COMPDYN 2015
5th ECCOMAS Thematic Conference on
Computational Methods in Structural Dynamics and Earthquake Engineering
M. Papadrakakis, V. Papadopoulos, V. Plevris (eds.)
Crete Island, Greece, 25–27 May 2015

STRUCTURAL INTEGRITY ANALYSIS AND OPTIMIZATION OF AN ELEVATOR FRAME, THROUGH FE MODELING AND EXPERIMENTAL TESTS

Dimitrios Giagopoulos[1], **Iraklis Chatziparasidis**[1], **Nickolas S. Sapidis**[1]

[1] Department of Mechanical Engineering, University of Western Macedonia
Kozani 50100, Greece
e-mail: dgiagopoulos@uowm.gr, ihatz@kleemann.gr, nsapidis@uowm.gr

Keywords: *FE modeling, Large scale structures, Model updating, Stress evaluation, Structural Dynamics.*

Abstract. *A systematic structural integrity analysis and optimization of an elevator chassis under real dynamic load conditions are presented in this work. The special feature of this paper is that the study was performed on industrial an elevator system (produced by Kleemann Hellas S.A.), including all details/complexities of a commercial system. The procedure proposed for solving and analysing this specific problem includes the following steps. First, the frame and the cabin of the elevator are modeled numerically by discretizing them geometrically according to the FE method. FE modeling of this structure is not straightforward because of these two aspects of the analysis. (a) When safety gear is activated and the elevator stops, braking forces act on the system, whose dynamic response must be accurately simulated. (b) An efficient modeling method is required for the various elevator parts which are in contact with each other and are connected together by screws through "oval type" holes. The initial FE model is updated and validated through an experimental investigation of its dynamic response when the elevator stops using instantaneous or progressive safety gear. These experimental tests were performed under real operating conditions, using an experimental device that was designed exactly for this purpose and aimed at recording the acceleration time histories at the connection points of the frame with the safety gear and at other locations used as reference points. The acceleration time histories at the connection points are subsequently used as base excitation for the FE model of the frame and the corresponding stresses developed are evaluated. On the basis of these numerical results, the critical points of the frame are selected, as corresponding to larger stresses. Finally, to test the reliability of the proposed method, strain gauges are placed at the critical points of the frame and measurements are carried out, under similar dynamic load conditions, in order to experimentally verify the stresses calculated above. Comparison of the numerical and experimental data verifies that the proposed "mixed computational-experimental" analysis method is quite reliable.*

1 INTRODUCTION

Ever evolving elevator design requirements frequently require improvements/modifications of either specific mechanical components or even of entire structures. FE modeling of elevator systems is not straightforward because of these two aspects of the analysis. (a) When safety gear is activated and the elevator stops, braking forces act on the system, whose dynamic response must be accurately simulated. (b) An efficient modeling method is required for the various elevator parts which are in contact with each other. To achieve these modeling issues, it is important to develop an accurate Finite Element Analysis procedure, in order to simulate the dynamic behavior of these systems. This is the main technical issue addressed by the present work.

The equations of motion of mechanical systems with complex geometry are first set up by applying classical finite element techniques. As the order of these models increases, existing numerical and experimental methodologies for a systematic determination of their dynamic response become inefficient. Therefore, there is a need for the development and application of new appropriate methodologies for investigating the dynamics of large scale mechanical models in a systematic and efficient way. Traditionally, in the area of structural dynamics this is done by first employing methodologies that reduce the dimensions of the original system. This paper applies a time domain reduction method [1-5].

The main objective of the present work is to demonstrate the advantages of applying appropriate numerical and experimental methodologies in order to accurately predict the dynamic response and the identification of the critical points in an elevator system. This is done by applying a numerical method for determining the equations of motion for the main elevator parts, while the dynamic behavior of the remaining components are taken into account through the application of appropriate experimental measurements. The special feature of this paper is that the study was performed on industrial an elevator system including all details/complexities of a commercial system.

The organization of this paper is as follows. In the following section, a brief but complete outline of the methodology that reduce the dimensions of the original system is presented. Then, in the third section, the effectiveness and accuracy of the Finite Element Analysis procedure is demonstrated, by presenting numerical and experimental results obtained for the real elevator system. Here, the reliability of the methodology was tested first analytically in the elevator chassis with the platform of the cabin and next in the full elevator system (chassis and cabin). The study concludes by presenting a summary of the results obtained.

2 CLASS OF MECHANICAL SYSTEMS EXAMINED – EQUATIONS OF MOTION

The equations of motion of mechanical systems with complex geometry are commonly set up by applying finite element techniques. Quite frequently, a systematic investigation of the dynamics of large scale mechanical structures leads to models involving an excessive number of degrees of freedom. Therefore, a computationally efficient solution requires application of methodologies reducing the numerical dimension of the original model [1-6, 8, 9]. Next, the basic steps of a time domain reduction method is presented briefly.

For simplicity, consider a mechanical system consisting of two subsystems, say A and B. Moreover, let the equations of motion for subsystem A be derived from the following classical form

$$\hat{M}_A \ddot{\underline{x}}_A + \hat{C}_A \dot{\underline{x}}_A + \hat{K}_A \underline{x}_A = \hat{\underline{f}}_A(t) \tag{1}$$

where $\hat{M}_A$, $\hat{C}_A$ and $\hat{K}_A$ are, respectively, the mass, damping and stiffness matrix of the subsystem A, with the vector $\underline{\hat{f}}_A(t)$ representing the external forcing. For a typical model, the number of these equations may be quite large. However, for a given level of forcing frequencies, it is possible to reduce significantly the number of the original degrees of freedom, without sacrificing the accuracy in the numerical results, by applying standard component mode synthesis methods [1, 6]. This can be achieved through an approximate coordinate transformation of the form

$$\underline{x}_A = \Psi_A \underline{q}_A \tag{2}$$

The transformation matrix Ψ_A includes an appropriately chosen set of the lowest frequency normal modes of component A, corresponding to support-free conditions [1]. The number of these modes depends on the accuracy required in the response frequency range examined. Consequently, matrix Ψ_A is completed by a set of static correction modes of component A [3, 4]. Employing this transformation, the original set of equations (1) can be replaced by a considerably smaller set of equations, expressed in terms of the new generalized coordinates $\underline{q}_A$. More specifically, application of the Ritz transformation (2) onto the original set of equations (1) yields the much smaller dimension set

$$M_A \underline{\ddot{q}}_A + C_A \underline{\dot{q}}_A + K_A \underline{q}_A = \underline{f}_A(t) \tag{3}$$

where

$$M_A = \Psi_A^T \hat{M}_A \Psi_A,\ C_A = \Psi_A^T \hat{C}_A \Psi_A,\ K_A = \Psi_A^T \hat{K}_A \Psi_A \text{ and } \underline{f}_A = \Psi_A^T \underline{\hat{f}}_A .$$

Moreover, the set of unknowns can be split in the form

$$\underline{q}_A = (\underline{p}_A^T \quad \underline{x}_b^T)^T$$

where $\underline{p}_A$ includes coordinates related to the response of internal degrees of freedom of component A, while $\underline{x}_B$ includes the boundary points of component A with component B. Next, similar sets of equations of motion are obtained for component B. Namely, the equations of motion are first set up in the form

$$M_B \underline{\ddot{q}}_B + C_B \underline{\dot{q}}_B + K_B \underline{q}_B = \underline{f}_B(t) \tag{4}$$

with coordinates

$$\underline{q}_B = (\underline{p}_B^T \quad \underline{x}_b^T)^T$$

Then, a proper combination of equations (3) and (4) leads to the equations of motion of the composite system in the classical form

$$M\underline{\ddot{q}} + C\underline{\dot{q}} + K\underline{q} = \underline{f}(t) \tag{5}$$

with coordinates

$$\underline{q} = (\underline{p}_A^T \quad \underline{p}_B^T \quad \underline{x}_b^T)^T .$$

The stiffness matrix of the composite system can be obtained by considering the total potential energy of the system. Likewise, the mass matrix of the composite system is obtained by considering the corresponding kinetic energy, while the forcing vector is determined by considering the virtual work.

3 EXPERIMENTAL APPLICATION TO AN ELEVATOR SYSTEM

In this section, the emphasis is placed on applying the methodology proposed to a real elevator system. The ultimate goal was to develop an accurately Finite Element Analysis procedure, in order to simulate the dynamic behavior of this system. The proposed procedure is presented in detail in the following sections.

3.1 Finite Element Model

The experimental system examined addresses a real elevator system which is shown in Figure 1. In this example, the system consists of two main substructures. The first part is the chassis of the elevator, while the second part is the cabin. The geometry of these structures was discretized using mainly rectangular and triangular shell finite elements. Additionaly, some other elements like solid (hexahedral) and rigid body elements were also used. The total number of degrees of freedom of the resulting model (chassis and cabin) was about 4,500,000. Due to the large size of this model, the solution of the finite element model was completed by using appropriate commercial software [13, 14]. The FE models of these two parts are presented in Figures 2a and 2b, respectively.

Figure 1. ELEVATOR SYSTEM.

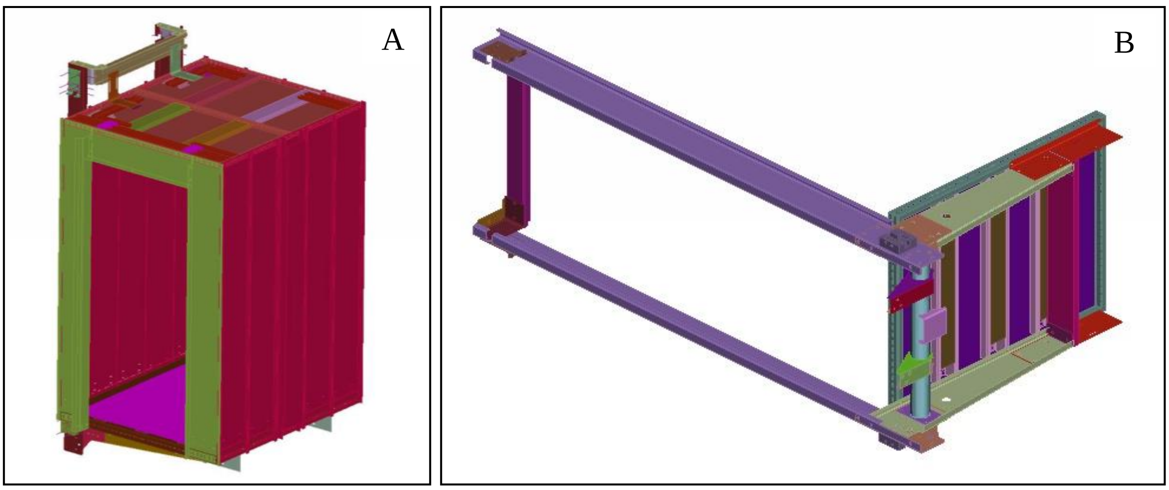


Figure 2. FINITE ELEMENT MODEL OF (A) THE CABIN AND (B) THE CHASSIS.

After development of the overall finite element model, including the coordinate reduction and synthesis part, the next step was to examine the dynamic response of system when safety gear is activated and the elevator stops. In order to accurately simulate this procedure, it's necessary first to verify the accuracy of the developed FE model and also identify the braking forces that acting on the system. To achieve this a combination of numerical and experimental methods was applied [9].

3.2 Determination of Acceleration Levels Under Real Dynamic Load Conditions

In order to verify the accuracy of the FE model of the system and also identify the braking forces acting on the system, we select, to examine only the elevator chassis including the platform of the cabin with full load. On this system, triaxial accelerometers are placed at (6) selected positions. These positions along with the measurement directions are presented in Figure 3: the two connection points of the frame with the safety gear (A1, A2) are included along with four other locations (A3-A6) which are used as reference points. A series of experimental trials were performed under real operating conditions (free fall of the elevator frame), using an experimental device designed and constructed exactly for this purpose (by Kleemann Hellas S.A), aimed at recording the acceleration time histories at the selected six points.

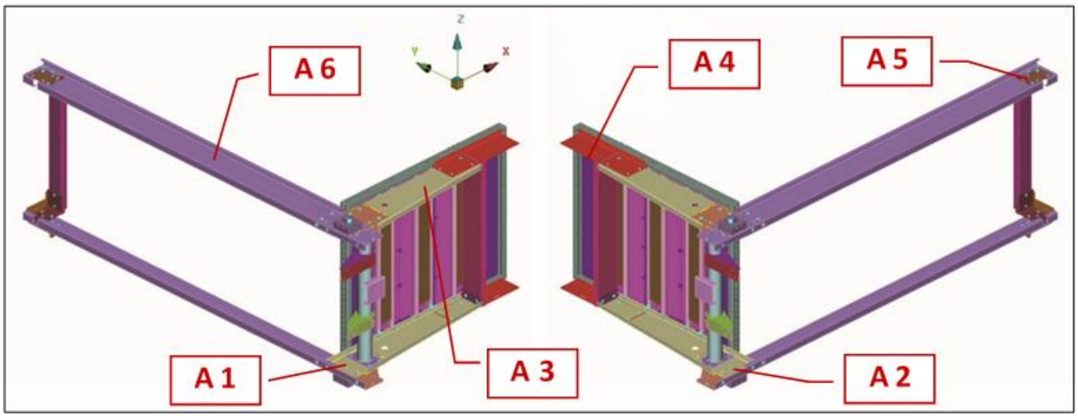


Figure 3. MEASUREMENT LOCATIONS OF ACCELERATION TIME HISTORIES.

These measurements were performed using a data acquisition system (CDAQ) by National Instruments, with the related software which was made at NI Labview environment. The measured frequency range for the tests conducted was selected to be 0-1024 Hz. Figure 4a depicts the data acquisition system, while Figure 4b presents a picture of the load (weight) placed on the elevator platform.

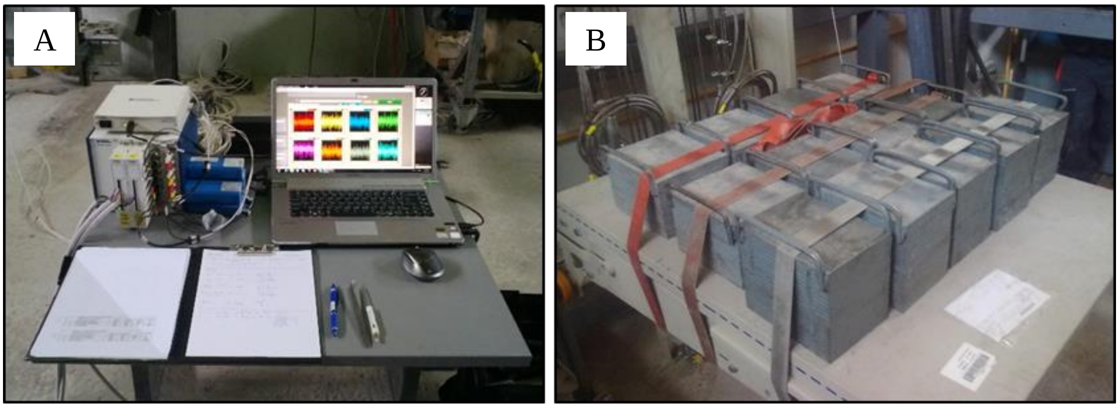


Figure 4. (A) DATA ACQUISITION SYSTEM and (B) ELEVATOR PLATFORM WITH LOAD.

Also, Figure 5 presents indicative photos of a two measurement positions.

Figure 5. ACCELERATION MEASUREMENT LOCATIONS AT A CONNECTION POINT (A1) AND AT A REFERENCE POINT (A4).

Next, Figures 6a, 6b and 6c present typical acceleration time histories in the three directions (X-longitudinal, Y-vertical, Z-transverse), at the connection point A2 of the chassis with the safety gear. These time histories were measured during one of the tests performed by using a progressive safety gear. Also, each figure presents the corresponding peak-to-peak and GRMS values.

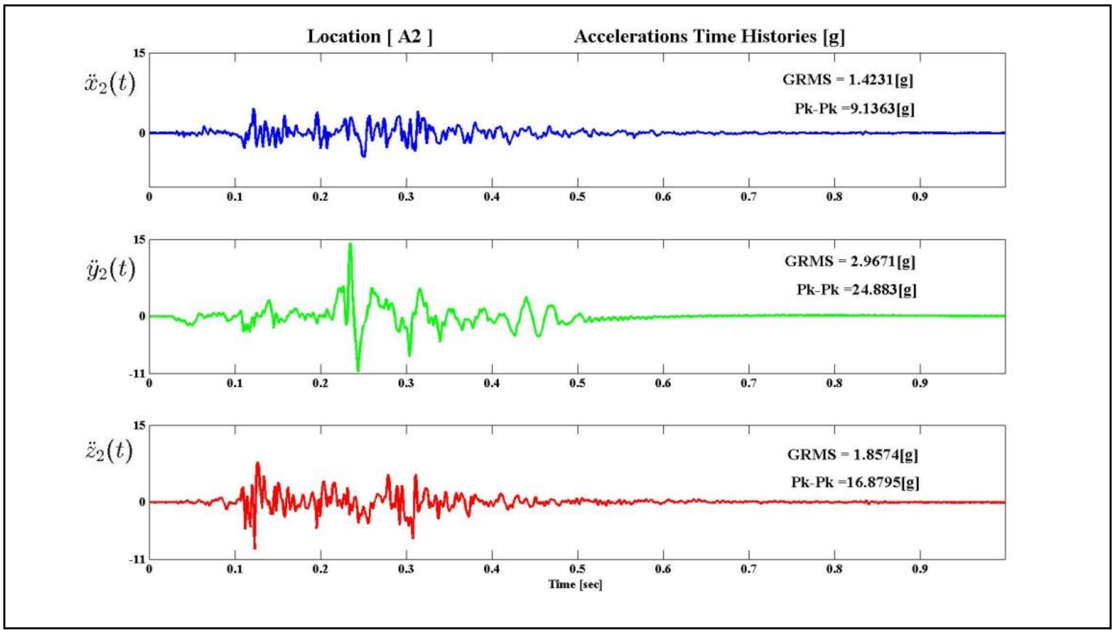


Figure 6. ACCELERATION HISTORIES IN THE: (A) X-LONGITUDINAL DIRECTION, (B) Y-VERTICAL DIRECTION AND (C) Z- TRANSVERSE DIRECTION, WITH PEAK TO PEAK AND GRMS VALUES.

3.3 Results of FE Model Analysis – Identification of Critical Points

Next, the measured acceleration time histories were used to determine the braking force that acting on the system. After several experimental trials, the form of the corresponding time-varying braking forces, in each progressive safety gear, were calculated. Then, these force time histories were imported as base excitation in the finite element model of the system. Also, in order to solve this transient response analysis problem in a computationally effective way, a reduction in the dimensions of the original system is performed, so that the results are accurate in the frequency range 0-500 Hz. The total number of degrees of freedom in the reduced model was about 3,500, which is much smaller than the number of degrees of freedom in the original model (1,200,000). The reduced model was solved numerically in order to calculate the maximum stresses developed for the given loading. The identified critical points of the structure include mainly areas of the chassis arms on which the platform is based. Figure 7 shows selected results, in which some indicative points of the superstructure with maximum stresses are presented.

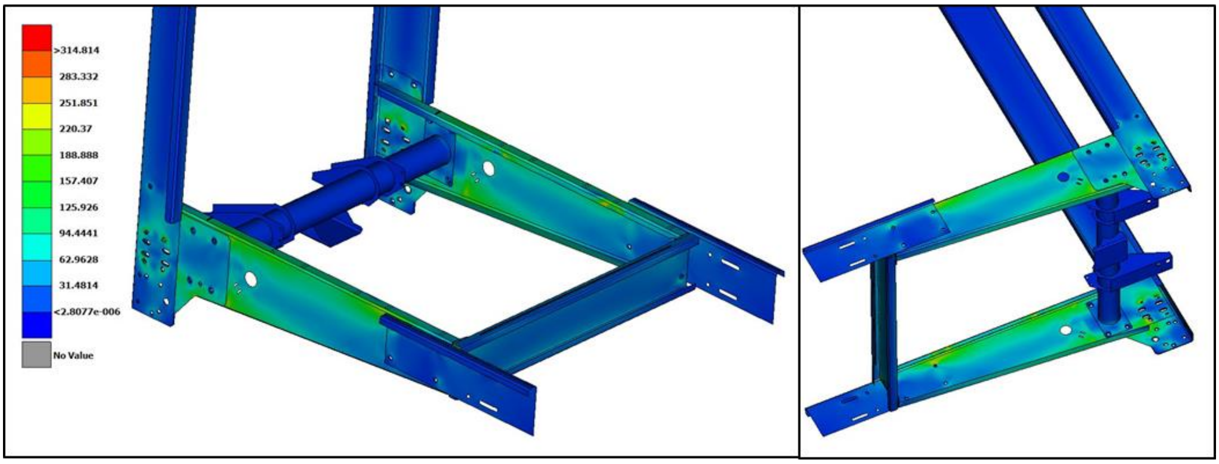


Figure 7. LOCATIONS OF THE CHASSIS WHERE MAXIMUM STRESSES APPEAR.

3.4 Validation of the Applied Methodology

In order to test the reliability of the method applied, strain gauges were placed at (5) selected critical points of the chassis and a new set of measurements was carried out under similar dynamic loading conditions, to experimentally verify the stress levels developed. These positions are presented in Figure 8 and include locations on the right side of the chassis arms (SG1, SG2 and SG3) and on the left side (SG4 and SG5).

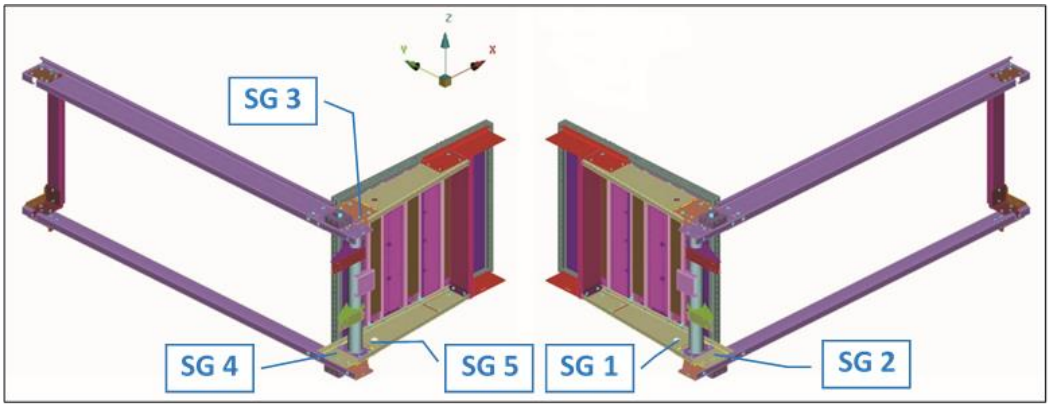


Figure 8. MEASUREMENT LOCATIONS WHERE MAXIMUM STRESS APPEARS.

For a complete monitoring of the stress state, three bridges with a 120° angle rosette were placed at each of the selected points. Figure 9 presents photographs of the strain gauges at the measurement locations.

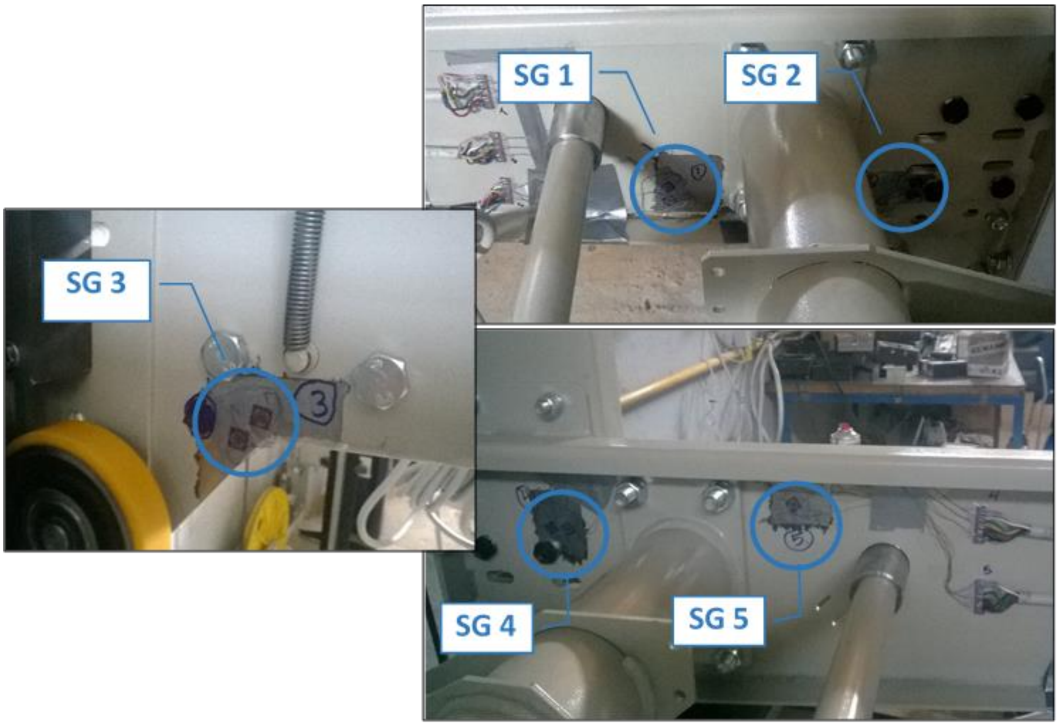
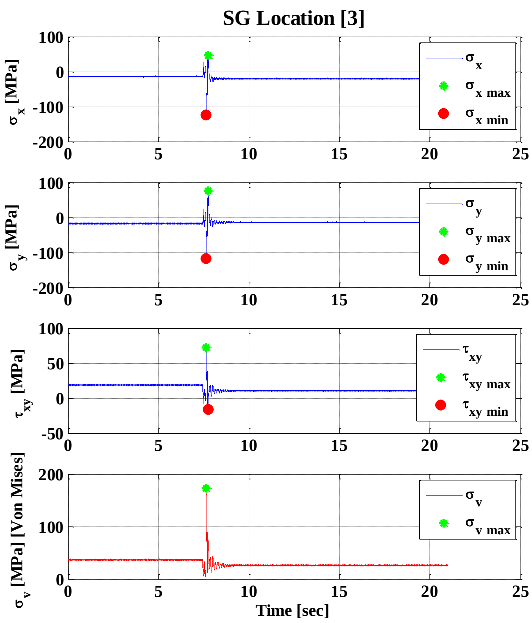


Figure 9. STRAIN GAUGES MEASUREMENT LOCATIONS.

At each measurement point, the stresses calculated are the normal stresses σ_x and σ_y, the shear stress τ_{xy} and the maximum equivalent von Mises stress. Figure 10 presents a typical part of the stress histories measured during one of the tests at the location (SG3). The first three figures show the histories of σ_x, σ_y and τ_{xy}, with the corresponding maximum positive and max mum negative values of these stresses. The last figure shows the history of the equivalent (von Mises) stress, together with the maximum value of this stress.



Progressive Safety Gear Test

	C1	C2	C3	C4	C5	C6
σ_x [MPa] =	-97.13	-14.69	47.12	61.81	-124.99	110.30
σ_y [MPa] =	-43.69	-17.74	76.84	94.57	-118.65	100.91
τ_{xy} [MPa] =	52.83	18.43	72.07	53.64	-16.57	35.01

C1 [Initial Static Stress]
C2 [Stress at Startup]
C3 [Maximum Positive Stress at Dynamic Load]
C4 [Maximum Change of Positive Stress]
C5 [Maximum Negative Stress at Dynamic Load]
C6 [Maximum Change of Negative Stress]

Maximum Value of Equivalent Stress [Von Mises] $\sigma_{v(max)}$ = 173.21 [MPa]

RMS Value of Equivalent Stress [Von Mises] $\sigma_{v(rms)}$ = 44.08 [MPa]

Figure 10. STRESS HISTORIES RECORDED DURING A TEST AT A MEASUREMENT LOCATION.

Some of the experimental and numerical results are summarized in Table 1. This table are presents the maximum values of the von Mises stress obtained in four tests (indicated by PG1-PG4) and for all the points where measurements were taken (denoted by SG1-SG5). More specifically, the third from the end column presents the maximum values for all tests, for each measurement location, the penultimate column presents the corresponding maximum values obtained by the finite element model and the last column presents their percentage difference. A comparison of the numerical and experimental data presented in Tables 1 verify that the proposed method is reliable.

	Maximum Value (peak) of Equivalent Stress Von Mises [MPa]				Maximum Value of all Structural Tests	Maximum Value of FEM Solution	Error (%)
Structural Tests	PG1	PG2	PG3	PG4			
Strain Gage Positions							
SG1	90.47	85.48	97.22	92.56	97.22	105.37	7.73
SG2	198.64	193.72	201.14	196.35	201.14	209.82	4.14
SG3	168.61	165.40	170.85	173.21	173.21	170.53	1.57
SG4	160.49	148.25	154.71	152.16	160.49	155.61	3.14
SG5	253.75	246.53	263.22	255.91	263.22	278.24	5.40

Table 1. MAXIMUM VALUE OF EQUIVALENT VON MISES STRESS [MPA].

3.5 Analysis of the FE Model of Full Elevator System (Chassis and Cabin)

Finally, in an effort to further illustrate the accuracy of the applied method, a similar analysis was applied on the finite element model of the full elevator system (chassis and cabin). More specifically, the model was solved numerically in order to calculate the maximum stresses developed for similar loading conditions, which were applied in the paragraph 3.3. Also, the corresponding experiment was set up. Figure 10, presents a comparison between the numerical and experimental results.

More specifically from the FE model analysis, arises that in some locations of the cabin the stresses developed exceeds the limits of the material yield strength. The most important of these locations are on the upper corners of the door, as shown in the Figure 11. The results of numerical analysis were confirmed from the experimental test. More specifically, the photo in figure 11, presents the deformation of the cabin at the top of the door, exactly in the same location where the FE model analysis results shows.

Comparison of numerical and experimental results indicated that the methodology applied gives accurate results and provides a useful tool in predicting the critical stress levels developed in the elevator systems under given loading conditions.

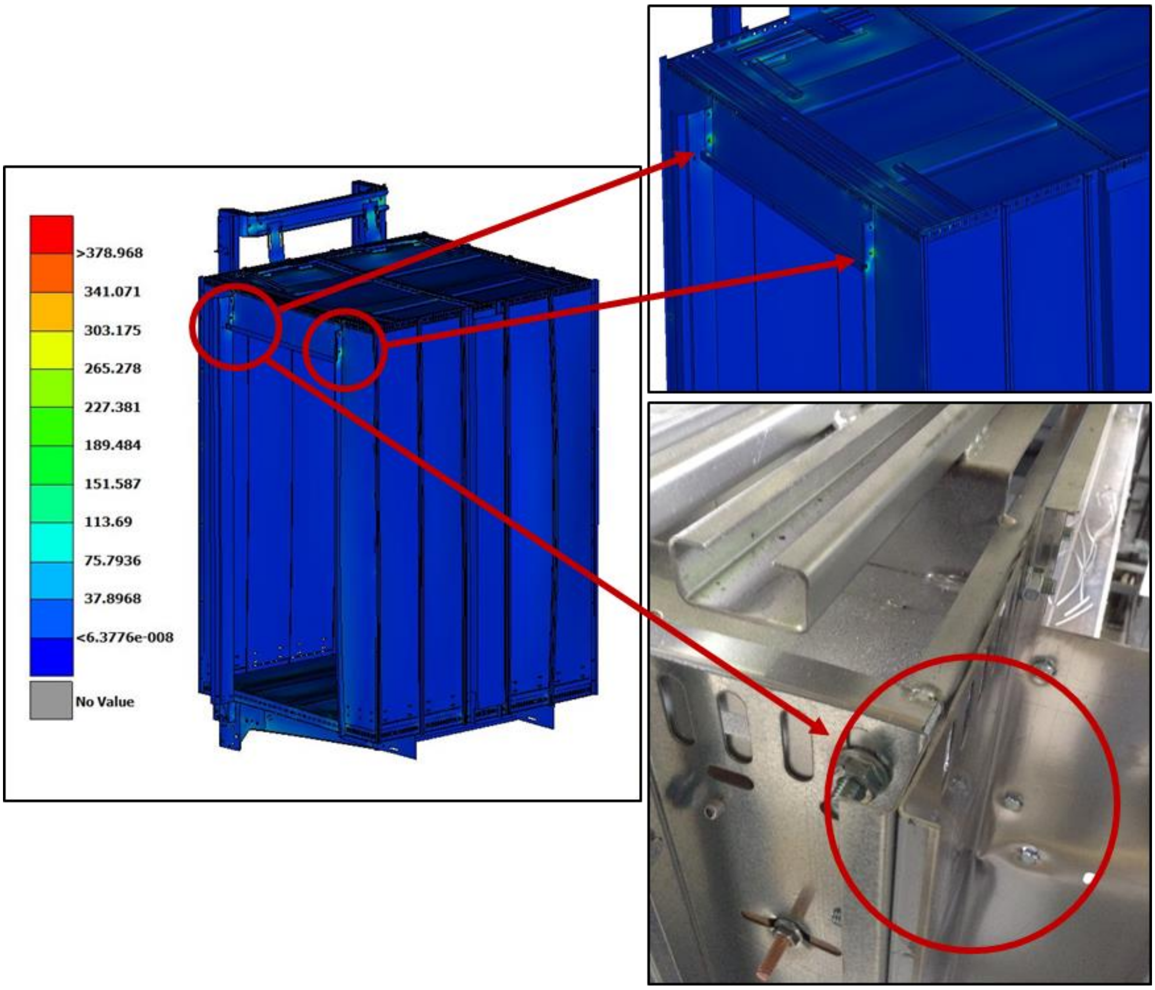


Figure 11. LOCATIONS OF THE ELEVATOR SYSTEM WHERE MAXIMUM STRESSES APPEAR AND COMPARISSON WITH EXPERIMENTAL RESULTS

4 SUMMARY

A systematic method was presented for determining the dynamic response and identifying the critical points of an elevator system when subjected to dynamic load conditions. The basic idea was to start the solution process by first discretizing geometrically the system with the finite element method. In order to verify the accuracy of the FE model of the system and also identify the braking forces acting on the system, examined only the elevator chassis including the platform of the cabin with full load. In this system, a series of experiments was performed in real operating conditions (free fall of the elevator frame), aimed at recording the acceleration time histories at selected locations. Next, a component mode synthesis method was applied in order to eliminate a substantial number of degrees of freedom of the original model. The measured acceleration histories were subsequently used in order to determine the braking force that acting on the system. Then, these force time histories were imported as a base excitation in the reduced finite element model of the system and the stresses developed under specific loading conditions were evaluated. From these numerical results, the critical points of the superstructure were selected, based on the level of the largest stresses. Finally, a new set of measurements was carried out in order to experimentally verify the stress levels developed. In an effort to further illustrate the accuracy of the applied method, a similar analysis applied in the finite element model of the full elevator system (chassis and cabin). Comparison of numerical and experimental results indicated that the methodology applied gives accurate results and provides a useful tool in predicting the critical stress levels developed in the elevator.

ACKNOWLEDGEMENTS

This research was supported by Kleemann Hellas S.A. (http://www.kleemannlifts.com.)

5 REFERENCES

[1] Craig, Jr., R.R., 1981, Structural Dynamics - An Introduction to Computer Methods, J. Wiley & Sons, New York.

[2] Craig, Jr., R.R., A review of time-domain and frequency domain component mode synthesis methods, International Journal Analytical and Experimental Modal Analysis, Vol. 2, pp. 59–72, 1987.

[3] Vermot des Roches, G., Bianchi, J.P., Balmes, E., Lemaire, R. and Pasquet, T., Using component mode in a system design process, Proceedings of the IMAC-XXVIII 2010, Jacksonville, Florida, USA, 2010.

[4] Craig, Jr., R.R., Methods of component mode synthesis, Shock and Vibration Digest Journal, Vol. 9, pp. 3–10, 1977.

[5] Farhat, C. and Geradin, M., On a component mode synthesis method and its application to incompatible structures, Computers and Structures, Vol. 51, pp. 459–473, 1994.

[6] Bennighof, J.K. and Kaplan, M.F., Frequency window implementation of adaptive multi-level substructuring, *ASME Journal of Vibration and Acoustics*, Vol. 120, pp. 409-418, 1998.

[7] MacNeal, R.H., A hybrid method of component mode synthesis, *Journal of Computers and Structures*, Vol. 1, pp. 581–601, 1971.

[8] Giagopoulos, D. and Natsiavas, S., Hybrid (numerical-experimental) modeling of complex structures with linear and nonlinear components, Nonlinear Dynamics, Vol. 47, pp. 193-217, 2007.

[9] Giagopoulos, D. and Natsiavas, S., Dynamic Response and Identification of Critical Points in the Superstructure of a Vehicle using a Combination of Numerical and Experimental Methods, Experimental Mechanics, Vol. 55, pp. 529-542, 2015.

[10] Ewins, D.J., 1984, Modal Testing: Theory and Practice, Research Studies Press, Somerset, England.

[11] Mottershead, J.E. and Friswell, M.I., Model updating in structural dynamics: A survey, *Journal of Sound and Vibration*, Vol. 167, pp. 347-375, 1997.

[12] Papadimitriou, C., Ntotsios, E., Giagopoulos, D. and Natsiavas, S., Variability of updated finite element models and their predictions consistent with vibration measurements, Structural Control and Health Monitoring, Vol. 19, pp. 630-654, 2012.

[13] DYNAMIS 3.1.1, Solver Reference Guide, DTECH, Thessaloniki, Greece, 2013.

[14] ANSA and META-Post, BETA CAE Systems S.A., Thessaloniki, Greece.