Implementation and Experimental Validation of a Dynamic Model of a 10 MW Nacelle Test Bench Load Application System

Muhammad Omer Siddiqui, Norbert Eich, Karin Eustorgi, Paul Feja
Fraunhofer Institute for Wind Energy Systems IWES, Am Luneort 100, 27572 Bremerhaven, Germany

muhammad.omer.siddiqui@iwes.fraunhofer.de

Abstract. The 10 MW Dynamic Nacelle Testing Laboratory of Fraunhofer IWES provides a controlled environment for performing electrical and mechanical tests on a wind turbine nacelle. Apart from physical testing, system level simulations are another paradigm in the framework of nacelle testing. In this contribution, the development of a dynamic model of the load application system for the 10 MW nacelle test bench is presented. The test bench load application system controls are integrated in the model via co-simulation in Simulink. The model is evaluated using experimental results. By utilizing the modal strain recovery method, a direct comparison of the strain results of the model with the experimental results is achieved. Moreover, it is shown that the actual applied loads on the device under test can be estimated by analysing the strain readings. The developed model provides a platform for developing a high fidelity virtual nacelle test bench.

1. Introduction
The necessity of developing reliable wind turbine systems has increased the focus on full scale testing and experimental validation of wind turbine drivetrains. Nacelle test rigs are required that provide a full scale nacelle testing platform with realistic and reproducible test scenarios. One such test bench is the 10 MW Dynamic Nacelle Testing Laboratory (DyNaLab). Located at Fraunhofer IWES, Germany, this test bench provides a controlled environment for performing electrical and mechanical tests for a wind turbine nacelle of up to 10 MW power [1]. The test bench consists of two powerful drive motors for rotation and torque application. A Stewart platform (hexapod) unit is coupled to the drivetrain for the application of non-torque loads. Together, this load application system (LAS) can emulate various wind loading scenarios on the nacelle and ensures that realistic loads are applied during tests.

Parallel to the physical testing, system level simulations are another paradigm in the framework of nacelle testing. Deeper insights in component as well as system level dynamics can be obtained by the use of high fidelity simulation models. Such simulation platforms also provide the advantageous possibility of investigating the turbine behavior for the planned loading scenarios and analysing controller performance prior to the actual commissioning. This facilitates the identification of desirable test scenarios and better planning for commissioning. Moreover, a validated simulation model may also serve as a valuable reference for the actual tests.

The developed virtual test bench models at Center for Wind Power Drives (CWD) [2] and Clemson University [3] have demonstrated these advantages. However, each of these test benches are of unique
design, size and operational capacity. Therefore, the findings from these virtual models depend highly on the design configuration of their respective test benches and might not be valid for other types of systems [4]. A test bench such as the DyNaLab with a twin-motor direct drivetrain system and a hexapod load application unit will have dynamic characteristics that are unique to this configuration. Therefore, the development of a dynamic simulation model of the DyNaLab test bench is required.

This contribution presents the development of a dynamic load application system model for the DyNaLab. The model is developed in a multibody simulation (MBS) environment and integrates the test bench LAS controls via co-simulation in Simulink. This allows implementing the loads on the device under test (DUT) in the same way as in the actual test bench. The modelling methodology for the test bench LAS and the DUT is described in section 2. A strain based approach for the validation of the dynamic LAS model is presented. The method to extract the strain results from the MBS model is discussed in detail in section 3. Validation of the model is achieved using experimental results from the hexapod load validation project (LastVal). In this project, instead of a nacelle, a reaction structure is installed as a DUT containing several load cells, strain gauges and acceleration sensors. Using this setup, the hexapod loads are measured and the system response is analyzed for several types of loading scenarios. The strain results from the MBS model of the test bench are compared with the experimental results for the static and dynamic load cases. Section 4 presents the comparison of results, discussion on the model performance and a methodology for performing strain based load calculation. The paper then closes with a conclusion in section 5.

2. Modelling
The DyNaLab drivetrain contains several mechanical components. Combined with the hexapod unit with six hydraulic actuators, the overall LAS is a complex mechanical system. Creating a dynamic model for such a large system using finite element (FE) modelling will result in a computationally expensive model. The requirement of performing time domain dynamic simulations with controls integration that require a very small time step makes the FE approach infeasible. Due to these reasons, a multibody simulation approach presents a reasonable solution. Therefore, the dynamic model of the LAS is developed using the MBS software Adams.

![Figure 1. Multibody simulation model of the 10 MW nacelle test bench.](image)

The DUT structure, flange adaptors, coupling and motor rotor are modelled as flexible bodies. These bodies are created by modal reduction of their respective FE models using the component mode synthesis (CMS) method [5]. This involves the generation of a modal reduction matrix \([\Phi]\) consisting...
of fixed normal modes and constraint modes. The physical displacements of the flexible body are defined as a linear combination of these modes along with their modal coordinates.

\[ \ddot{x} = [\Phi] \cdot \ddot{q} \]  

where

- \( \ddot{x} \) is the vector of physical displacements
- \([\Phi]\) is the modal reduction matrix
- \(\ddot{q}\) is the vector of modal coordinates

Reduced mass and stiffness matrix are obtained by the matrix transformation using the modal reduction matrix according to Equation (2) and Equation (3) respectively.

\[ [\hat{M}] = [\Phi]^T [M] [\Phi] \quad (2) \]
\[ [\hat{K}] = [\Phi]^T [K] [\Phi] \quad (3) \]

\([M]\) and \([K]\) denote the mass and stiffness matrix, respectively, whereas \([\hat{M}]\) and \([\hat{K}]\) denote the corresponding reduced matrices. This matrix transformation leads to a reduced system of equations suitable for solving dynamic simulations with low computational effort.

All components in the model are connected with each other and the ground using constraints according to their configuration in the actual system. Figure 2 depicts this connectivity in a simplified manner.

**Figure 2.** Model topology of the MBS model.

2.1. Hexapod Load Application System

To evaluate the dynamic behavior of the LAS, an emulation of the LAS control scheme is implemented and coupled with the dynamic LAS model. To control the movement of the hexapod, the system’s inverse kinematics need to be solved iteratively to find the hexapod position for given individual cylinder displacements. For pure load application with negligible hexapod movements, the mapping from LAS cylinder forces to the actual loads at the test rig’s interface flange is assumed constant. This mapping takes place by utilizing a 6x6 Jacobian matrix that can be deduced from the system’s geometry and current position. The Jacobian enables to calculate the desired force set points at the six individual LAS cylinders for a given target load situation at the load application point according to Equation (4).
\[
F_{cyl} = [J]^{T} \cdot F_{LAP}
\]  \hspace{1cm} (4)

where

- \( F_{cyl} \) represents the six cylinder forces
- \( F_{LAP} \) represents the target load at the load application point
- \([J]\) represents the Jacobian matrix

For the simulations presented in this paper, the LAS control calculates the cylinder forces according to this scheme by establishing a co-simulation interface with Simulink.

### 2.2. Drivetrain System

The DyNaLab drivetrain system comprises of two synchronous drive motors arranged in tandem configuration by means of a stiff connection. With a nominal operational power of 10 MW, the drive motors transfer torque up to 8.6 MNm at 11 rpm speed to the DUT. A coupling connects the drive shaft and the DUT shaft, compensating any misalignments and ensuring that only torsional loads are transferred through the drivetrain. In the MBS model, the drive rotor and coupling are modelled as flexible bodies with joints at the connection interfaces. The coupling arms are modelled as rigid links. The coupling is connected to the hexapod via a moment bearing that is modelled by a bushing element with stiffness and damping properties provided by the manufacturer.

### 2.3. DUT Model

The DUT is a steel reaction structure that is installed with several sensors for measuring forces, accelerations and strains resulting from the applied loads. The sensor locations have been determined by analyzing the deformation behavior of the DUT for different loadings via FE simulation. For the validation of the DUT model, an experimental model analysis (EMA) is performed using a modal shaker device to identify the eigen-frequencies of the DUT. A simulation using the FE model is performed for the same DUT configuration and the eigen-mode shapes and frequencies are determined. The results are compared with the experimental data for validation of the dynamic behavior (see Table 1).

| Mode | Eigen-frequency [Hz] | Simulation | Experiment |
|------|----------------------|------------|------------|
| 1    | 19.0                 | Not identified |
| 2    | 20.8                 | 19.9       |
| 3    | 31.5                 | 32.1       |
| 4    | 46.8                 | 47.8       |
| 5    | 50.2                 | Not identified |
| 6    | 63.6                 | 62.9       |
| 7    | 64.7                 | Not identified |
| 8    | 72.0                 | 73.1       |

The calculations from the FE model show good agreement with the EMA results for most of the eigen-frequencies. However, in the experiment, not all of the calculated eigen-frequencies from simulation were observed. The minimum excitable frequency by the shaker device was 20 Hz. This might be the reason why the 1\textsuperscript{st} calculated mode could not be excited in EMA. The 5\textsuperscript{th} calculated mode has a very small amplitude in the simulation due to which it could not be identified in EMA. The 6\textsuperscript{th} and 7\textsuperscript{th} calculated modes have frequencies very close to one another. This could be the reason why the 7\textsuperscript{th} mode was not identified in EMA.
The eigen-modes from the EMA are displayed on a simplified wire frame model of the structure. The mode shapes have been scaled by a factor of 100 for better visualization. Figure 3 shows a comparison between FE model and EMA for eigen-modes at 32 Hz, 47 Hz and 62 Hz.

![Mode Shapes Comparison](image)

**Figure 3.** DUT mode shape comparison of the FE model (left) and EMA (right).

A flexible multibody model of the DUT is generated by modal reduction of the FE model using the CMS method. The bolt connection interfaces are assumed to be perfectly bonded. The DUT and the steel platform are considered as a single flexible body with master nodes constrained to the boundary surfaces.

3. Modal Strain

The reduced flexible model of the DUT also offers the feature of extracting strain results for dynamic simulations using the modal recovery method [6]. This method enables the calculation of the strain distribution of the flexible body with very fast computational time compared to the full dynamic analysis in a FE program. The method requires the calculation of a modal strain matrix \([\Phi_\varepsilon]\) during the CMS reduction phase of the flexible body according to Equation (5). The modal strain matrix relates the strain component associated with each of the mode shapes of the flexible body.

\[
[\Phi_\varepsilon] = [B][\Phi]
\]

where

- \([\Phi_\varepsilon]\) is the modal strain matrix
- \([B]\) is the function matrix of FE geometry relating strains to displacements
- \([\Phi]\) is the modal reduction matrix

For the dynamic simulation, strain results are obtained by combining the modal strain matrix and the modal coordinates in the post processing phase according to Equation (6).

\[
\vec{\varepsilon} = [\Phi_\varepsilon] \cdot \vec{q}
\]

where

- \(\vec{\varepsilon}\) is the strain vector for the flexible body

Since the finite element method is an approximation method involving domain discretization using elements of finite length, a discretization error exists that depends on the mesh size [7]. For the displacement calculation, the convergence rate of the discretization error is faster than the convergence rates for strain calculation [8]. Since the flexible body dynamics depend on the ability of its mesh to correctly capture the displacements, a mesh suitable for dynamics of the flexible body might not be sufficient for calculating strains using the modal strain recovery. Therefore, mesh refinements were made for capturing both the dynamic effects and strains. For validation of the flexible DUT model, strains calculated using the flexible model and the high fidelity FE model are compared for the case of a stepped load (see Figure 4).
Good agreement is observed for the deformation behavior of the reduced flexible model against the high fidelity FE model of the DUT for load cases that lead to smaller displacements. For higher loads in horizontal and vertical directions, deviations exist between the reduced flexible body and the high fidelity FE model. The high fidelity FE model allows for non-linear deformation of the structure, whereas the deformations in the reduced flexible are linear. This could be a reason for strain deviations at higher loads in the horizontal and vertical direction.

4. Results and Discussion

In the LastVal project, the validation tests for the hexapod unit were performed for the non-torque loading scenarios. The results from these tests were utilized for comparison with the simulation results from the MBS model. Forces at the load application interface between the DUT and the hexapod unit were determined during experiments using load cells. For every load case, the input loads to the six hydraulic actuators in the model are given using these experimentally determined loads at the load application interface. This ensures that the result comparison between simulation and experiments refer to the same loading conditions on the DUT. Experimental readings from the strain gauges are compared for one of the strain gauge locations on the DUT with the simulation results of the dynamic MBS model for various load cases. In the following sections, the results for static and dynamic load cases for the three loading directions (axial, horizontal and vertical) are presented.

4.1. Static Load Cases

Static loads are applied on the DUT in the form of steps in the axial, horizontal and vertical directions. Figure 5 shows the comparison of the strain between measured values and simulation results for the same location on the DUT for each step load case. To better visualize the differences in strain amplitudes, the initial strain values of simulation and experiment when no loads are applied is offset to zero.

The results for axial loading show good agreement of the simulated and the experimentally obtained values. However, the strains show deviation at higher loads in the horizontal and vertical direction.
4.2. Dynamic Load Cases

Sinusoidal loads have been applied on the DUT in the axial, horizontal and vertical directions with increasing frequencies. Figure 6, Figure 7 and Figure 8 show the strain comparison between the experimentally measured values and simulation results at different frequencies for each loading direction.

![Figure 6. Strain comparison for dynamic loads in axial direction.](image1)

![Figure 7. Strain comparison for dynamic loads in horizontal direction.](image2)

![Figure 8. Strain comparison for dynamic loads in vertical direction.](image3)

For each loading direction, the load amplitude was kept constant while changing the load frequencies. For better visualization of the differences in strain amplitudes, the initial value of strain from simulation and experiment when no load is applied is offset to zero. The simulation results for axial and horizontal loading show good agreement with the experimental values. However, the strain amplitudes from simulations for dynamic loads in vertical direction are higher than the experimental results.

In the actual hexapod system of the DyNaLab, the weight of the hexapod structure is partly compensated by two additional hydraulic cylinders. In the developed MBS model, the weight of the hexapod unit is fully supported by the DUT and not compensated by the hydraulic cylinders. This could be a reason for deviations in strain readings for loads in the vertical direction.

The MBS model assumes perfectly bonded contact for bolted connections. The translational movement of the hydraulic cylinder is modelled by a perfect translational joint, whereas the spherical bearing connections are modelled as perfect spherical joint. In the actual physical systems, friction exists in bolted connections, hydraulic systems, and bearings. The friction existing in these
connections might have an influence for different loading directions and their absence in the model could also be a factor for the deviations in simulation results. Therefore, the inclusion of frictional effects are needed to be analyzed in detail and will be part of future work.

It is also important to mention that the reduced flexible model represents a linearly deforming structure. In case the actual structure is deforming non-linearly, the model results will start deviating from the experimental results.

4.3. Strain Based Load Calculation
The applied loads on the DUT can be estimated by measuring strain at the adaptor flange connecting the DUT with the hexapod unit (see Figure 9). The adaptor flange offers a simple geometry to calculate forces via strain readings on the flange surface. For an axial load case, normal strain can be measured on the adaptor flange in axial direction from the strain gauge. The axial force \( F_a \) can then be calculated according to Equation (7).

\[
F_a = \varepsilon EA
\]

where

- \( E \) is the elastic modulus of the adaptor material
- \( A \) is the cross-sectional area of the adaptor
- \( \varepsilon \) is the normal strain in axial direction

To evaluate this approach for future application in nacelle testing, the developed MBS model is used and the force calculated via modal strain from simulation is compared with the theoretical value at the load application point. The results for the static and dynamic axial loads are shown in Figure 10.

\[\text{Figure 9. Strain gauge locations on the adaptor flange.}\]

\[\text{Figure 10. Comparison between strain based calculated force and the theoretical force for static (left) and dynamic (right) load case.}\]
The simulation results show that the strain based method for load estimation matches well with the analytical values of loads on the DUT. Strain gauges are simple measurement devices that are easy to install, utilize and can offer a convenient approach for estimating the actual applied load by the LAS on the nacelle during testing. However, this method needs experimental validation which will be part of future work.

5. Conclusions
In this contribution, the development of a dynamic multibody simulation model of the LAS for the DyNaLab has been presented. The model integrates the LAS force generation scheme for the hydraulics via co-simulation with Simulink. A high fidelity FE model of the DUT is first validated via experimental modal analysis and then integrated in the MBS model via modal reduction method. A strain based approach has been utilized for comparison between simulation and experimental results. The strains calculated in MBS via modal strain recovery showed good agreement to the experimental results for the axial and horizontal load cases. Notable deviations were observed in strain results from simulations for loads in the vertical direction. Using the developed MBS model, a strain based methodology for estimating the actual applied loads by the LAS on the DUT has been presented. The method could provide a convenient approach for load estimation on the nacelle in future testing. The developed model provides a platform for further developing a high fidelity virtual nacelle test bench.

References
[1] M Pilas, M Riezu, Functional prototype testing of an 8MW wind turbine, German Wind Energy Conference DEWEK 2017, Bremen, 2017
[2] J Berroth, L Bi, D Matzke, A Werkmeister, R Schelenz and G Jacobs, Nacelle test benches for model validation, Simpack Wind and Drive Train Conference, 2015
[3] R Schkoda, Virtual testing of full scale wind turbine nacelles, Simpack News February 2016
[4] L Bi, Dynamic simulation of full-scale wind turbine nacelle system test benches, RWTH Aachen, Mainz G, 2015
[5] R J Craig, A review of time-domain and frequency-domain component mode synthesis methods, International Journal of Analytical and Experimental Modal Analysis, Vol.2, No.2, 1987
[6] MSC Software : Adams/Durability help – MSC Adams 2011
[7] J O Dow, A unified approach to the finite element method and error analysis procedures, Pages 387-404, Academic Press, 1999
[8] C Shah, Mesh discretization error and criteria for accuracy of finite element solutions, Ansys Users Conference 12, Pittsburgh, 2002