

# ram reports in applied measurement

## Improving finite element analysis accuracy using physical measurements

Marc Juwet ,  
Karel Kenis,  
Filip De Bal,  
Katholieke Hogeschool (KaHo),  
Sint-Lieven, Ghent, Belgium

### Introduction

Increasingly more engineers use computer aided analysis tools in the design process of a structure or part. One of the most commonly used methods is finite element analysis (FEA). The main application area is mechanical stress and strain analysis. Also in other engineering domains, the FEA has proven its importance. A difficulty with this and with all simulation techniques is that the analyst should be sure his model is a good representation of the real structure. For simple structures e.g. clamped cantilever beams, the FEA has proven to be accurate. The problems arise with special features like weldings, bolt connections and composite materials. How can we make sure the mathematical model of the structure behaves exactly like the original, real part or structure? The problem in some cases is that we do not have a clear idea of the behavior of the real life structure.

### Problem solutions

Two kinds of physical tests can be done to deal with this problem:

One or more important and difficult-to-model parts of the structure are built either full size or scaled. Such a part is loaded in reality and in a computer model. The computer model is adapted until the computed results correspond

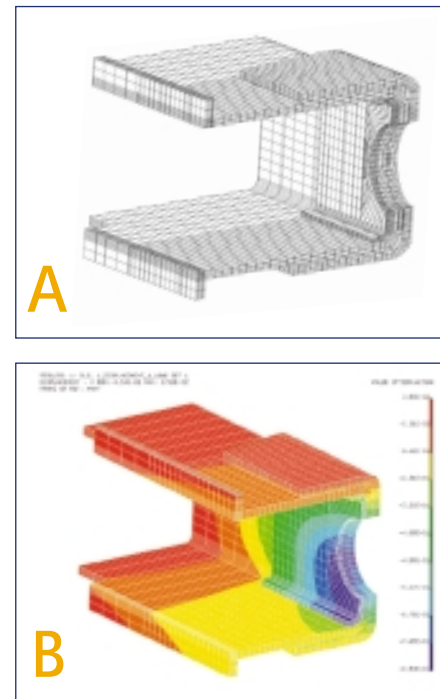
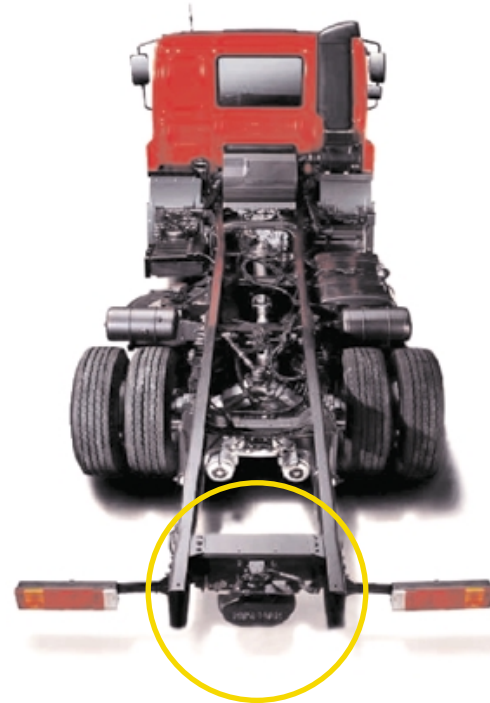


Fig. 1: Testing the rear beam of a truck using finite element analysis (FEA)  
A: FE computer model, B: The result of stress analysis

to the measured results. The modified computer model can then be used as part of a larger model that is supposed to react as the real structure as yet not built. The large computer model can be used for optimization purposes. A typical example of this type of test is the rear beam of a truck, shown in Fig. 1.

In many cases it is not feasible or not safe to apply all possible loads to a prototype of a structure or machine. One or more loads are applied and measured. Displacements and strains are compared to computed values. It is supposed that the physical structure will react as the computer model under other load cases.

### Finite Element Analysis

It is not the goal of this article to explain finite element analysis, but the following paragraph gives a summarized explanation of the method. The finite element analysis is an approximate solution method for a range of equations. It is inherent to this method that the results are only an approach to the behavior of the real life structure.

FEA divides the volume to be calculated into small volumes (elements) with a relatively simple shape. Depending on the type of structure to be examined, the elements are given different properties. These elements are intercon-

nected in a discrete number of points (nodes). In FEA, the external forces acting on the structure cluch at the nodes.

For each element a force balance can be stated which results in a set of algebraic equations. After introducing the boundary conditions the set of equations can be solved resulting in the displacement of the nodes. From that, the strains, deformations and reaction forces can be derived. It is clear that the approximation usually becomes better when more elements are used. In finite elements jargon, that is called mesh refinement.

Up to now the method requires heavy computer infrastructure because of the setup and the solving of the equations. Computation times vary from minutes to hours and it is not unusual to have problems requiring days of computer times. Therefore the analyst always tries to minimize the number of nodes, and thus the number of equations, in the model. Methods to achieve this goal are the use of symmetry and the restriction of the model by modeling only part of the structure. Another method is to do only mesh refinement in the zones where high strain concentration is expected.

Without going into detail the following example should make things clear. Assume we want to calculate stresses and strains in a cantilever beam. Fig. 2 illustrates the problem. The plate is fixed in a wall at one side, while a force  $F$  is acting on its opposite side. Fig. 3 gives a finite element model. The elements are clearly visible just like the force and the boundary conditions. After calculation, the results in Fig. 4 are ready for interpretation. Each color corresponds to a range of strain, according to the legend. Note that this example can be solved without the finite element analysis, but it gives a good idea of the method of working and of some approximations of FEA: limited numbers of elements, load applied in 6 nodes, one face completely fixed.

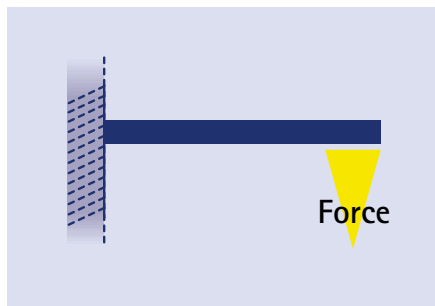


Fig. 2: Stresses and strains in a cantilever beam fixed at one side with forces acting on it

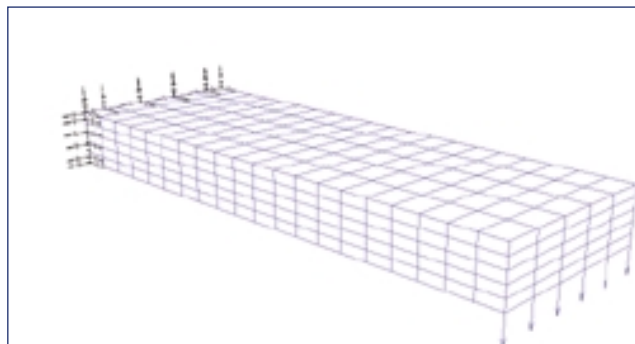


Fig. 3: FE model of a cantilever beam

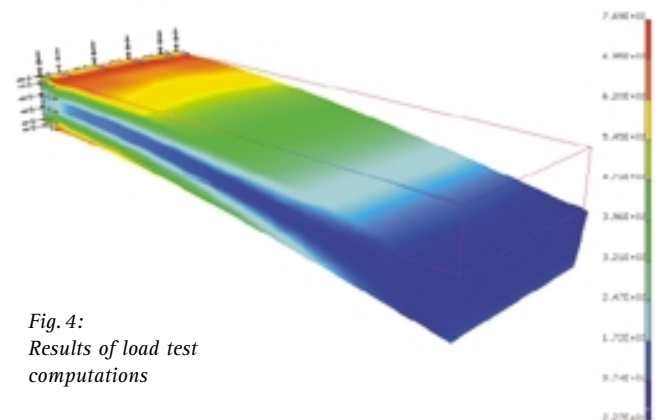


Fig. 4: Results of load test computations

### Testing equipment

KaHo Sint-Lieven has built up some experience in comparing FEA models and real structures. A wide range of material testing machines is available going from little analog measurement amplifiers to a computer controlled state-of-the-art material testing machine, with a capacity of 250kN. The restriction of all the available machines is that it is impossible to test a structure under moving load.

Therefore a new testing machine with a capacity of 10kN is developed in cooperation with HBM Benelux. The design and the construction of the machine has been carried out at KaHo. The testing machine (referred to as "press" in the rest of this article), has a table size of 1,200x600 mm. The part to be examined can be positioned anywhere on this plateau. The load, applied by an electromechanical spindle, can be controlled using force or displacement feedback and has a maximum of 10kN. The position of the spindle can be anywhere within an area of 600x800 mm.



Fig. 5: Measuring equipment and test stand

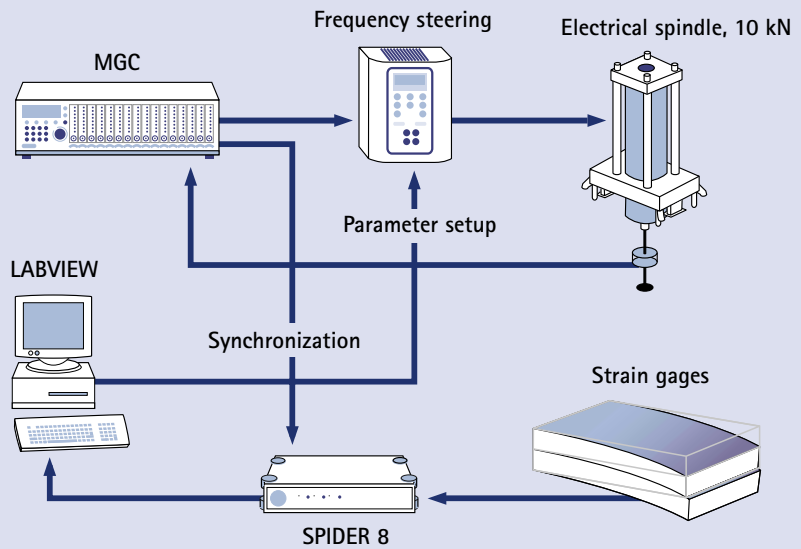


Fig. 6: Schematic overview of the line-up

The frame of the machine has been calculated using FEA, so that, in the load range of the machine, it reacts almost infinitely stiff. Measuring on the machine, the deformation of its frame can thus be neglected. Fig. 5 gives an overall view of the measuring equipment.

The measuring part of the press is completely built with HBM material. We use a Spider8 for the simultaneous acquisition of 8 channels. In most cases channels are connected with strain gage setups. Besides these 8 measuring channels, we need one channel for the steering of the spindle: the force (or the position in a position-controlled setup) should be measured and must be converted into an analog signal. This analog channel is needed for the frequency steering of the spindle. Since the Spider has no analog output an MGC amplifier (MGC is the predecessor of the MGCplus) with one channel (AP01) giving a signal of 0...10V is used. Yet one problem had to be conquered: there wasn't a link between the signals measured with the Spider8 and the force (assume that it's force controlled) measured by the MGC. This link is important because we want to see the reaction of the structure to the applied load,

also for time varying loads. To beat this problem the MGC is linked with one input channel of the Spider, which is capable of reading analog inputs. The force transducer is thus one of the eight channels and an analog measuring of the applied force and the reaction of the structure is made very simple. A Labview program controls the whole press. The processing of the measuring data coming from the Spider8 is either done with home made software or with the HBM catman, package. Fig. 6 is a schematic overview of the line-up.

### Example

The difficulties in setting up a test depend on the results to be measured, the boundary conditions of the structure under investigation and the type of load applied.

The problem described in this article comes from a manufacturer of industrial equipment. A little crane (Fig. 7) uses a rubber track for its locomotion. This track contains metal parts used for the propulsion of the track by the sprocket. It is this metal part which is the subject of our study. The problems with this part

(we will refer to it as the 'bar') arise when the crane is hitting an obstacle (Fig. 8). The bar is then subject to a simple bending load.

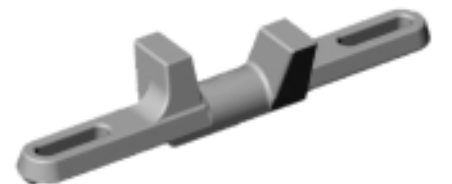


Fig. 7: Crane rubber track with bar

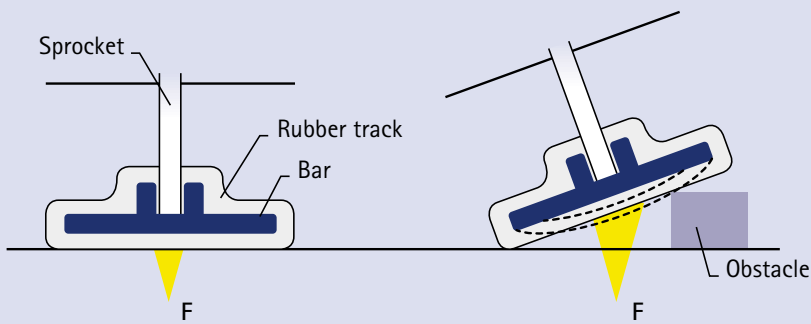


Fig. 8: Bar subject to simple bending load

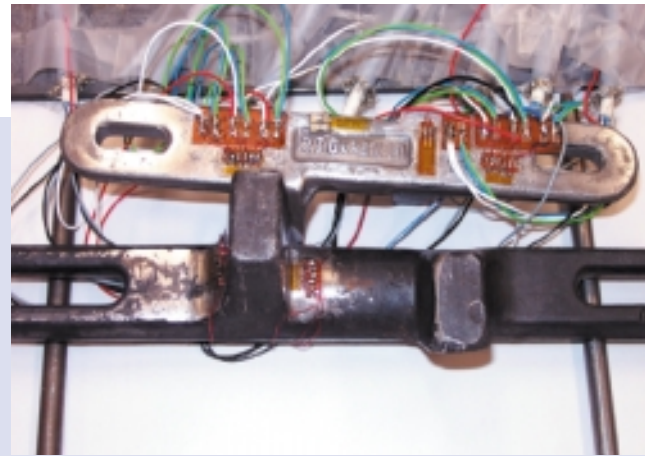


Fig. 9: Rubber track bar with attached strain gages

### A. Physical test

The part is supported on two round bars. Strains are measured at 3 different places, using RY81 and LY11 strain gages. In the fillets RY81 rosettes are used because the principals are not known. Strain gages give an average strain of the zone on which they are attached. Therefore we use very small rosettes (RY81-06/120) in the fillets where strain is expected to vary substantially. Using other technology such as Electronic Speckle Pattern Interferometry (ESPI) for high strain gradients is possible, but quite expensive. The Spider8 doesn't allow us to work with quarter bridges, so we need to build half bridges with passive gages attached to another bar made of the same alloy. In Fig. 9 the bar with strain gages attached can be seen.

### B. FEA model

The part is modeled in FEA software using 3,256 tetrahedral parabolic elements and 6,962 nodes (Fig. 10). This results in a set of 20,886 equations.

The supports are modeled as line contacts, nodes on this line are supported. Reaction forces in these nodes will be very high and corresponding contact stresses will not be realistic. The force is applied using a cylindrical bar, both in the physical test and the computer model. It takes about half an hour for the computer to calculate strains and the stresses derived from that. The calculated Von Mises stress can be found in Fig. 11.

### Conclusion

Comparing the results of the physical measurements and the FEA calculation, one can state that in this case the FEA model is a good representation of the real life behavior of the bar. Further tests are not necessary and the optimization of the bar design can be done with FEA. This statement cannot be generalized: in many cases it is necessary to update the FEA model to achieve a correct behavior.

We want to end this article by stating that FEA calculations and physical tests are complementary: initial FEA calculations are showing us interesting places to attach strain gages while measurements are giving us confidence that the FEA model used is reliable.

Even in very simple cases, where the FEA method has proven to be accurate, physical measurements can help to convince FEA rookies that their model is good.

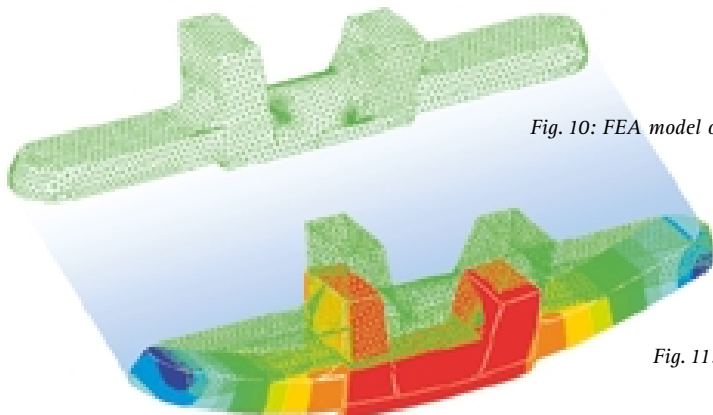


Fig. 10: FEA model of the bar

Fig. 11: Calculated Von Mises stress