Hybrid testing procedure development

An experimental study a hybrid simulation of a steel truss element.

Authors: Ali Alhameedi
Supervisor LNU: Torbjörn Ekevid
Examinar, LNU: Björn Johannesson

Course Code: 4BY35E
Semester: Spring 2019, 15 credits

Linnaeus University, Faculty of Technology
Department of Building Technology/Mechanical Engineering
Abstract

Hybrid simulation is a numerical and experimental structural testing technique in which the critical structural members are tested experimentally, while the rest of the structure is modeled numerically. During hybrid testing, the numerical model is updated continuously based on the output of the experimental test. In this study, a quasi-static hybrid simulation was conducted on a steel truss structure. A single truss member was considered as the physical substructure, while the remaining members were modeled using Abaqus finite element software. The communication between the physical and numerical models was established using OpenFresco as a middleware. Using this setup, the structure was loaded until the physical substructure failed due to buckling. Finally, the results of the hybrid simulation were verified numerically and experimentally.

Keywords: Hybrid simulation test, substructures methods, quasi static test
Acknowledgement

Firstly, I would like to thank my advisors Dr. Osama Abdeljaber and Torbjörn Ekevid of the Faculty of technology/Building technology at Linnaeus University for their guidance and support. I would also like to thank the examiner of this thesis Prof Björn Johannsson for his valuable comments. My warm thanks to Berhil Enqvist and Jonas Klaeson for theirs help in executing the tests. Lastly, I should thank all those who contributed to achieve this academic work. Thank you so much.

Ali Alhameedi

Växjö 25th of June 2019
# Table of Content

**ABSTRACT** .................................................................................................................. II

**ACKNOWLEDGEMENT** ............................................................................................... III

**TABLE OF CONTENT** ................................................................................................ IV

## 1 INTRODUCTION .......................................................................................................... 1

1.1 BACKGROUND ............................................................................................................. 1
1.1 PROBLEM DESCRIPTION ......................................................................................... 2
1.2 AIM AND PURPOSE ..................................................................................................... 3
1.3 RELIABILITY, VALIDITY ............................................................................................ 3
1.4 AUTHOR CONTRIBUTION ......................................................................................... 3

## 2 LITERATURE REVIEW ................................................................................................. 4

## 3 THEORY ....................................................................................................................... 7

3.1 HYBRID TESTING ....................................................................................................... 7
3.2 TRUSS STRUCTURES ................................................................................................... 8
   3.2.1 Buckling failure in truss structures ..................................................................... 8
   3.2.2 Analysis of truss structures .............................................................................. 9

## 4 METHODS AND IMPLEMENTATION ........................................................................... 15

4.1 NUMERICAL MODELING ............................................................................................ 15
4.2 EXPERIMENTAL SETUP ............................................................................................ 16
4.3 COMMUNICATION BETWEEN THE NUMERICAL AND EXPERIMENTAL SUBSTRUCTURES .............................................................................................................. 17
4.4 OVERALL PROCEDURE ............................................................................................. 18

## 5 ANALYSIS AND RESULTS .......................................................................................... 20

5.1 EXPERIMENTAL TESTS ............................................................................................ 20
   5.1.1 Experimental test 1 (elastic mode) ..................................................................... 20
   5.1.2 Experimental test 2 (elastic and plastic) ............................................................ 20
5.2 NUMERICAL TEST ..................................................................................................... 21
5.3 HYBRID SIMULATIONS ............................................................................................. 22
   5.3.1 Hybrid simulation 1 .......................................................................................... 22
   5.3.2 Hybrid simulation 2 .......................................................................................... 23
   5.3.3 Hybrid simulation 3 .......................................................................................... 23

## 6 DISCUSSION .................................................................................................................. 25

6.1 DISCUSSION OF THE EXPERIMENTAL TESTS ....................................................... 25
6.2 DISCUSSION OF THE HYBRID SIMULATIONS ....................................................... 25
6.3 SOURCES OF ERROR .................................................................................................. 26

## 7 CONCLUSIONS ............................................................................................................. 27

**REFERENCES** .............................................................................................................. 28
List of figure

FIGURE 1. HYBRID SIMULATION PROCEDURE ................................................................. 2
FIGURE 2. DIVIDED REFERENCE STRUCTURE INTO NUMERICAL AND EXPERIMENTAL SECTIONS [8] ................................................................. 4
FIGURE 3. BUCKLING IN SLENDERS MEMBERS [14] ...................................................... 8
FIGURE 4. NODAL DISPLACEMENT IN TRUSS STRUCTURE [16] ........................................... 9
FIGURE 5. AXIAL LOAD IN BAR ELEMENT [15] ........................................................... 10
FIGURE 6. BAR ELEMENT IN GLOBAL COORDINATE SYSTEM [15] ................................. 11
FIGURE 7. BAR ELEMENT IN LOCAL COORDINATE SYSTEM [15] ................................. 11
FIGURE 8. THE DISPLACEMENTS IN GLOBAL COORDINATE SYSTEM .......................... 13
FIGURE 9. THE TRUSS STRUCTURE .............................................................................. 15
FIGURE 10. MTS QUASI STATIC HYBRID SIMULATION [20] ............................................. 16
FIGURE 11. THE EXPERIMENTAL SUBSTRUCTURE .................................................... 17
FIGURE 12. EXPERIMENTAL HYBRID SIMULATION SETUP ........................................... 17
FIGURE 13. HYBRID SUBSTRUCTURING METHOD ......................................................... 18
FIGURE 14. DEFORMED TEST ELEMENT ................................................................. 19
FIGURE 15. FLOWCHART OF THE HYBRID TEST ......................................................... 19
FIGURE 16. FORCE VS. DISPLACEMENT IN TEST 1 (ELASTIC MODE) ............................. 20
FIGURE 17. EXPERIMENTAL TEST 2 (ELASTIC AND PLASTIC) ..................................... 21
FIGURE 18. FE MODEL OF THE STEEL TRUSS ......................................................... 21
FIGURE 19. REACTION FORCE VS. THE APPLIED DISPLACEMENT AT NODE 1 CALCULATED USING ABAQUS .............................................................. 22
FIGURE 20. DISTRIBUTION OF VON MISES STRESSES CALCULATED USING ABAQUS ................................................................................................. 22
FIGURE 21. FORCE VS. DISPLACEMENT PLOT FOR HYBRID SIMULATION 1 .................. 23
FIGURE 22. FORCE VS. DISPLACEMENT PLOT FOR HYBRID SIMULATION 2 .................. 23
FIGURE 23. FORCE-DISPLACEMENT GRAPH FOR HYBRID SIMULATION 3 .................... 24
FIGURE 24. FORCE-DISPLACEMENT PLOTS FOR ALL THE HYBRID SIMULATION WITH PURE ABAQUS ANALYSIS ................................................... 26
LIST OF TABLES

Table 1. Steel element material properties and cross-section details .................................................. 15
1 Introduction

1.1 Background

Full-scale testing of large and complex structures can be complicated and very expensive since a large open space, special fixtures and a high capacity hydraulic system are needed [1][2]. Another possibility is to identify the most critical elements of the structure and perform physical testing of those elements in combination with a finite element analysis (FEA) of the remaining part of the structure [3]. This analysis method is known as hybrid testing or hybrid simulation. Nowadays, hybrid simulations are used in various fields like civil, automotive and aerospace engineering [4]. The basic principal in hybrid testing is to combine numerical simulation of a structure with experimental testing of a small part of the structure in an attempt to obtain equivalent results as a full-scale test of the entire structure.

Hybrid simulations have been performed by Japanese researchers as early as the 1960s [5]. The method was used to examine a system subjected to seismic loading in order to achieve more realistic simulations. Hybrid simulations were improved when computers became more available and during the 1970s. The method was further improved by Takanashi [5], who made it possible to analyze systems subjected to slowly applied loads, known as quasi-static loading. In Japan and United States, hybrid simulations continued to be enhanced from the 1970s to 1990s, while other countries started to use the method from mid 1990s [4]. During this time, some problems with experimental error propagation and integration algorithms were identified.

Currently, hybrid simulation is applied to study the behavior and performance of large buildings under seismic loading [3]. The method is also used to examine the behavior of structures undergoing large deformation. Hybrid simulation is mostly used for quasi-static tests, pseudo dynamic tests (PsD) and real-time hybrid tests (RTHT), which are all different types of dynamic tests ranging from very slow to very fast dynamic movement.

In this study, a hybrid test of a simple steel truss was conducted using the FE software Abaqus [21] together with an MTS biaxial test machine. The entire truss was modeled in Abaqus. A single steel bar element was defined as the experimental substructure, which was used as a physical test specimen as shown in Figure 1. The rest of the truss was defined as a numerical substructure. A communication interface was established between the FE-model and the test specimen so that they can exchange information about displacements and reaction forces. Next, the response of the truss under a quasi-static load was simulated. The computed displacements of the experimental substructure were transferred to the test specimen via hydraulic
actuators. After that, the reaction forces were measured and sent back to the FE-model to be used to update displacements of the entire truss. This process is then iterated for a certain number of time steps until the hybrid simulation is terminated. The accuracy, reliability and stability of the hybrid test was analyzed and optimized in an attempt to achieve realistic behavior of the truss compared with pure FE analysis.

![Hybrid simulation procedure](image)

Figure 1: Hybrid simulation procedure.

1.1 Problem Description

As mentioned earlier, physical tests and analysis of large structures can be complicated and costly, especially if several full-scale structures are going to be tested [6], [7]. Another method to analyze large structures is to make computer aided numerical models and use FE analysis, which often is reliable and frequently used [7]. Unfortunately, for some cases like inelastic, nonlinear and rate dependent behavior of structures it is difficult to make realistic and accurate FE-models. In order to work out these problems, hybrid simulation is often used nowadays, since experimental testing is the best method to obtain correct information about the structural behavior.

Although hybrid simulations have many advantages, there are some disadvantages that need to be addressed [4]. For example, more knowledge is required about experimental error propagation, integration algorithms and substructuring techniques. Experimental errors occur due to limitations of the actuators and incorrect setup of test specimens. Consequently, this reduces the accuracy of the hybrid test since errors may accumulate and lead to substantial error as the simulation progresses and incorrect displacement and force measurements are looped in the system to solve the equations of motion. Furthermore, the integration algorithms that are used to solve the differential equations need to be carefully selected and applied in order to obtain efficient and accurate calculations. Substructuring techniques, which
are user to divide a structure into numerical and physical models, need to be properly applied and evaluated to ensure efficient calculations.

1.2 Aim and purpose

This thesis aims at investigating the practicability of performing and developing a hybrid test procedure for a steel truss using the Abaqus and MTS810 biaxial test machine. The work presented in this study involves developing a communication interface between the test equipment and Abaqus using the middleware OpenFresco. The hybrid simulation is then optimized by running several experimental and numerical tests. Finally, to verify the accuracy of the hybrid simulations, the results are compared to pure FE analysis and pure experimental testing of the entire truss.

The purpose of this study is to obtain knowledge and experience for making accurate, reliable and stable hybrid testing of large and complex structures.

1.3 Reliability, validity

Reliability will be ensured by detailed description of how the hybrid simulation is going to be performed. In addition, reliable sensors and middleware (OpenFresco) were used to guarantee accurate measurements. Furthermore, a well-established FE software (Abaqus) was used to create the numerical model. The hybrid simulation analyses were verified against physical reference models and simulations of numerical reference models. Several experiments using the quasi-static testing method were executed to ensure the reliability and check the strength of steel truss material.

Validity was reached by carefully studying the relevant variables to ensure that no other variables can affect the results. This was conducted by comparing between the outputs of the hybrid simulation and those obtained by pure numerical and physical tests.

1.4 Author contribution

In this thesis, I have used the available resources to conduct my research. I utilized the laboratory equipment to accomplish this research work as authorized and advised. Using the knowledge from the available literature, I was able to set up the test environment with the assistance of my advisor and laboratory technicians. I have also created and validated an FE model of the structure using Abaqus.
2 Literature Review

Numerous studies in the literature have investigated different aspects of hybrid simulations including the following:

1. The substructuring method.

2. The type of structural responses to be communicated between the experimental and numerical models (static or dynamic).

3. The middleware that links the FE-substructure to the reference experimental test.

In this chapter, the previous studies on hybrid simulations are reviewed in terms of the above-mentioned aspects.

The most relevant work to the topic of this thesis is the study conducted by Høgh et al. [8] on quasi-static hybrid simulation of a single composite beam with multi-axis control. The beam was divided into two substructures: an FE-model and a physical test specimen with three degrees-of-freedom as shown in Figure 2. To verify the accuracy and validity of the hybrid simulation, the results were compared to a standard bending test of the full-scale beam. The FE model is created using ANSYS 15.0, while the communication between the model and the physical test specimen was established using LabVIEW software.

The results showed discrepancies in displacements between the FE model and the test specimen. This was corrected by Digital Image Correlation (DIC), which improved the accuracy considerably. The stiffness of the test specimen was inaccurate, however, the accuracy was improved by applying linear extrapolation to compensate for the communication delay. The experiment also revealed that when testing a single component, the test rig must be carefully set up so that the displacements can be applied correctly at the shared boundary.

Figure 2. Divided reference structure into Numerical and experimental sections [8].
In another relevant study, Lam et al. [9] performed an online hybrid test using a finite element program and an explicit integration scheme. The numerical substructure was analyzed using ABAQUS/Explicit. The system uses a separated-model framework, operator-splitting integration scheme and a socket mechanism for data exchange. This system was verified over a simple moment-resisting frame. The results demonstrated the ability of the system to simulate the seismic response of the structure.

D. McCrum et al. [10] conducted a hybrid test on a flexible substructure using conventional testing devices. The hybrid simulation was developed in a way that made it possible to divide a structure into several substructures and analyze them numerically and physically at different locations with various software and computers. It was shown that conventional loading devices such as hydraulic actuators are robust and easy to transport to the test locations. In addition, the loadings and the control modes of these actuators are easy to regulate. A software was developed to make it easy to coordinate the shared boundary displacements among the substructures. An interface was used to communicate the response between the substructures.

B. Wu et al. [11] conducted a hybrid simulation of a full-scale steel frame structure in an attempt to improve a nonlinear model of the structure. The parameters of the numerical model were updated and corrected online by using data from an experimental substructure. To minimize errors, a so-called sectional constitutive model was employed, where axial-force and bending-moment coupling were used as boundaries. An unscented Kalman filter was used to estimate the model’s parameters. A single column of the steel frame was considered as the experimental substructure of this hybrid simulation. The column was loaded by three hydraulic actuators. The results showed that the proposed method is valid and effective.

Krenk et al. [12] performed a hybrid simulation on a zipper-braced steel frame under earthquake excitation. This type of structures have a complex force redistribution. Thus, a hybrid model was developed in which both physical and analytical sub-assemblies exhibited a complex nonlinear behavior. This was done by developing a new hybrid simulation architecture of the experimental setup. Also, the predictor and corrector algorithms had to be customized to assure stability. The results from the hybrid and the numerical simulation were in good agreement, proving the accuracy of the models. An energy error indicator showed that the accuracy of the hybrid simulation was sufficiently low.

You et al. [22] investigated the compatibility and equilibrium between the numerical and physical parts of a hybrid simulation in which OpenFresco was used as a middleware and servo-hydraulic actuators were used to apply loads. The objective of the hybrid simulation was to check the fatigue response of wind turbine blade. FE analysis was carried out on the rest of the turbine structure using ANSYS.
Wang et al. [25] conducted a study on a steel moment-resisting frame. One of the columns, considered as the physical substructure, was subjected to fire load according to the ISO834 fire curve. The remaining parts of the frame were analyzed numerically using Abaqus. The physical and numerical parts were interconnected through a UT-SIM framework, which allowed applying the target displacement and receiving the reaction forces simultaneously. The study concluded that there are differences in the results between the numerical and experimental outputs. The reason behind that was the experimental layout restrictions.
3 Theory

3.1 Hybrid testing

A structure can be defined as a discrete-parameter system or as a discrete spring mass system with a certain amount of degrees of freedom (DOFs) controlled by an actuator in a quasi-static fashion [4], [13]. Thus, the following equation of motion is to be solved [13]:

\[ M \ddot{x} + C \dot{x} + Kx = F \]  

(1)

Where \( M \) and \( C \) are the mass and damping matrices, \( \dot{x} \) and \( \ddot{x} \) are the nodal velocities and acceleration, respectively, and \( F \) is the external excitation. In a quasi-static analysis of a linearly elastic system, there are no dynamic effects. Therefore, both mass and damping matrices in equation 1 are set to zero:

\[ K \ddot{x} = FM \ddot{x} + C \dot{x} + Kx = F \]  

(2)

In a hybrid testing procedure, equation 2 can be solved as follows [4][13][23]:

\[ R^{int}[i+1] = R^{int}[i] + K \Delta uM \ddot{x} + C \dot{x} + Kx = F \]  

(3)

\[ \Delta u = u_{i+1} - u_iM \ddot{x} + C \dot{x} + Kx = F \]  

(4)

\[ R^{int}[i] + K \Delta u = R^{ext}[i+1]M \ddot{x} + C \dot{x} + Kx = F \]  

(5)

\[ K \Delta u = R^{ext}[i+1] - R^{ext}[i]M \ddot{x} + C \dot{x} + Kx = F \]  

(6)

where \( R^{int}[i+1] \) and \( R^{int}[i] \) are the internal force in new increment, and the initial internal force, respectively, \( K \) is the stiffness matrix of the specimen, \( u_{i+1} \) is the displacement in new increment, \( u_i \) is the initial displacement and \( R^{ext}[i+1] \) is the new external force.

The equations above can be used to conduct a hybrid simulation according to the following steps:

1- The FE-model is subjected to loading.

2- The computed displacements of the experimental substructure in the FE-model are translated to the test specimen in a quasi-static manner by jacks or hydraulic actuators.

3- The reaction forces of the test specimen are measured by load transducers.
4- The reaction forces are sent back to the FE-solver in order to calculate the new displacements of the FE-model.

5- Steps 2 to 4 are iterated in a closed loop.

3.2 Truss structures

3.2.1 Buckling failure in truss structures

A truss structure is made up of members that are arranged and connected into triangles so that the entire assembly can act as a single object [14]. This helps the whole structure to carry the loads and transfer them to other supporting structures such as walls and load-bearing beams. A truss structure consists of two-force members.

There are five commonly known failure modes of trusses: fractures (ductile and brittle fractures), yielding, deflection (insufficient stiffness), fatigue, creep, and buckling.

Buckling takes place when a truss element is subjected to compressive stress as shown in Figure 3. Buckling failure is dangerous since it might occur even before material failure.

Figure 3. Buckling in slender members [14].

The critical compression load at which a truss member buckles is given as:

\[ P_{cr} = \frac{\pi^2 E}{L^2} M \ddot{x} + C \dot{x} + Kx = F \]  

(7)
Where $E$ is the modulus of elasticity, $I$ is the moment of inertia and $L$ is the member length. When the applied load $P > P_{cr}$ as shown in Figure 3, buckling will increase until the element collapses.

3.2.2 Analysis of truss structures

In FE analysis, a truss is treated as an elastic structure, which means that the change in length of any member $\Delta L$ is proportional to the load applied in the truss $P$, as shown in Figure 5. This results in displacements at the nodes A and B ($u_A$ and $u_B$).

![Figure 4. Nodal displacement in truss structure [16].](image)

The change in length induces a normal force $N$ in the member [15], as shown in Figure 6. The resulting normal force is given as:

$$N = A \sigma$$  \hspace{1cm} (8)
Where \( A \) is the area of the cross-section. Assuming that the bar is linearly elastic, it will follow Hooke’s law,

\[
\sigma = \varepsilon E
\]

(9)

Where \( E \) is Young’s modulus and it is the normal strain defined as

\[
\varepsilon = \frac{u_2 - u_1}{L}
\]

(10)

Where \( u_1 \) and \( u_2 \) are the displacements of node 1 and 2 with the positive direction along the \( x \)-axis.

When combining Eq. 4 and 6 the normal force can be expressed as

\[
N = k(u_2 - u_1)
\]

(11)

Where \( k \) is the bar stiffness defined as

\[
k = \frac{AE}{L}
\]

(12)

For a two-dimensional truss, a bar element is loaded in the \( xy \)-coordinate system [15]. Each element has global nodal points, \( i \) and \( j \), as shown in Figure 7. There are forces (\( P \)) and displacements (\( u \)) at \( i \) and \( j \) acting in the positive \( x \)- and \( y \)-direction.
In order to determine the relationship between the global forces and displacements, a local coordinate system is used, which is rotated with an angle about the xy-system, as shown in Figure 8 [15]. Hence, there are new forces (P) and displacements (u) acting at node 1 and 2 in the positive x'- and y'-direction.

Accordingly, the following equations are obtained in matrix form [15],

$$k \begin{bmatrix} 1 & -1 \\ 1 & 1 \end{bmatrix} \begin{bmatrix} u_1' \\ u_2' \end{bmatrix} = \begin{bmatrix} p_1' \\ p_2' \end{bmatrix}$$  \hspace{1cm} (13)

Thereby, equation (9) is modified and expressed as [15]:

$$K^{er} \alpha^{er} = f^{er}$$  \hspace{1cm} (14)

$$\alpha^{er} = \begin{bmatrix} u_1' \\ u_2' \\ u_3' \\ u_4' \end{bmatrix} \quad f^{e'} = \begin{bmatrix} p_1' \\ p_2' \\ p_3' \\ p_4' \end{bmatrix}$$  \hspace{1cm} (15)

$$K^{er} = \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$  \hspace{1cm} (16)
When small displacements are assumed, there are no displacements 
\(u_2' = u_4' = 0\) along the \(y'\)-axis, and therefore no forces \(P_2' = P_4' = 0\).

The following geometrical relationship between the global and local displacements can be obtained [15]:
\[
\begin{align*}
    u_i &= u_1' \cos \phi - u_2' \sin \phi, \\
    u_{i+1} &= u_1' \sin \phi + u_2' \cos \phi
\end{align*}
\]  
(2)
\[
\begin{align*}
    u_j &= u_3' \cos \phi - u_4' \sin \phi, \\
    u_{j+1} &= u_3' \sin \phi + u_4' \cos \phi
\end{align*}
\]  
(3)

When combining (17) and (18) the expression becomes:
\[\alpha^e = L^e e'\]  
(4)

Where
\[
\alpha^e = \begin{bmatrix}
    u_i \\
    u_{i+1} \\
    u_j \\
    u_{j+1}
\end{bmatrix}
\]  
(20)
\[\alpha'^e = \begin{bmatrix}
    u_1' \\
    u_2' \\
    u_3' \\
    u_4'
\end{bmatrix}\]

And
\[L^e = \begin{bmatrix}
    \cos \phi & -\sin \phi & 0 & 0 \\
    \sin \phi & \cos \phi & 0 & 0 \\
    0 & 0 & \cos \phi & -\sin \phi \\
    0 & 0 & \sin \phi & \cos \phi
\end{bmatrix}\]  
(21)

Where \(L\) is the transformation matrix [15]. Consequently, it follows that
\[L^e L = L L^e = 1\]  
(22)

Thus, multiplying (15) with \(L\) gives
\[\alpha'^e = L \alpha^e\]  
(23)

In the same way the relationship between the global and local forces is
\[\alpha''^e = L \alpha^e\]  
(24)

Where
Finally, when combining (23) and (14), the relationship between the global element forces and displacements becomes:

$$K^e a^e = f^e$$  \hspace{1cm} (27)

Where the global stiffness matrix ($K^e$) is given as:

$$K^e = k [\begin{array}{cccc}
\cos^2 \theta & \cos \omega \sin \theta & -\cos^2 \theta & -\cos \omega \sin \theta \\
\cos \omega \sin \theta & \sin^2 \theta & -\cos \omega \sin \theta & -\sin^2 \theta \\
-\cos^2 \theta & -\cos \omega \sin \theta & \cos^2 \theta & \cos \omega \sin \theta \\
-\cos \omega \sin \theta & -\sin^2 \theta & \cos \omega \sin \theta & \sin^2 \theta \\
\end{array}]$$  \hspace{1cm} (28)

In addition, the relationship between the displacements (Figure 9) and forces is given as:

$$k \begin{bmatrix} 1 & -1 \\ 1 & 1 \end{bmatrix} \begin{bmatrix} u'_1 \\ u'_2 \end{bmatrix} = \begin{bmatrix} p'_1 \\ p'_2 \end{bmatrix}$$  \hspace{1cm} (29)

$$\begin{bmatrix} u'_1 \\ u'_2 \end{bmatrix} = \begin{bmatrix} \cos \omega \sin \theta & 0 \\ 0 & \cos \omega \sin \theta \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix}$$  \hspace{1cm} (30)
Consequently, the force can be calculated as shown below:

\[
\frac{L^r}{k} \frac{L}{K^s} u = p
\]  

(31)
4 Methods and Implementation

In this study, a hybrid test was conducted on the steel truss structure shown in Figure 9. As mentioned earlier, hybrid testing requires dividing the structure into numerical and physical substructures. The numerical substructure was modeled using Abaqus, while the physical substructure (i.e. member 1 in Figure 9) was tested experimentally. This chapter explains the methods used for numerical modeling, experimental testing, and communication between the numerical and physical substructures.

4.1 Numerical modeling

Abaqus was used for modeling and analysis of the numerical substructure of the steel truss. Each element was modeled as a deformable 2D planar element. Once the truss was modeled in Abaqus, it was assigned with a section defining its elastic material properties given in Table 1 along with cross-sectional area. Displacements and DOF constraints were applied as boundary conditions. The FE analysis was carried out using the explicit solver in Abaqus.

Table 1. Steel element material properties and cross-section details

<table>
<thead>
<tr>
<th>User steel element of size (700 x 20 x 20 x 3 mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Elastic Modulus</strong></td>
</tr>
<tr>
<td><strong>Density</strong></td>
</tr>
<tr>
<td><strong>Cross-sectional area</strong></td>
</tr>
</tbody>
</table>
4.2 Experimental setup

The equipment used for hybrid testing are often similar to those used for conventional quasi-static testing. As shown in Figure 10, the hybrid test setup consists of the following components [13]:

1. A servo-hydraulic system, consisting of an actuator, controller, and pressurized hydraulic oil supply.

2. A physical specimen with the actuators attached at the DOF, where the displacements are to be imposed.

3. Sensors to measure the responses of the physical specimen as displacements and restoring forces.

4. A data acquisition system.

Member 1 of the truss shown in Figure 9 was considered as the physical substructure. The double-angled steel member depicted in Figure 11 was manufactured and welded at Linnaeus university/Building technology Lab, with dimensions (700x20x20x3 mm). A testing machine (MTS810) was used to apply loads on the specimen. As shown in Figure 12, the tested specimen was mounted on the machine with a load cell directly above it. The crosshead was elevated to a height that would fit the sample size with the upper moving head.
4.3 Communication between the numerical and experimental substructures

Abaqus allows writing a user-defined element as a FORTRAN90 subroutine called UEL. This subroutine takes the output of the experimental test and control the element behavior accordingly. Therefore, as illustrated in Figure 13, the behavior of the substructure (member 1) in the FE model was governed by the response of the tested specimen measured experimentally.

OpenFresco was used as a middleware to establish reliable and stable communication between the UEL subroutine of the Abaqus model and the testing machine. In each increment, the reaction force measured by the load cell was acquired using the data acquisition system and sent to the FE model through an Ethernet TCP/IP connection controlled by OpenFresco.
4.4 Overall procedure

The overall procedure of the hybrid test conducted in this study can be summarized as follows:

1. Establish a communication interface between the UEL subroutine and the experimental setup through the middleware OpenFresco.

2. Use the testing machine to apply a compressive load on the specimen in a quasi-static manner.

3. The reaction forces recorded by load cell at the end of the increment are sent to the FE model to update it before the next increment.

4. The specimen will fail after a certain amount of time (Figure 14) and a popup window will prompt the user to terminate the hybrid simulation.

5. Both experimental and FE results are imported and compared to calculate the error.

Figure 15 explains the overall hybrid testing procedure in details.
Figure 14. deformed test element.

Figure 15. Flowchart of the hybrid test.
5 Analysis and Results

Three studies were carried out on the steel truss. In the first study, two experimental tests were conducted on the specimen shown in Figure 12 without any interaction with the numerical model. In the second study, a numerical simulation of the entire truss structure was performed using Abaqus without any interaction with the physical substructure. The third study involved conducting three different hybrid simulations in which the steel member was loaded and its response was recorded and sent back to Abaqus as explained in Chapter 4. The following sections present the data obtained from the experimental, numerical, and hybrid tests.

5.1 Experimental tests

Two tests were carried out on the steel member using the setup shown in Figure 12. In both tests, the deformation rate was set as 3 mm/s. The reaction forces were recorded by load cells.

5.1.1 Experimental test 1 (elastic mode)

Figure 16 shows the response of the member during the first experimental test. During the loading step, the measured reaction force increased in a linear elastic manner with the applied deformation. Once the specimen was unloaded, the member returned to its original length, indicating that the maximum stress did not exceed the yield strength of the steel member.

![Figure 16. Force vs. displacement in test 1 (elastic mode).](image)

5.1.2 Experimental test 2 (elastic and plastic)

In the second experimental test, the specimen was loaded in 2 steps. In the first step, the sample was loaded elastically until 47 KN (slightly less than
the critical buckling load) and then unloaded. In the second step, the sample was compressed until buckling and then unloaded. The force-displacement graph for test 2 is shown in Figure 17.

![Force-displacement graph](image)

*Figure 17. Experimental test 2 (elastic and plastic).*

5.2 Numerical test

A numerical Abaqus analysis was conducted according to the procedure explained in the Chapter 4. The FE model of the truss structure is shown in Figure 18.

![FE model of the steel truss](image)

*Figure 18. FE model of the steel truss.*

The reaction force at node 1 (see Figure 9) under the applied displacement is shown in Figure 19. The calculated maximum load was about 41 kN. The distribution of von Mises stresses is shown in Figure 20. No buckling or plastic failure was observed at the maximum load.
Figure 19. Reaction Force vs. the applied displacement at node 1 calculated using Abaqus.

Figure 20. Distribution of von Mises stresses calculated using Abaqus.

5.3 Hybrid simulations

Three hybrid simulations were conducted. In the first and second simulations, the member was loaded within its elastic limits, while in the third simulation, the sample was loaded beyond the critical buckling load, and therefore failed due to buckling.

5.3.1 Hybrid simulation 1

The force-displacement graph for the first hybrid simulation is shown in Figure 21. As can be noticed, the maximum reaction at node 1 is 27.5 kN. The sample was loaded and unloaded within the elastic zone.
5.3.2 Hybrid simulation 2

Hybrid simulation 2 is similar to the first simulation except that the maximum load was increased from 27.5 kN to 30 kN as can be seen in Figure 22. The behavior of the physical substructure in this hybrid simulation was also linear elastic.

5.3.3 Hybrid simulation 3

In the third hybrid simulation, the truss was loaded until the reaction force measured at node 1 was slightly less than the critical buckling load of the steel member. As shown in Figure 23, the member fully recovered its original length when the structure was unloaded. Next, the hybrid structure was loaded beyond the critical load until buckling failure occurred.
Figure 23. Force-displacement graph for hybrid simulation 3
6 Discussion

6.1 Discussion of the experimental tests

The response in the first experimental test was within the elastic regime, since the maximum load was kept below the critical buckling load. The specimen was loaded and unloaded. During the loading step, the reaction force increased linearly indicating a linear elastic behavior. The second experimental test involved two steps. In the first step, the member was loaded and unloaded within its elastic limits. In the second step, the load was gradually increased until the sample failed in buckling. The error between experiment 1 and the first step of experiment 2 was minimal (see Figures 16 and 17). This indicates that the response measured by the load cells was accurate.

6.2 Discussion of the hybrid simulations

A comparison between the reaction force estimated numerically and that obtained from the three hybrid simulations is shown in Figure 24. It can be noticed that the maximum reaction force obtained in the first and second hybrid simulations (HS1 and HS2) are lower than that obtained in the numerical test and the third hybrid simulation (HS3). This is because the reaction force in HS1 and HS2 was deliberately kept low to ensure a linear elastic response.

The comparison indicates a very good agreement within the elastic zone between the numerical results and the output of the third hybrid simulation (HS3). However, since the FE model was defined only within the elastic region (i.e. only the elastic material properties were assigned to the truss elements), the buckling failure could not be simulated numerically.

This highlights the efficiency of hybrid simulations in the situations where the numerical models are lacking. Hybrid testing enable structural engineers to obtain accurate simulations of complex structural phenomena without requiring a full-scale experimental test or sophisticated numerical modeling.
Figure 24. Force-displacement plots for all the hybrid simulation with pure abaqus analysis

6.3 Sources of error

There are several reasons behind the error between the numerical and experimental results in terms of both displacement and forces:

1. The losses associated with the experimental process and equipment.

2. The uncertainties associated with measurements and sensor calibration, which are called measurement system uncertainty.

3. The frictional forces and losses at the joints of the actuators.

Therefore, the accuracy of hybrid simulations can be further improved by eliminating the sources of error discussed above.
7 Conclusions

This study presented and demonstrated a hybrid simulation approach, which was used to test the critical element of a truss structure. The entire truss was modeled numerically using Abaqus and coupled with an experimental setup. The coupling between the experiment and numerical simulation was facilitated using the middleware OpenFresco.

The results of the hybrid tests were in good agreement with the output of the experimental and numerical tests. The hybrid simulation was successful in predicting the failure load of the steel truss element. Besides, it provided a lot of insight into the crushing time of the whole system when subjected to a predefined displacement.

In conclusion, the integration of the data collected from the physical substructure into the hybrid simulation was successful. The most important outcome of the hybrid simulation is that a satisfactory accuracy was achieved without large experimental testing or sophisticated numerical modeling.

For future work, the hybrid setup used in this study can be employed for simulating more challenging structural systems (e.g. structures with a large number of degrees-of-freedom).
References


[25] Xugang wang,Robin E.kim,Oh-Sung Kwon and Inhwan Yeo.hybrid simulation method for a structure subjected to fire and its application to a steel frame.
Faculty of Technology
351 95 Växjö, Sweden
Telephone: +46 772-28 80 00, fax +46 470-832 17