Development Process at NABI Rt. in 2000

Ervin Kerekes
Structural Analyzer
NABI Rt.
ekerekes@vnet.hu

NABI Rt. is a Budapest based dynamically growing bus manufacturer. The NABI Group produces about 1000 buses a year, and it not indifferent what kind of products we produce, especially for the US market. The bus industry faces a unique challenge, the product must have a long operating life, and at the same time it is produced in small series. In case of cars the industry has plenty of opportunities to carry out experiments and real tests, because the costs are absorbed by the huge market. This possibility is not available for bus manufacturers. That is why we use lots of simulations for establishing the preferred versions, which can be manufactured and which might comply with the test with a high probability.

The classical, quasi-static designing process

Similarly to most of the domestic (and international) companies NABI used to start the strength calculation of the internally designed and manufactured vehicles with a quasi-static procedure using linear models and specifying limit values. The principle is illustrated in Fig. 1.

The model was established with the finite element software ANSYS based on the Pro ENGINEER drawings. The ANSYS/Pro ENGINEER link is rather good, for this reason the most simple procedure includes the importing of 3D CAD models, from which so called 3D SOLID models can be produced easily. However, the modeling of the bus frameworks is much more complicated task. For this reason the much simpler shell and bar models are being used for preparing frame models for saving efforts and time.

The loads acting on the structure are very hard to define. The dynamic loads of the bus caused by the road, by the operation of various structural parts (e.g. engine), by the intervention of the driver (e.g. steering), etc., could be measured long time ago, but no sufficient calculating methods or sophisticated computers were available to analyze these loads in a comprehensive way. For this reason a calculation method based on fictive linear loads have been established. The loads
Considered in the process include: gravitation load, acceleration, deceleration, crosswise acceleration, lifting of the respective sole points of wheels, roof load, towing, etc. Various limit values were identified for the loads based on experience. If the structure could not meet the specified limit values, then modifications were made, and the process was repeated all over again.

The calculation were verified by means of tests, which were often more close to the reality than the calculations, because in practice the dynamic signals coming from the real loads could be considered much earlier.

Fig 1: Flow chart of a classical quasi-static dimensioning procedure

*Dynamic dimensioning procedure with the use of own software*

About one year ago NABI introduced a novel calculating procedure, which was much closer to reality. The prerequisite for this introduction was the development of several life cycle calculating methods [6.], [7.], [8.] by utilizing the possibilities of ANSYS. The basis of the methods was derived from the German standard DIN15018 [2.] and the British Standard BS5400 [1.] (Fig. 2)
With the help of these methods the calculation process is modified as follows. Based on the drawing prepared with Pro ENGINEER we establish a finite element model in the usual way. Although the loads are still quasi-static, but certain stress distributions caused by various amplitude/mean value loads and the relevant cycle number can already been matched with some limitations. With the help of such information we can calculate damage (probability of break) according to the given standard. In addition to that we can also consider the weakening effect of the weld seams!

Naturally, the calculations are also verified by means of tests, which are now much closer to the reality.

A life cycle planning software called FEMFAT [4] emerged in August this year, and a dynamic software SIMPACK was released early in November. With these software items the dimensioning procedure was supplemented to form a unified planning scheme (Fig. 3).

The initial step of the former process (i.e. model generation from CAD drawing) basically remained the same. The only difference is, that the bar model is
no longer sufficient for the life cycle estimate, at least a shell model is necessary. (In the model we can consider also the welded joints for the life cycle estimate to be made later on.) After completing the model we have two options.

One of the possibilities is to execute a dynamic simulation within ANSYS, which means that we can introduce signals to the structure with various time frames, and we obtain the response stress distribution for each moment. We can use this result to link ANSYS to the FEMFAT TRANSMAX module. As a result we obtain a probability variable for the life span for the given load. 

The other option is to conduct a modal analysis only in ANSYS, and the resulting modal characteristics are exported to SIMPACK. This software is suitable for executing dynamic analysis of Multi-Body Systems. It is possible to use various wheel modules for executing realistic maneuvers, and it is also possible to introduce the finite element models. With the help of SIMPACK it is possible to simulate longer signals, and to monitor the influence of various parameters. The result can be retransformed either to ANSYS, where the actual stresses could be obtained, or to the FEMFAT ChannelMAX. From the results of the ANSYS modal analysis and the SIMPACK simulation we can carry out a life span estimate probably with a much smaller set of data. This method is used also by several major West European vehicle manufacturers [5], [6].

Fig. 3: The present development process
Practical examples for the new methods

Testing T joints with the help of standards BS5400 and DIN15018

I would like to show the usability of various methods first with the help of a T joint. The experiments were carried out by AUTÓKUT [3.], and the verification calculations were made by NABI Rt. The photograph of the experimental jig can be seen in Fig. 4. The stress calculated with BS5400 for the bar elements is shown in Fig. 5 and 6. The same, calculated for shell elements according to the German Standard DIN15018, is shown in Fig. 7 and 8.

![Experimental testing of bent T joint](image)

We used bar element corresponding to the classical mechanics for the calculations. For this reason the mesh fineness has not much influence on the determination of maximum values as can be seen in the stress distribution as well as in the damage picture.
Fig. 5: Stress distribution calculated for the bar elements

Fig. 6: Damage distribution calculated with BS5400
It is interesting to note that the stresses obtained in the vicinity of the weld exceed the ultimate tensile strength as a result of the stress considered in a linear way. But the damage values are in very good correlation with the test results. Even the locations of breakage coincides in case of finer mesh.

Fig. 7: Stress distribution with various mesh densities

Fig. 8: Result of the calculation of damage factor
Analyzing the longitudinal girder of the engine suspension frame by means of FEMFAT

The second example illustrates the usefulness of the FEMFAT software with a simple well controlled test. The test equipment and the broken specimen can be seen in Fig. 9. The tests were carried out by the Faculty of Bridges and Structures at BME [9]. The aim of the measurement was to investigate the effect of the small lug welded to the longitudinal girder of engine suspension frame in case of various loads. The results of FEMFAT are shown in Fig. 10 and 11.

Fig. 9: The test equipment and the location of breakage

The results of testing are in well correlation with those of calculations. The location of breakage is clearly the same, while the calculated life span is within the standard deviation. A unique feature of the calculation is that the weakening effect of the welds is also considered within a structure consisting of relatively simple bodies.
Fig. 10: Damage distribution caused by a given dynamic load

Fig. 11: Damage distribution within the structure
Further examples on actual vehicle structure

Testing the wheel drum

The next example shows a wheel drum environment with two different designs. For this purpose we constructed the whole bus as a bar model, with the examined parts made from shell. Fig. 12 illustrates the condition when the bus passes an obstacle during simulation. (Naturally, with ample magnification). For a better understanding the original equilibrium state is also plotted.

![Fig. 12: Result of the dynamic simulation of articulated bus](image)

The objective of the testing was to select the best method in terms of life span of the bus. Many possibilities occurred during the calculation naturally, but only the least favorable solution (Fig. 13) and a much more favorable solution (Fig. 14) are shown here for saving time.
Fig. 13: Original solution

Fig. 14: Redesigned more favorable solution
This solution resulted in a probability of breakage ten times less than that of the original design in a dangerous environment. At the same time certain improvements could be reached also at other locations.

Testing the joint of the CH frame radius link

In this case the calculation was started with a global, simplified model, similarly to the above example. Then a more comprehensive investigation was carried out with a more detailed model produced with the "submodel" technique (Fig. 15). The probability of breakage was also determined with FEMFAT, as shown in Fig. 16.

Fig. 15: "Submodel" of the global CH frame and the link environment

This is a welded structure, for this reason we have to proceed very carefully in determining the locations of welds. The stress accumulating effect of the weld can be seen very well in the picture, and it is also evident here that the design is rather inadequate. The small picture at the bottom right illustrates the damage distribution printed for the selected welds.
Fig. 16: Damage distribution in the „submodel”

Summary

I hope I was able to illustrate the development taking place at NABI Rt. during the past several years by showing the above examples. With the introduction of the "virtual prototype production" in the development process it is accepted that we could maintain a competitive edge at the market with a much higher probability by offering much better, high standard and less expensive products.

Bibliography


[6]. Kerekes E., Szabó B., B. Unger, - Prestressed bolt joint under dynamical load

[7]. Kerekes E., Petrovics J. - New Features in ANSYS developed by NABI FE Group –CADFEM Users’ Meeting 2000, Friedrichshafen


[9]. Kerekes E - Fatigue examination of weld seams – VSDIA 2000, Budapest