METALLIC GASKET MODELING USING MSC/PATRAN ADVANCED FEA

Michael de Jesus Fernandes
Sabó Indústria e Comércio Ltda
E-mail :eng.juntas@sabo.com.br
Phone : 0055 011 36772663
Fax : 0055 011 36772639
São Paulo - Brasil

Abstract

This work presents the modeling methodology for multi-layer and single-layer-steel gaskets, MLS and SLS. Among the approached themes discussed are some concepts regarding metallic gaskets, the design methodology itself, main considerations and assumptions, as well as the most common problems in regards to the input data acquisition, and in the convergence process. The entire approach is carried out considering a fictitious problem example.

INTRODUCTION

Nowadays there are many types of sealing technologies. Among them are cork with rubber, rubber molded by compression, injected rubber, paper, soft material reinforced with metal, and most recently, metallic and liquid gaskets. Each technique has specific applications, which depends on the work temperature, chemical resistance, cost, work pressure, etc. This diversity has been created due to the growing demands of emissions and weight reduction, life and gasket increased performance, as well as the system as a whole.

So, the metallic gaskets have been detached, mainly due to their durability, versatility, and stability. Most of them consist of steel sheets with micrometric elastomeric coating. These gaskets are usually embossed to allow a local increase of the sealing pressure and to follow eventual movements of the mating surfaces once the beads begin to work as elastic elements. In this case, it can be said that the sealability depends on the gasket embossement yield, rubber coating thickness, the effect of mating surface stiffnesses, variation in fastening load due to thermal expansion, micromovements, creep relaxation, surface finish, and numerous other factors.

For this reason numeric simulation tools are becoming indispensable in the development of the gaskets, mainly between those constituted by metal and rubber. Because the reliability degree, and design lead-time reduction demanded by technological tendencies Besides the fact that today great concern is taken to the component influence in the system, in other words, a configuration of the gasket could even have a better performance and still support more rigorous work conditions, however it is considered unacceptable because a possible distortion in mating surfaces.

Therefore, this work presents an alternative finite element approach to the modelling of an exhaust manifold gasket in a fictitious problem.

PROBLEM DEFINITION

In some cases it is necessary to begin the development of the gasket without the complete definition of the mating surfaces, consideration of the geometry, acceptable out of flatness, fastening load and so on. Among them, the most critical information is the geometry because of the fact that the gasket loads distribution closely depends on it´s stiffness. When this happens, it is necessary to simplify the geometry and gauge the model afterward.

The model shown in the figure below presents a situation where the stiffness of the exhaust manifold is coarsely approched by the displayed solid and the engine head is described as a rigid surface.

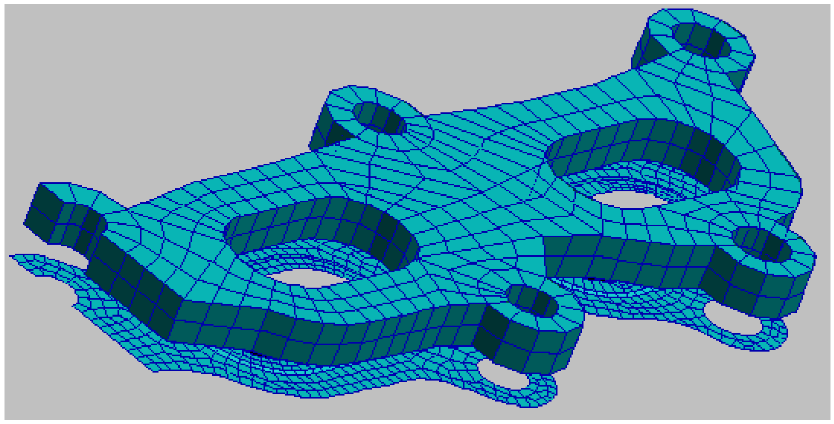

Figure 1

Concerning to materials data definition, it is necessary to evaluate them with care, once one of the objectives of the analysis is verify the existence of regions where there is stiffness loss caused by plasticity phenomenon. In MSC/PATRAN Advanced FEA, it is possible to approach the material behavior considering an isotropic hardening, creating tables called "fields". Moreover, depending on the component application, the temperature effect can be significant because the material can suffer high stiffness loss and consequent reduction of capability to follow the flange movements, This is a result of stored energy that is lost with the stress relaxation produced by temperature.

Therefore, to modelling like this, we start by determining the Young´s Module and Tangent Module reduction with temperature. The material is defined through the true stress vs strain curve, in order to consider the effect of the temperature. So, the material behavior is separated in two stages, elastic and plastic. In the case of elastic, it is necessary just to create a table to identify the reduction of the Young´s module with the temperature. In the plastic case, a three-dimensional matrix is generated with stress, real plastic deformation and temperature data. The illustration 2 presents a simplified true stress vs plastic deformation graph for the material used in each analyses.

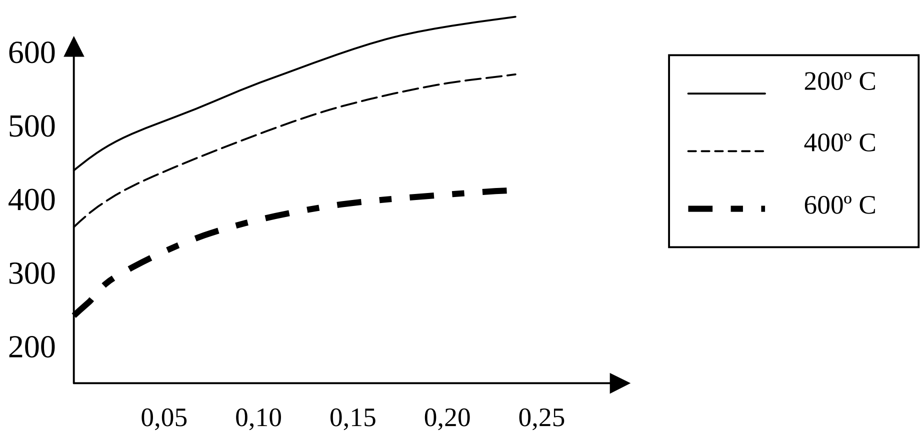

Figure 2

Then a thermal analysis is made to identify the distribution of the temperature in the component to associate the mechanical properties of the steel with its temperature distribution. Figure 3 presents the subject being analysed, where only prescribed temperatures in the exhaust manifold internal and external area were considered, as the problem boundary conditions.

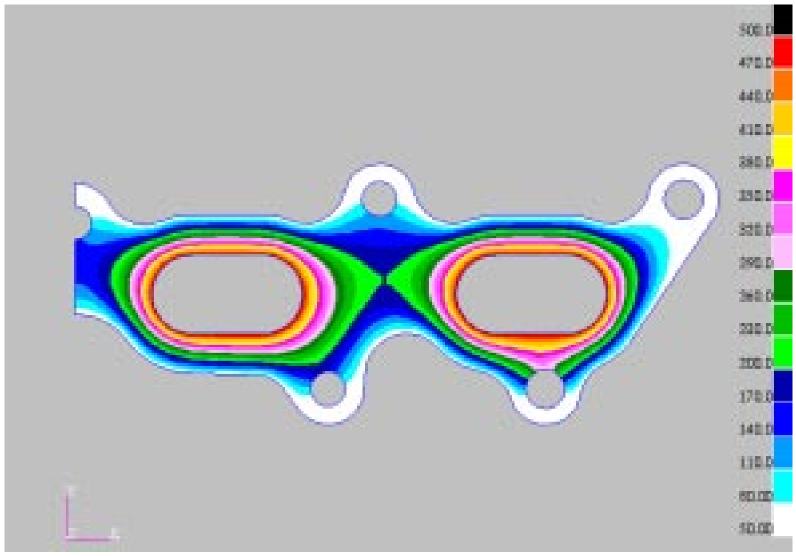

Figure 3

With this data, a non-linear static analysis is executed to evaluate the distribution of the loads in the piece. To do this**,** the solid is projected toward the gasket to compress it. Observe the results in the illustration 4 below:

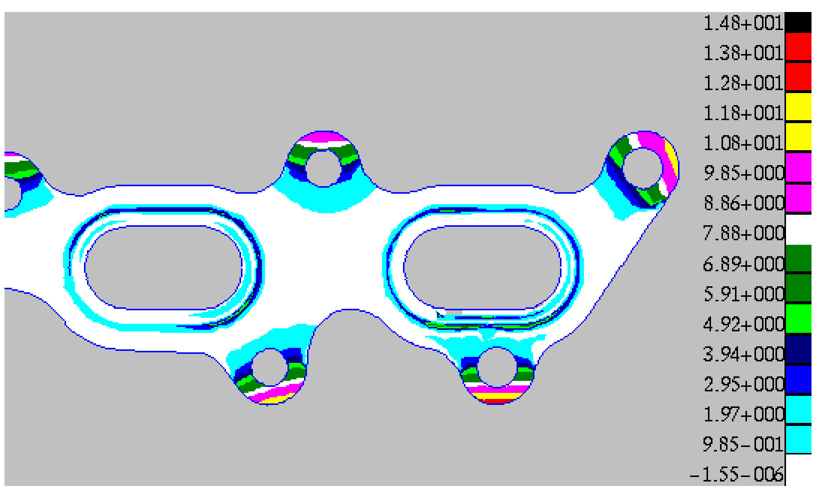

Figure 4

The objective of the global model is therefore to observe the global performance of the component identified by the loads and stresss distribution, temperatures and contact pressures. Thus, it is possible to identify the critical areas under the sealing point of view. In other words, the ones that have a smaller applied load to provide the sealing, as well as the areas submited to the most rigorous conditions, where the plasticity phenomenon can be witnessed, which is deeply undesirable. Afterthat, the creation of the detail model can take place based on this information.

In agreement with the gasket type and with exact point of interest, bi-dimensional analysis are executed with axisymmetric elements and/or plane strain elements, which present good results, when the only point of interest is the contact pressure.

Figure 5 presents a model of this nature, in which, the superior and inferior critical conditions of load, that allows for the stress, strains, and contact pressures evaluation were applied. These should be analyzed together with the information related to surface conditions, like acceptable flatness and roughness, for each project.

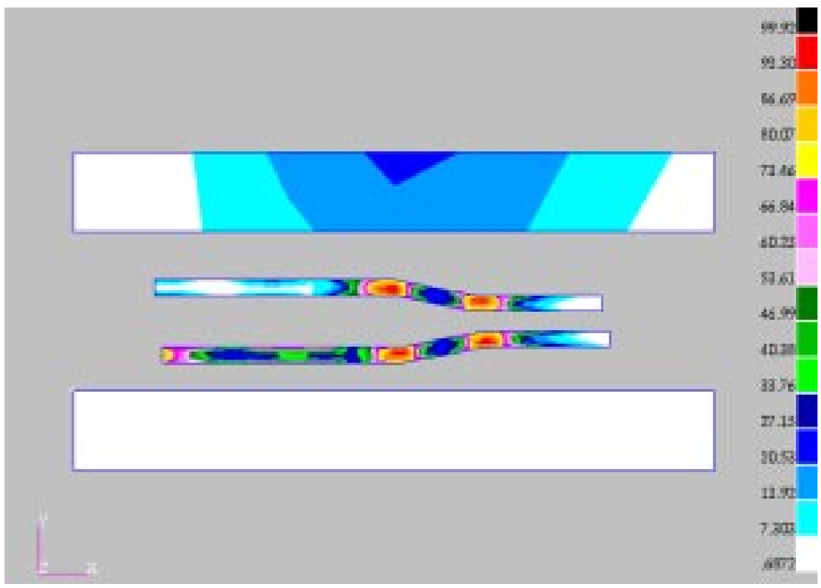

Figure 5

After a certain number of iteration, it is possible to reach the final geometry of the sealing surface, even it is possible predict situations where misalignment between the embosses can occurs, as shown above. These successive evaluations are carried out based on sudies that consider the integrated use of Finite Element Method and Design of Experiments. For example, through effect graph analysis it is feasible to estimate the influence of each geometric entity in the gasket performance.

DISCUSSION

This kind of model generally presents great difficulties in the convergence process. This is because the gasket thickness is extremely reduced, when compared to other part dimensions, which leads to very small strains. The use of a PC platform restricts the number of significant digits, that interferes directly with the analysis result, once the cut errors accumulate themselves, creating difficulties to reach the equilibrium and, naturally, the solution to the problem. Furthermore, difficulties are imposed by the non-linearities inherent to uncertain contact problem, which determines the existence of a variable load vector. Another detail refers to the geometry approach. The use of contact elements determines the choice of linear elements, to avoid mistakes related to the load application in the intermediate nodes of the element edges. As the elements have isoparametric formulation, the geometry is also defined by linear interpolation functions. As an unavoidable consequence, there exist sharp corners linking elements, in curvilinear surfaces definition.

In some cases, the convergence problems associated with the contact can be minimized using rigid surfaces to define the flange. This is because MSC/PATRAN Advanced FEA creates a Bezier surface internally in order to soften the contact. However, in metallic gasket modelling, this technique doesn't perfectly represent the physical phenomenon. Therefore, as described previously, the sum of these effects characterizes a high cost of convergence process.

The only favorable point is that most of the time, metallic gaskets are rubber coated. For this reason, a certain contact type is usually used once this methodology uses "penalties" instead of Lagrange multipliers. Consequently the convergence is reached faster, because it becomes easier to achieve equilibrium.

In addition caution with mesh must be taken. Figure 6 shows differences in contact pressure distribution, because of element size. For example, at the region near the right bottom screw, it is possible to see a second pressure line, as in some other areas, specifically because of mesh refinement.

Notice that only static non-linear analysis took place. To execute a dynamic analysis it would be necessary to model the exhaust manifold and the engine head with a mesh sufficiently refined to evaluate the natural frequencies and to make a harmonic overlap, because the time domain calculus would be able to simulate just a few seconds. This whole process is slow, accumulates a series of assumptions, and probably doesn't supply much more significant information than the ones obtained with static models.

Thus, the maximum and minimum stresses at critical points, referring to the critical loading conditions, can be evaluated with static analysis. Once you have this data, an alternating stress is admitted as being half of the difference of the maximum and minimum value. If this value is small, the fatigue limit stress won't be reached. So, in this case, these stresses don´t produce cyclical plastic deformations, and the formation and propagation process of cracks doesn't occur, inhibiting the fatigue phenomenon completely.

On the other hand, there are configurations subject to the action of high temperatures, as in this example, where the temperature influence on the material is critical. Then, it is necessary to evaluate a creep analysis. This process is not part of the scope of this work.

Concerning the model´s approval criteria, the validation can be made through pressure sensitive film impressions. Figure 6 shows an impression obtained in a prototype. Based on this impression the model is gauged regarding the gasket´s initial conditions ("cold assembly" of the flanges). After gauging the cold model using the pressure sensitive film impression, it is possible to consider that the work conditions approach has an acceptable error.

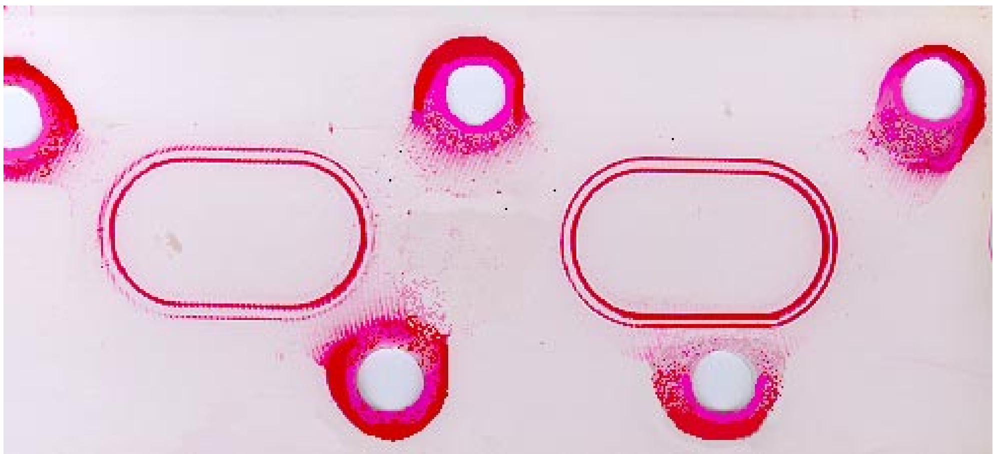

Figure 6

CONCLUSION

To succeed in the metallic gaskets design, it is mandatory to take a consistent correlation between the field conditions and the numeric simulation, and to know the influence of each geometric entity in the bead profile. Studies are necessary to quantify the smallest contact pressure that guarantees the sealing, where the superficial roughness of the flanges and the existence of elastomeric coating are considered, as well as the thickness of it.

Once the sealing pressure is known, it is enough to look for the profile of a more appropriate embossment in order to reach it. More important is the choice of the material, that better supports the operating conditions, because it should avoid the loss of excessive stiffness with the temperature.

In some cases, it is possible to reach good gaskets design results, just by performing static analisis.

REFERENCES

[1] M. Robinson, G. Whitham, T. Tronel and S. Schulze, "Application of Advanced Finite Element Techniques to the Design and Development of Automotive Gaskets". SAE Technical Paper 960218.

[2] G. Novak, G. Schwerdtfeger, and E. Widder, "Sealing Performance of an Embossed Rubber Coated Metal Gasket". Technical Paper 950328.

[3] L. Oricchio, A. Garcia, and J. Reichert, "The use of DOE for PTFE Oil Seals Optimization". Technical Paper 982924.

[4] E. Widder, M. Sadowski, and G. Novak, "Gasketed Joint Analysis Including Flange Bending Effects". Techinical Paper 930120.

[5] T. Ishigaki, J. Kitagawa, and A. Tanaka, "New Evaluation Method of Metal Head Gasket". Technical Paper 930122.