International Journal of Impact Engineering 44 (2012) 29–39

Contents lists available at SciVerse ScienceDirect

International Journal of Impact Engineering

journal homepage: www.elsevier.com/locate/ijimpeng

Dynamics of a clapper-to-bell impact

Jernej Klemenc [a,*], Andreas Rupp [b], Matija Fajdiga [a]

[a] University of Ljubljana, Faculty of Mechanical Engineering, Askerceva 6, SI-1000 Ljubljana, Slovenia
[b] University of Applied Sciences Kempten, Bahnhofstrasse 61, D-87435 Kempten, Germany

A R T I C L E I N F O

Article history:
Received 3 October 2008
Received in revised form
25 October 2011
Accepted 21 December 2011
Available online 29 December 2011

Keywords:
Clapper
Bell
Full-scale model
Impact dynamics
Finite-element method

A B S T R A C T

Church bells are exposed to severe loading conditions during ringing, which results in different damage modes due to material wear, fatigue loading, material deficiencies, different clapper-to-bell layouts, etc. As part of the activities of an EU-funded project called Maintenance and Protection of Bells (PROBELL), experimental investigations and finite-element simulations of the local contact between the clapper and the bell were carried out to study the wear-related damage to bells. First a simplified model was built to assess under the laboratory-controlled conditions the consequences of the repetitive impacts between a spherical body made from steel and a flat block made from bronze. After the results of the finite-element simulations for a simplified model were in reasonable agreement with the measured data a full-scale finite-element model for simulating the repetitive clapper-to-bell strokes was built. The simulations with the full-scale model were performed for variations of the parameters that influence the structural behaviour of the bell and the clapper: the clapper material, the clapper mass, the relative impact velocity of the clapper, the shape of the clapper, the clapper's pin support, the clapper's impact angle, the clapper's guide accuracy, the bell's sound-burp thickness and the coefficient of friction between the clapper and the bell. The agreement between the simulated and the measured results and the relation between the local stress–strain state and the damage to the bell in the contact area are discussed.

1. Introduction

Church bells are musical instruments, closely connected to cultural heritage, as well as being structures that are exposed to severe loading conditions, which differ from region to region, as the bells are ringing. The dynamic system of a bell, its clapper, the yoke and a supporting belfry has been developed for centuries, based mainly on intuition and tradition. However, a bell can be damaged as a result of material wear, fatigue loading, material deficiencies, the clapper material, the shape and weight of the clapper, different clapper-to-bell layouts, the characteristics of the belfry, the ringing conditions, etc. It appears that the wear of bells that were equipped with clappers made from new steels 50–80 years ago is larger than the wear that was observed over the centuries. In the recent past several projects looking at the life of the bells were performed, but none of them was able to achieve adequate results due to insufficient resources and the huge complexity of the problem related to the study of bell damage [1]. That is why an EU-funded project called Maintenance and Protection of Bells (PROBELL) was begun in 2005 to investigate the damage mechanisms relating to bells and to give directions for smooth ringing, with a consequent reduction in the damage done to bells. The project consortium consisted of eight bell founders, a clapper manufacturer, three universities, the TÜV SÜD and the Church.

To study the wear-related damage at the contact point between the clapper and the bell, experimental investigations and numerical simulations of the local contact were carried out as part of the PROBELL project. Fletcher et al. [2] showed previously that the contact surface between the clapper and the bell is flattened by continuous ringing, which results in the bell having a different sound. However, they did not describe the details of the local elastic–plastic deformations of the clapper and the bell that led to the local wear. One of the objectives of the PROBELL project was to thoroughly understand and describe the causes and effects of this local wear at the clapper-to-bell contact point.

Before the full-scale finite-element model of the bell with the clapper was built, the finite-element approach was tested with the help of a simplified model. In this simplified model the clapper was replaced by a steel cylinder with a spherical tip that was dropped against a flat bronze block, which represented the wall of the bell. For the simplified model a series of cylinder-drop experiments were performed by University of Applied Sciences Kempten and the

* Corresponding author. Tel.: +386 1 4771 504; fax: +386 1 4214 074.
E-mail address: jernej.klemenc@fs.uni-lj.si (J. Klemenc).

0734-743X/$ – see front matter © 2012 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ijimpeng.2011.12.006

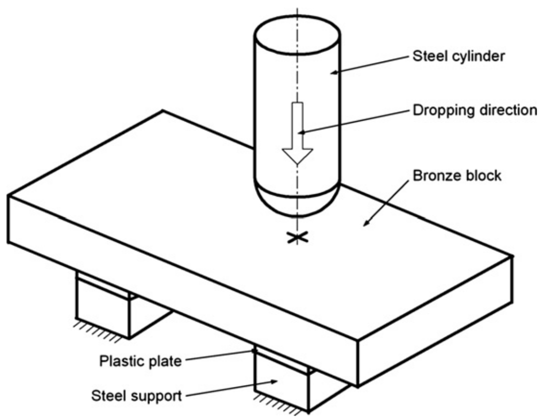

Fig. 1. Arrangement of the simplified cylinder-block models.

corresponding finite-element simulations were carried out by University of Ljubljana. A comparison of the measured and simulated results showed an acceptable agreement [3,4].

After the finite-element model of the simplified cylinder-drop test was extensively tested a full-scale finite-element model of the bell with a clapper was built. A 3D model of a 320-kg bell with a 15-kg clapper was used for the finite-element simulations. With the full-scale finite-element model the influences of different parameters on the structural behaviour of the bell and the clapper were studied. The influential parameters were as follows: the clapper material, the shape of the clapper, the clapper's radius, the relative impact velocity of the clapper, the clapper-pin support, the clapper's impact angle, the clapper's guide accuracy, the bell's sound-burp thickness and the coefficient of friction between the clapper and the bell. The results of the finite-element simulations made it possible to calculate the stress and strain distributions and identify the wear-risk zones around the clapper-to-bell contact point. By combining certain results of the finite-element calculations into the Ruiz–Chen parameter [5,6] it was also possible to identify the potential risk zones where the fretting-fatigue cracks could occur. A direct comparison between the simulated and the measured results was not as easy as in the case of the simplified model, because there was not much experimental data available relating to the accelerations of similar bells and clappers. However, using the simulations we were able to determine the same fundamental behaviour of the bell and the clapper as with the measurements, which means some parallels could still be drawn between the measurements and the simulations.

After the introductory section, the simplified model for assessing the consequences of the repetitive impacts between a spherical body made from steel and a flat body made from bronze is presented. A short discussion of the agreement between the simulations and the experimental data for the simplified model is also given. In the third section a description of the different finite-element models for simulating the repetitive clapper strokes are described. In this section the results of the finite-element simulations are also presented, compared with the experimental data, and discussed. The concluding section is followed by the acknowledgements and a list of references.

2. Simplified model for studying the repetitive cylinder-to-block strokes

2.1. Finite-element model and boundary conditions

The simplified model for assessing the consequences of the repetitive cylinder-to-block strokes is presented in Fig. 1.The dimensions of the cylinder were: $L = 270$ mm, $R = 80$ mm. The dimensions of the block were $560 \times 220 \times 50$ mm. The weight of the steel cylinder was approximately 10 kg. The bronze block weighed approximately 56 kg.

The LS-DYNA 9.70 explicit finite-element code was applied for simulating the repetitive impacts between the steel cylinder and the bronze block (see Hallquist [7], the LS-DYNA user manual [8], and Zienkiewicz and Taylor [9,10] for details). Due to the symmetry of the simplified model only half of the experimental arrangement was modelled by the finite-element model. The influence of three parameters on the dynamic behaviour of the block and the cylinder during the impact was studied: the cylinder-tip radius ($R = 50$ mm/$R = 150$ mm), the cylinder material (mild steel DIN 17102 TStE285/hard steel DIN EN 10083 C45E), the cylinder-drop height ($H = 80$ mm/$H = 100$ mm).

Two finite-element meshes that were built for the two cylinders with different tip radii are presented in Fig. 2. The cylinder, the block, the two supports of the block and the plastic plates between the block and the supports were modelled with 3D fully integrated solid elements, and the vertical guidance of the cylinder was modelled with shell elements. A series of parallel linear spring and linear damper elements were added below the two steel supports to absorb a portion of the kinetic energy during the impact – see Klemenc et al. for details [3]. They were connected to the nodes of the two steel-support elements on one side and were clamped at the other side. At the symmetry plane the symmetry boundary conditions were applied. Six automatic surface-to-surface contact entities were defined for each of the two finite-element models for the following contact pairs: cylinder-to-block, cylinder-to-guidance, block-to-plastic plate and plastic plate-to-support. The coefficient of friction in the contacts was 0.1 (Friction factors, table 4 [11]). The statistics of the two simplified finite element are listed in Table 1.

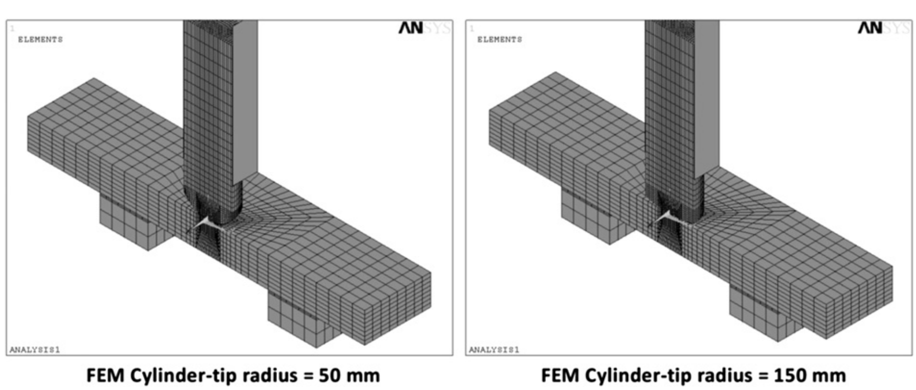

Fig. 2. Two finite-element models for the simplified cylinder-block models.