the results are significantly different when clamping the top frame compared to clamping the bottom frame, where in practice, both studies should yield the same results because the same rigid multibody problem is solved. Clearly, this method, which is the current state of the art, is not suitable to determine the stresses in this case where the beam is not rigidly connected at either end.

7.2. Model optimization

Before applying the novel method on the Afterburner, some values can be optimized to improve accuracy and decrease computational costs. These studies also give an indication about the accuracy of the flexible model used in Simscape. In this paper, attention will be paid to two important factors: the weight distribution and the internal modes.

For the optimization studies, the beam is rigidly connected at its top side. Two studies have been executed: one static and one dynamic. For the static study, a constant loading of $-100\,\mathrm{kN}$ is put in the x direction. The displacement of the end in the x direction is compared between the results of the modal order reduced model of

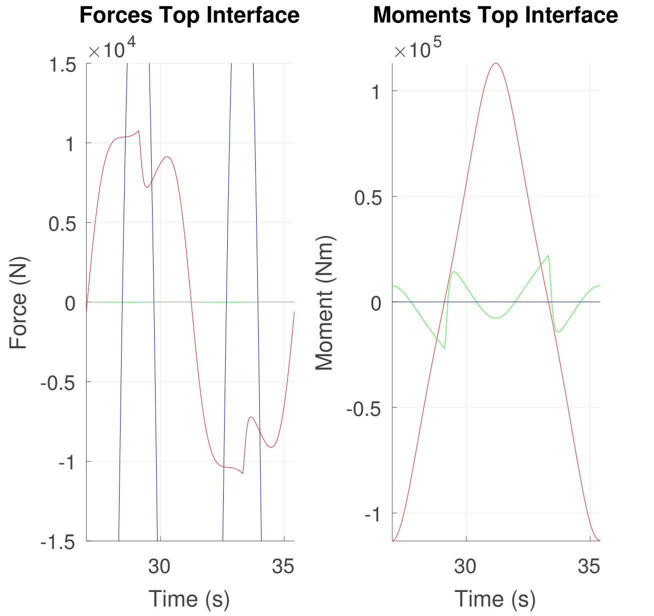

Figure 9: Constraint forces and moments on the top hinge during one period for the rigid model. The x direction is red, y direction green and the z direction blue

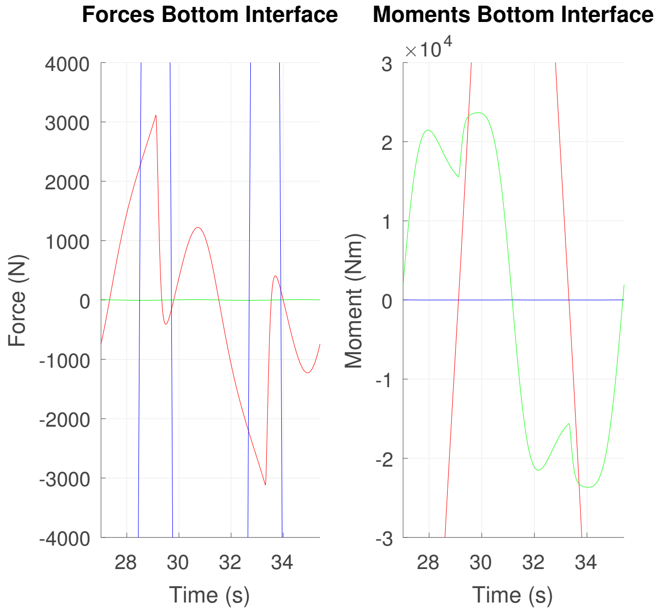

Figure 10: Constraint forces and moments on the bottom hinge during one period for the rigid model. The x direction is red, y direction green and the z direction blue

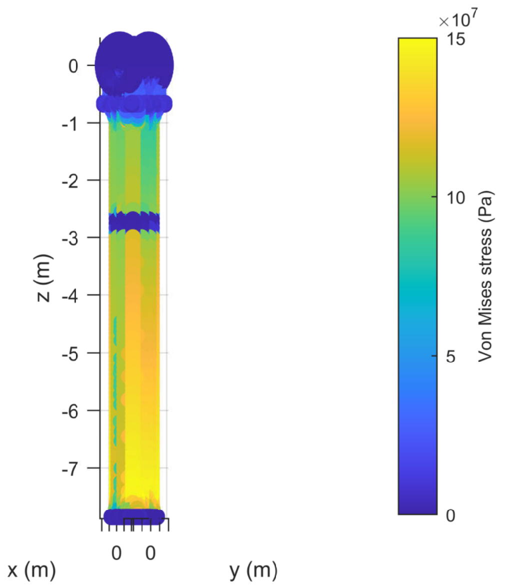

Figure 11: Maximum stress per node during all 64 load cases applied at the top frame. Maximum equivalent stress is 1.53×10^8 Pa

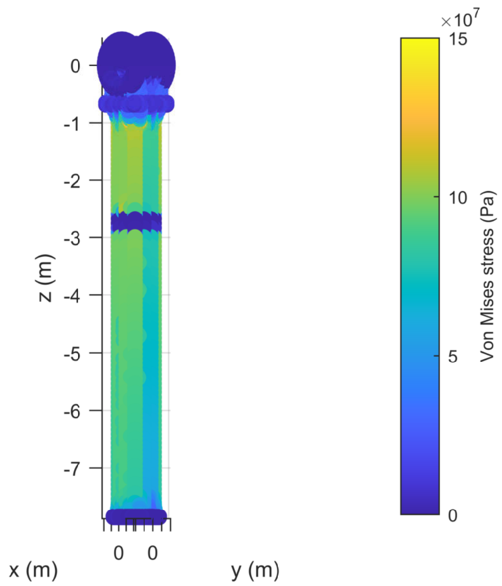

Figure 12: Maximum stress per node during all 64 load cases applied at the bottom frame. Maximum equivalent stress is 1.28×10^8 Pa

Table 1: Effect of internal modes on the frequency response

Nr of internal modes	Frequency difference (%)	Static difference (%)	Simulation time Simscape (s)
0	16.77	0.27	2.2
2	16.77	0.27	2.2
4	16.73	0.27	3.0
40	16.73	0.27	10.0

Table 2: Effect of mass division on the frequency response

Mass fraction on boundary frames (%)	Frequency difference (%)	Simulation time Simscape (s)
HRZ lumping	16.77	2.2
1	2.72	2.0
0.1	4.33	2.0
0.01	4.46	2.0
0.001	4.50	2.2
0.000001	4.50	2.6

Simscape and a fully meshed model in Ansys Workbench. For the dynamic study, the main vibration frequency in this direction is compared by giving the beam a short impulse on the free end and letting it vibrate freely. Again, the Simscape model is compared to the Ansys Workbench model.

To start, the influence of internal modes is evaluated. For this study, HRZ-lumping is used for the mass division. The results for different amounts of internal modes can be found in Table 1. The frequency difference is defined as $|\frac{\omega_{\mathrm{wb}}-\omega_{\mathrm{sc}}}{\omega_{\mathrm{wb}}}|$, with ω_{wb} the lowest vibration mode in Ansys Workbench and ω_{sc} the lowest vibration mode in Simscape Multibody.

It is striking that increasing the internal modes barely has any effect on the frequency response and static displacement. For the static displacement, this behavior is expected, as internal modes do not influence the static displacement of the boundary frames. A possible explanation for the dynamic behavior can be found in [3], which states that internal modes with frequencies more than 4 times higher than the main excitation frequency do not have a significant influence on the dynamic behavior of the system. Looking at Figure 6, the lowest internal mode has a frequency of 72.24 Hz, whereas the main frequency is more than 4 times lower with 8.80 Hz, fulfilling this rule of thumb. Increasing the number of internal modes does increase the simulation time of the Simscape Model, although many internal modes have to be considered for the simulation time to increase significantly.

Secondly, the two methods to divide mass over the frames discussed section 5.2.3 are compared with the results of Ansys Workbench. The results can be seen in Table 2. The mass division has no influence on the static displacement and is therefore not shown in the table. Clearly, putting the most mass on the mid frame results in more accurate dynamic results than HRZ lumping. For this case, the results seem to stabilize at a mass fraction of 0.01 %. Above this percentage the results barely change, but simulation time increases. Therefore, a mass fraction of 0.01 % will be used for the final simulation.

In the complete model of the Afterburner, the seats are fixed to the now free end of the beam. As the mass of the seats is substantially higher than the mass of the beam, the influence of mass division within the beam becomes smaller. With this mass added to the end of the beam and using HRZ lumping, the error in the vibrational frequency is only 1.55 % whereas placing most mass on the mid frame, the error decreases to 0.02 %. Still, using this last simulation method results in the best distinction between accuracy and simulation time.

7.3. New method: Transient and modal analysis

The methods outlined in section 5 and section 6 should give better insights in the stresses occurring in the beam over time. First, comparing the constraint forces and moments between the rigid and flexible multibody model, Figure 13 and Figure 14, it is clear some vibrations occur within the flexible model. However, as the system is quite stiff, the differences are small compared to the rigid study. Therefore, setting up load cases using the data of the flexible study instead of the rigid study will result in similar inaccurate results and does not provide additional benefits.

The deflections of the boundary frames of the beam following from the flexible multibody model can be seen in Figure 15 and Figure 16. This data is used for both the transient analysis as the modal analysis.

To compare both the results from modal superposition as well as the transient study in Ansys, for both studies the equivalent von Mises stress at the node which undergoes the highest stresses during the simulation is compared. This is the same node for both studies. In Figure 17 and Figure 18, the stresses calculated using both methods are plotted. Comparing the stresses, the resemblance is striking. Only minor differences between the two methods are visible. The absolute mean difference between the two methods is 0.11 %. Although the differences are small, the modal superposition only takes 1/52 of the computational time and the results file takes only 1/21 of the storage space, making it more attractive from a computational standpoint.

For this case, the simulation runs faster than real time. This enables many possibilities in real time applications and real time control problems. Setting up such a control loop is also quite straightforward, as Simscape Multibody works seamlessly with Simulink, offering many tools and supports multiple platforms to work with.

Comparing the maximum stresses during the transient analysis (95.41 MPa) with the maximum stresses during

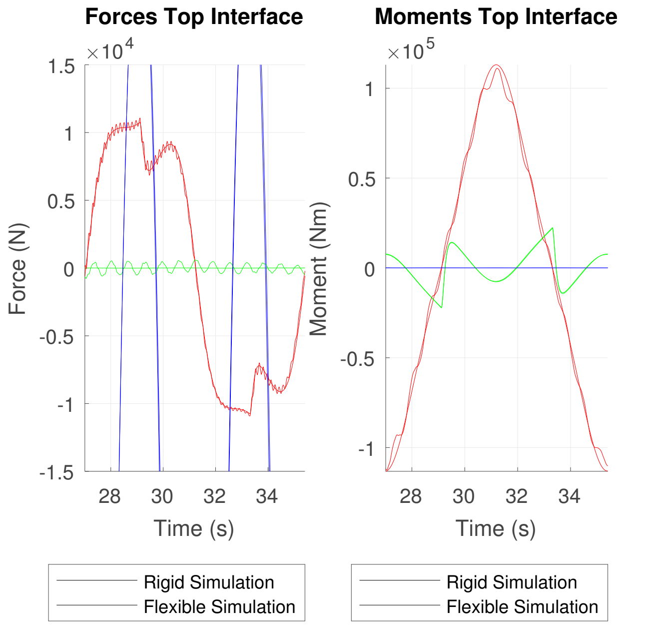

Figure 13: Constraint forces and moments on the top hinge during one period for the rigid and flexible model. The x direction is red, y direction green and the z direction blue

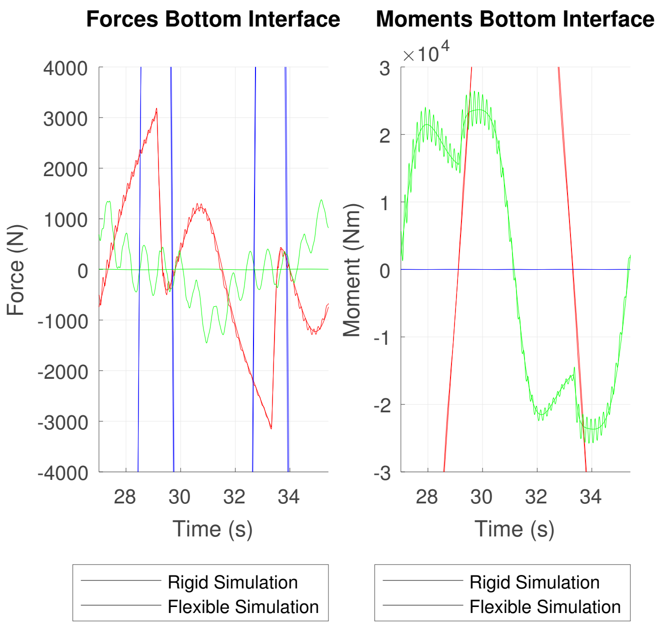

Figure 14: Constraint forces and moments on the bottom hinge during one period for the rigid and flexible model. The x direction is red, y direction green and the z direction blue

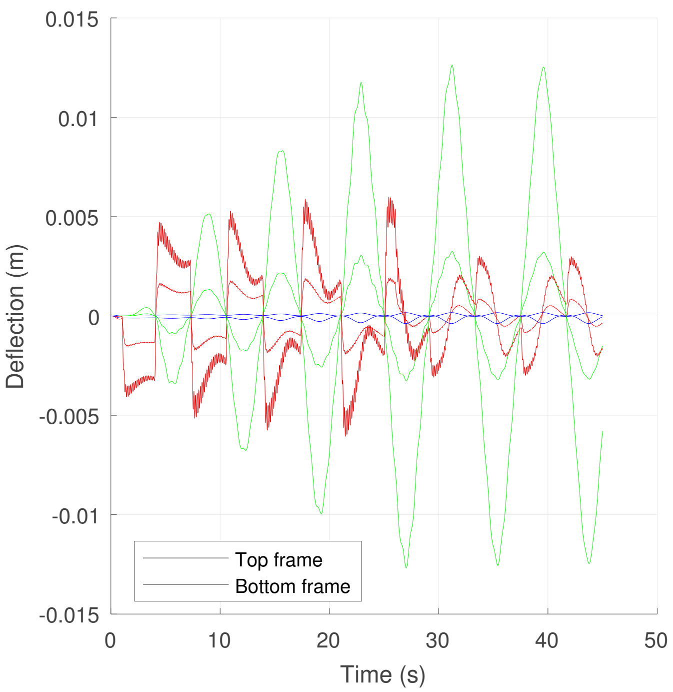

Figure 15: Linear deflection of the top and bottom boundary frame with respect to the body fixed frame

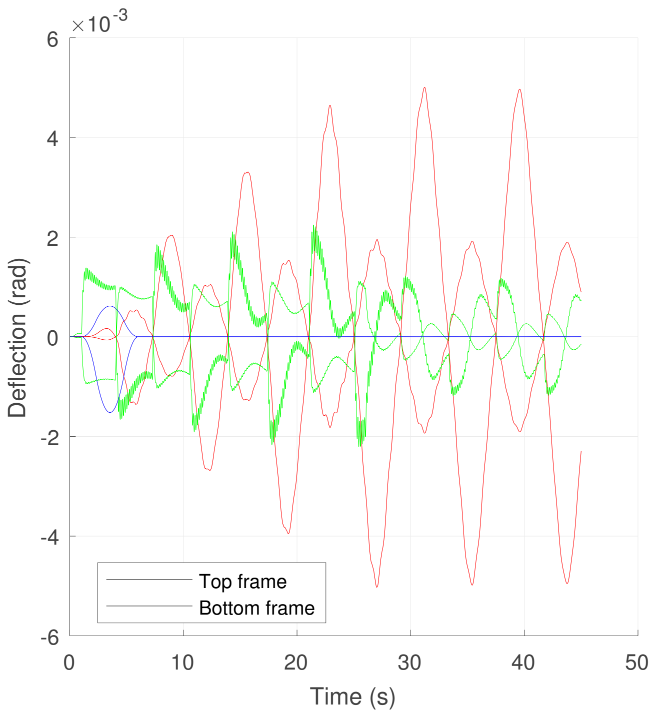

Figure 16: Rotational deflection of the top and bottom boundary frame with respect to the body fixed frame

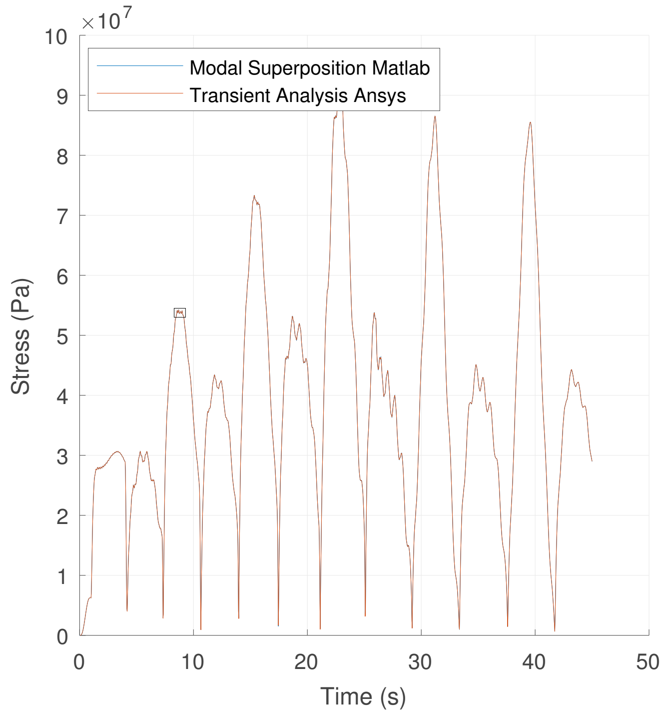

Figure 17: Comparison of the transient and modal analysis

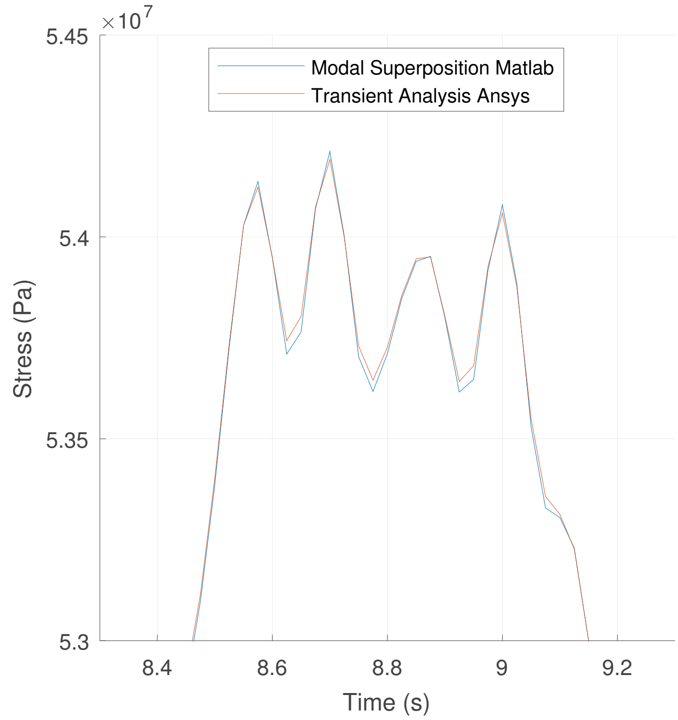

Figure 18: Zoomed in comparison of the transient and modal analysis

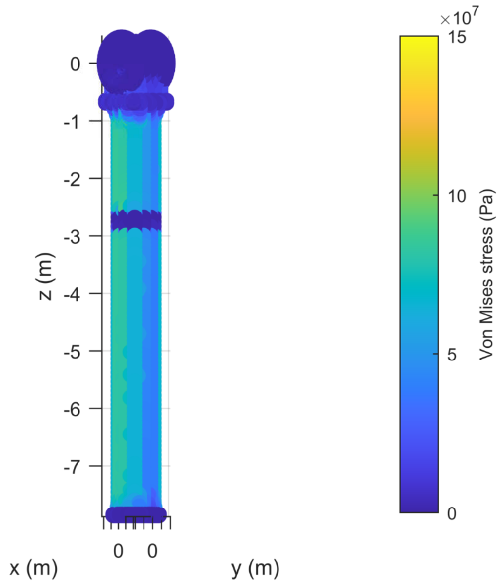

Figure 19: Maximum stress per node during the transient analysis using modal superposition. Maximum equivalent stress is 0.95×10^{8} Pa

the quasi-static load cases (153 MPa and 129 MPa), it is clear that quasi static load cases overestimates the stresses significantly. This effect can clearly be seen by comparing Figure 19 with Figure 11 and Figure 12. Using a transient analysis instead of the commonly used quasi-static method, this might lead to reducing the weight of a structure, which is very beneficial for a movable fairground attraction.

8. Conclusions

This paper has successfully developed a novel method of determining transient stresses inside a multibody structure. This is done by combining existing methods like modal superposition and programs like Matlab, Simscape Multibody and Ansys Workbench. It is shown that the current state of the art to determine stresses within a body, using quasi static load cases, often overestimates the stresses greatly. Also, the location of the maximum stresses is not always realistic. The method discussed in this paper helps solving these problems.

Comparing the new method with a full mesh transient study, the stresses are in great resemblance. However, after some manual setup, the new method is computationally much lighter without losing accuracy. This enables real-time simulations of complex structures and the possibility to include flexible behavior of bodies in control problems.

References

[1] S. Miller, T. Soares, Y. V. Weddingen, J. Wendlandt, Modeling flexible bodies with simscape multibody software, Technical paper, MathWorks (2017).

[2] R. R. Craig, M. C. C. Bampton, Coupling of substructures for dynamic analyses., AIAA Journal 6 (7) (1968) 1313–1319.

[3] R. D. Cook, D. S. Malkus, M. E. Plesha, R. J. Witt, Concepts and Applications of Finite Element Analysis, 4th Edition, John Wiley & Sons. Inc., 2001.

[4] A. Cardona, Superelements modelling in flexible multibody dynamics, Multibody System Dynamic 4 (2000) 245–266.

[5] C. A. Felippa, Introduction to finite element methods, Boulder, Colorado, USA (2001).

[6] K. H. Yang, Chapter 8 - modal and transient dynamic analysis, in: K. H. Yang (Ed.), Basic Finite Element Method as Applied to Injury Biomechanics, Academic Press, 2018, pp. 309 – 382.

[7] E. Hinton, T. Rock, O. C. Zienkiewicz, A note on mass lumping and related processes in the finite element method, Earthquake Engineering & Structural Dynamics 4 (3) (1976) 245–249.