A Gasket Element Representation of An Intumescent Mat Material

J. Webb[*], R Richmond[*], J. Choi[**], T. Dalrymple[**]

[*] Delphi Powertrain Systems
[**] ABAQUS, Inc.

Abstract: An intumescent mat is a very popular material for mounting a ceramic catalyst inside the steel can of an automotive catalytic converter. The major roles of this mat material in a converter are providing a tight seal between catalyst and can, and holding the catalyst without breakage during assemblage and operation stages. Choosing the right mat material is critical for the performance and durability of the catalytic converter. Numerical simulation of the mat material and catalytic converter system provides a good tool for evaluating the mat material performance, and the accurate material modeling is essential. A mat material has a unique response to thermal loading. This unique irreversible thermal expansion during the first thermal loading cycle is caused by vermiculite particles inside of the mat material. Furthermore, the thermal expansion is occurring mainly in the thickness direction and expansion in the in-plane direction is minimal. The mechanical response of the mat can be assumed to be transversely isotropic, and very compressible so that the Poisson's ratio is assumed to be zero. Two representative tests for a mat material, the relative expansion and displacement controlled tests, validate these assumptions. These experimental observations indicate that the mechanical stress – strain relation is loading rate dependent and the thermal expansion is related to pressure, temperature, thermal loading cycle number, and the maximum temperature reached through the first thermal loading cycle. Delphi Automotive and ABAQUS, Inc. developed a gasket element representation with an additive decomposition of the total strain into the mechanical and thermal strains. The temperature and field variable dependent property definition capability of ABAQUS was used for this purpose. A mechanical behavior was defined by the multiple pressure – closure curves for each loading rate in the loading and unloading directions. The temperature and field variable dependent orthotropic thermal expansion coefficients enable the accurate prediction of the thermal behavior of a mat. The developed gasket element representation of a mat material was verified by the comparison with test results, and the applicability to the full 3D model was also shown.

Keywords: Automotive, Catalytic Converter, Intumescent mat, Gasket element

1. Introduction

An intumescent mat material, which is used as a mounting material of a catalytic converter system, has a unique response to thermal loading. This unique irreversible thermal expansion during the first thermal loading cycle is caused by vermiculite particles inside of the mat material. Furthermore, the thermal expansion is occurring mainly in the thickness direction and expansion in the in-plane direction is minimal. The mechanical response of the mat can be assumed to be transversely isotropic, and very compressible so that the Poisson's ratio is assumed to be zero.

A number of research efforts have been focused on developing a mathematical or empirical model for this uniquely behaving intumescent mat material from 1990s. One of the earliest researchers was DKS Chen (Chen, 1990). He used the gap element in the ABAQUS software to analyze the canning process of a catalytic converter. His approach was limited to one–dimensional analysis in the thickness direction at room temperature. Muju *et al.* (Muju *et al.*, 1998) used a hyperfoam material definition by using different stress – strain curves at different temperature. He could analyze the canning process and thermal cycle of operation. However, the isotropic thermal expansion leads to inaccurate results. The physical intumescent mat material expands mainly in the thickness direction, and the expansion in the in-plane direction is minimal. This issue was resolved by Fuehne *et al.* (Fuehne *et al*., 2000) and Taylor *et al.* (Taylor *et al*., 2000) by employing a modified isotropic hyperelastic formulation, which is based on the total strain decomposition into three components, mechanical, thermal, and chemical strains (Kim *et al.*, 2002). They developed an ABAQUS User Material (UMAT) subroutine to implement their theories, and it was successfully applied to simulating physical catalytic converter problems. The usability of multiple layer elements through thickness direction and the material stability are still issues to be resolved.

This work outlines a simple and reliable empirical gasket element representation of an intumescent mat material for the assemblage process and operation condition analyses. The proposed model reproduced test data accurately and efficiently. The gasket element in ABAQUS was chosen for several reasons. First, the gasket element uses a conventional continuum mesh with the thickness direction determined automatically by the node connectivity for 2D and 3D problems. This leads to the ease of mesh generation. Second, the intumescent mat material shows transversely isotropic properties, which is perfect for the gasket elements. The gasket element properties in the thickness and in-plane directions can be defined separately. And the gasket elements allow defining the thickness direction properties by direct definition of the pressure – closure curves with several sets of loading and unloading paths. It is straight forward to convert the test data to the corresponding pressure – closure curves.

The key idea of this development is that the mechanical and thermal behaviors of an intumescent mat material can be decomposed additively. The mechanical and thermal responses are calculated separately, and added together for the total response. The mechanical part defines the room temperature behavior, which depends on the loading rate. The thermal part governs both the pure thermal expansion (recoverable) and the chemical expansion (irreversible) together. The thermal strain of this work covers the thermal (recoverable) and chemical (irreversible) strains of Kim *et al.* (Kim *et al.*, 2002).

The developed gasket element representation was applied to a simple model for correlation with test data, and was also applied to a real 3D converter model. The developed gasket model showed an excellent correlation with test data, and it is applicable to large and complex 3D finite element models without any convergence difficulties and provides physically reasonable results.

The observations on test data will be described in Chapter 2, and the details of the ABAQUS model development will be discussed in Chapter 3. Chapter 4 will illustrate the simple model analyses and the correlation with test results, and Chapter 5 will demonstrate the 3D model analysis. And this paper will end with some concluding remarks.

2. Test data observation

The test data used in this paper are shown in Figure 1 and Figure 2. The data shown were publicly published in papers (Kim *et al.*, 2002). The authors digitized data from those papers for use in this work.

The relative expansion test (RET) data is shown in Figure 1. The RET test procedure is as follows;

- Prepare a number of specimens with the same size.
- Put a known weight, which correspond to the needed pressure, on top of a specimen.
- Start monitoring the thickness of a specimen.
- Increase temperature at a constant rate and keep monitoring a thickness.
- Once the temperature reaches its pre-determined peak value, decrease temperature until it reaches room temperature.

The temperature and pressure dependent behavior of an intumescent mat material is observed.

Figure 2 shows the displacement controlled test (DCT) data at room temperature. The mechanical behavior of a mat material at room temperature is dependent upon the loading rate.

It is very clear that the developed material model has to take care of the effects of the loading rate (from DCT data), the temperature and pressure (from RET data).

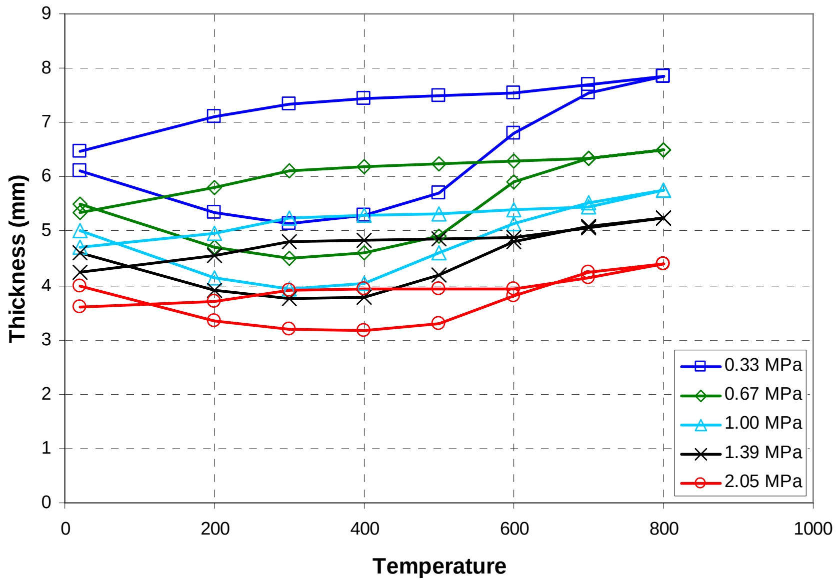


Figure 1. Relative Expansion Test (RET) data.

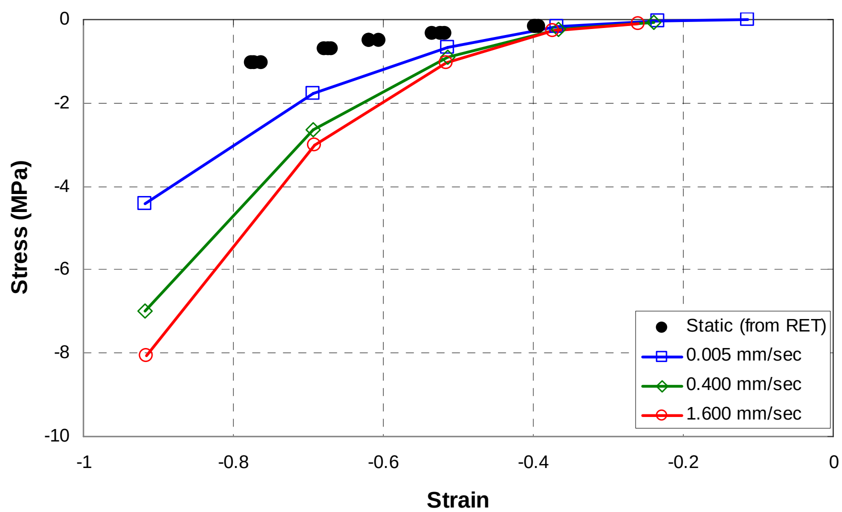


Figure 2. Displacement Controlled Test (DCT) data.

3. Developed gasket element model

In this chapter, the developed gasket element model for an intumescent mat material to reproduce the RET and DCT test data will be described in detail.

3.1 Basic assumptions

The gasket model was developed under the assumption that the total strain can be additively decomposed into mutually independent pure mechanical and pure thermal strains as;

$$\varepsilon_{tot} = \varepsilon_m + \varepsilon_{th} \quad (1)$$

where ε_m and ε_{th} are the mechanical and thermal strains, respectively. The pure thermal strain of this work includes both the recoverable thermal expansion and irreversible chemical expansion of a mat material.

The room temperature strains shown in Figure 2 are assumed to be pure mechanical strains (ε_m). The stress – strain relations shown in Figure 2 can be converted to the loading rate dependent pressure – closure curves of gasket elements.

The RET data shown in Figure 1 can be transformed into temperature – strain (ε_{tot}, total strain) relation as shown in Figure 3 by taking the original thickness of the mat material (10 mm for the mat used). The pure thermal strains can be obtained from the total strain shown in Figure 3 and Equation 1, and the resulting temperature – thermal strain relation is shown in Figure 4. The temperature dependent thermal expansion coefficients are back calculated from these temperature dependent thermal strains. Another assumption for the thermal strains is that this thermal expansion occurs only in the mat's thickness direction, not in the in-plane direction. The thermal strain will be calculated from the orthotropic thermal expansion coefficients.

The thermal strain shown in Figure 4 has two different paths; (i) the 1st heat-up and (ii) the 1st cool-down and the 2nd or more heat-up/cool-down cycles. The 1st heat-up path was modeled as shown in Figure 5, and the 1st cool-down path was assumed to be linear as shown in Figure 6 with dashed lines. The 2nd and more heat-up and cool-down cycles follow the linearized path in parallel according to the pressure level and the maximum temperature experienced during the 1st heat-up cycle.

The direct gasket behavior definition in ABAQUS is specified by the 'pressure – closure' curve of a single layer through the thickness direction by default. The left-most diagram in Figure 8 shows the idealized deformation mode of a single layer of gasket elements. In order to get the same deformation with a double layer of gasket elements, then the first assumption would be that each layer deforms half the total deformation to accumulate the same amount of total deformation as shown in the middle of Figure 8. This assumption is valid when the initial geometry of the multi-layer gaskets, so called 'stacked gasket', has the same initial thickness for each layer.

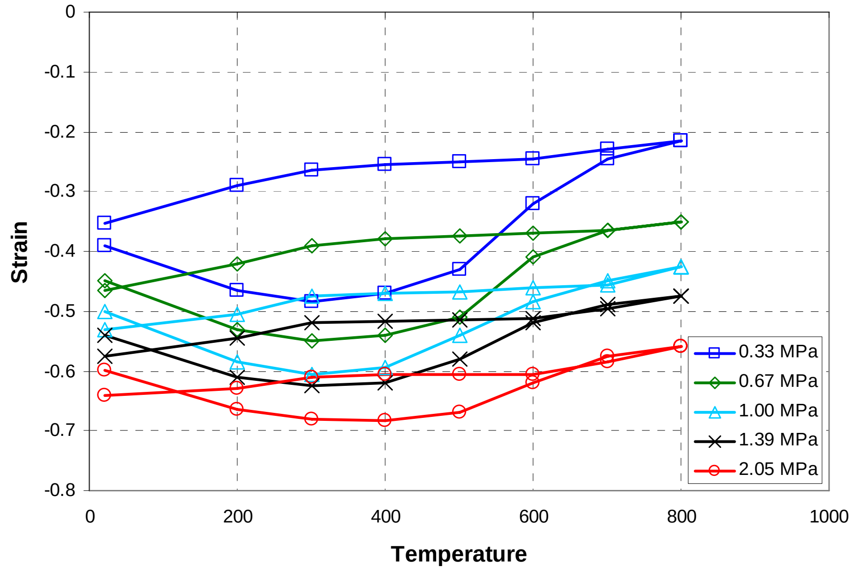


Figure 3. Conversion of displacement in Figure 1 to the total strain.

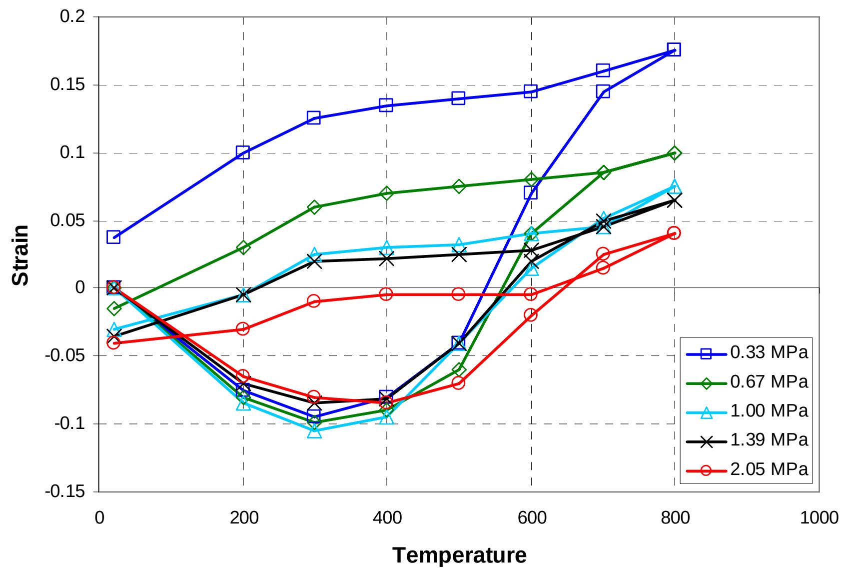


Figure 4. The pure thermal strain curves.

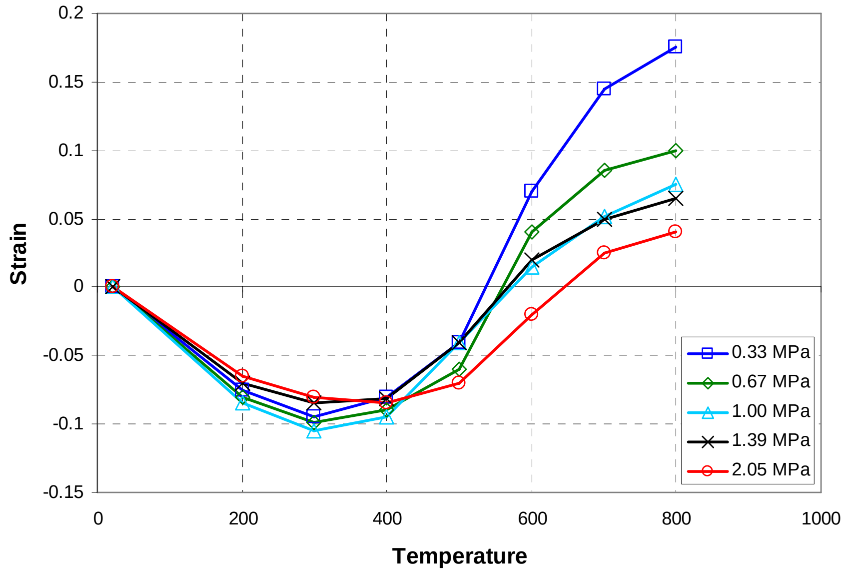


Figure 5. The 1st heat-up path.

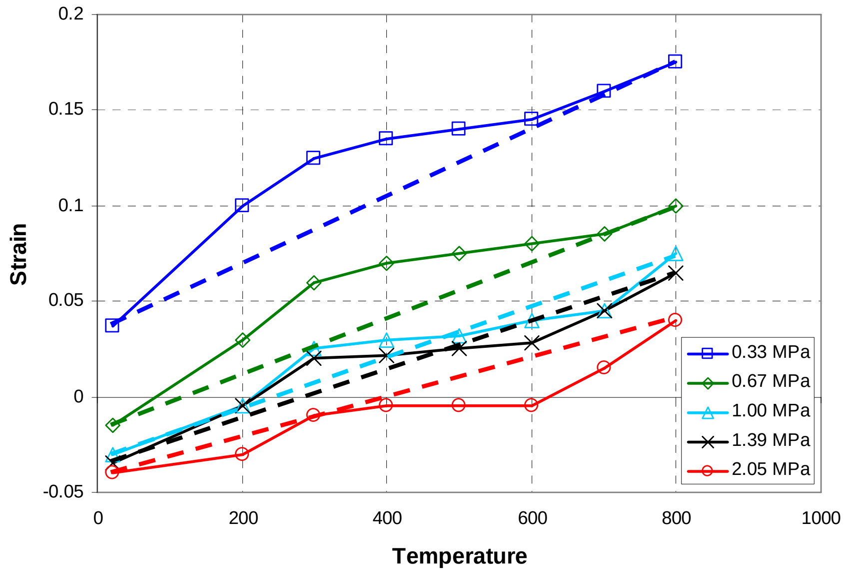


Figure 6. The 1st cool-down path.

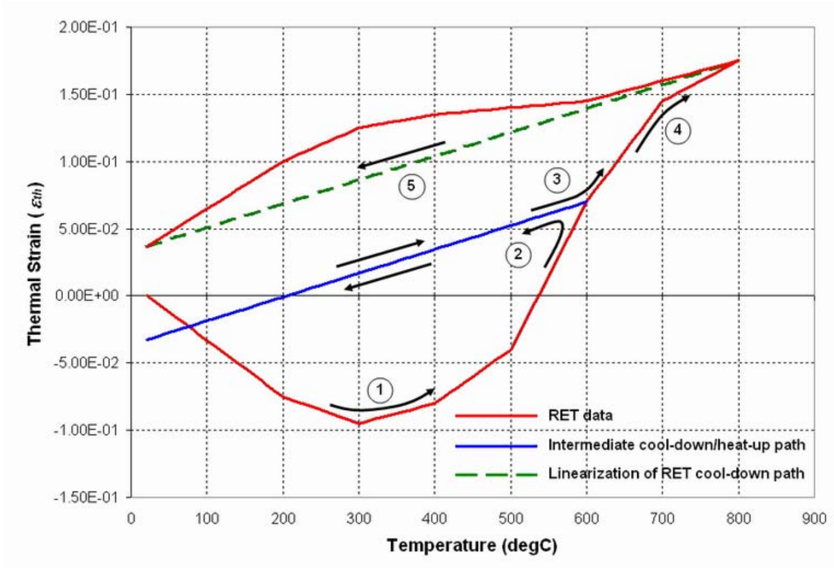


Figure 7. Schematics of the multiple heat-up and cool-down cycles (0.33 MPa).

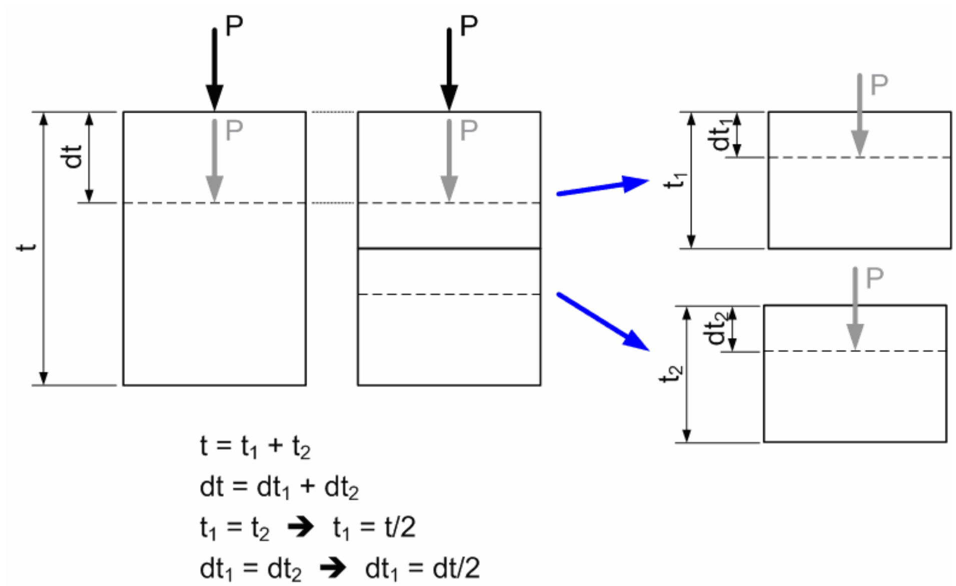


Figure 8. A schematic of the single and double layer gasket deformation.

3.2 Gasket behavior definition

The gasket behavior can be defined with the built-in (or user defined) material model or direct gasket behavior model in ABAQUS. The direct method was used in this paper to be able to define the finite strain formulation with orthotropic thermal expansion. The nonlinear elastic properties with damage and the thermal effects in the thickness direction were used.

3.2.1 Mechanical behavior

The mechanical behavior of gasket elements representing an intumescent mat material was defined by the nonlinear elastic properties with damage in the thickness direction. This thickness direction behavior depends on the loading rate as shown in Figure 2. The ABAQUS syntax for this definition is as follows:

```
*Gasket thickness behavior, type=damage, dependencies=1
Pressure, Closure, , Field Variable1 (FV1)
```

Field Variable1 (FV1) values are the nodal velocity, which correspond to the loading speed values of Figure 2.

3.2.2 Thermal behavior

The thermal behavior was defined by defining orthotropic thermal expansion coefficients for the gasket elements. The ABAQUS syntax for this definition is:

```
*Expansion, type=ORTHO, dependencies=4
α11, α22, α33, Temperature,, FV2, FV3, FV4
```

The 'type=ORTHO' parameter indicates that orthotropic thermal expansion coefficients will be defined. α_{11}, α_{22}, and α_{33} are the thermal expansion coefficients in the thickness, in-plane (direction1), and in-plane (direction2), respectively. Since the thermal expansions in the in-plane directions are assumed to be zero, the 2nd and 3rd column data are always zero.

The '*Temperature*' values define the temperature dependencies for the thermal expansion coefficients. A total of four field variables are needed to complete the definition. The first field variable, FV1, is already assigned for the *Gasket thickness behavior keyword, and it has no effect on the *Expansion keyword. So, the column for FV1 is left blank. The second field variable, FV2, specifies the pressure values (see Figure 1, Figure 3, and Figure 4) to define a pressure dependency of the thermal expansion. The third field variable, FV3, specifies the maximum temperature reached during the 1st heat-up path. The fourth field variable, FV4, is a thermal cycle identifier (0: the 1st heat-up, 1: the 1st cool-down and more cycles).

3.2.3 Multiple layers of gasket elements

As assumed in Section 3.1, the ABAQUS syntax for the multi-layer gasket definition would be;

```
*Gasket thickness behavior, type=damage
0., 0.,
P1, C1/L,
P2, C2/L,
……
```

where P_1, P_2, etc. are pressure values, C_1, C_2, etc. are the corresponding closure values for each gasket element when the number of layer is one, and L is the number of layers.

3.3 Field variable assignment

As described in the previous sections, the current gasket element representation for an intumescent mat material is defined by the loading rate dependent 'pressure – closure' curves and the pressure, temperature, and thermal cycle dependent thermal expansion coefficients. All these dependencies were defined by using field variables in ABAQUS as explained in Section 3.2. The assignment of all field variables was done by using an ABAQUS user subroutine 'UFIELD.'

The first field variable (FV1) covers the loading rate dependence. Since the DCT test is performed in the thickness direction only, it is possible to get the loading rate when the nodal velocity of the gasket element in the thickness direction is monitored. The user subroutine 'URDFIL' was used to monitor the nodal velocity and '.fil 'file output is required for 'URDFIL' to read output variables. The nodal velocity read in by 'URDFIL' is transferred to 'UFIELD' through a common block in order to assign user defined field variable values.

The second field variable (FV2), pressure, is assigned by monitoring pressure values of the gasket elements through 'URDFIL' and 'UFIELD.' The third field variable (FV3), the maximum temperature reached in the previous heat-up cycle, is assigned by the 'Temp' variable of 'UFIELD.'

The fourth field variable (FV4), thermal cycle identifier, is assigned by monitoring 'Temp' variable of 'UFIELD.'

3.4 ABAQUS input file preparation

The developed gasket element representation of an intumescent mat material is very simple and reliable, and it is an empirical model. Since it is an empirical model, the full set of test data (RET and DCT) are essential and the accuracy of simulations is determined by the quality of this test data.

The conversion of the full set of test data to corresponding ABAQUS keywords is very time consuming and a tedious task. Therefore, the authors developed a python script to automate this process. As the result of this automation, the partial input file with node and element definitions and Microsoft^{TM} Excel spreadsheets containing RET and DCT test data are the only requirements needed to create a ready to run ABAQUS input file.

4. Simple model analysis and correlation

4.1 Model preparation

A simple finite element model shown in Figure 9 was built to verify the developed model. Only a quarter of a full circular shape was modeled, and symmetric boundary conditions were applied. A single layer of gasket elements with 10mm original thickness are in between the top and bottom steel plates. Simulations using multiple layers of gasket elements were also tested.

Linear elastic material properties, E = 200GPa and ν = 0.3, were used for the steel plates. The multiple loading rate dependent pressure – closure curves and the pressure and temperature dependent thermal expansion coefficients, as described in Chapter 3, were defined for the gasket elements representing the intumescent mat material.

The analysis procedure was defined with 3 or 5 steps, 3 steps for one thermal cycle analysis and 5 steps for two thermal cycle analysis. The first step is a pressure loading step for all cases with 5 different pressure levels, 0.33, 0.69, 1.00, 1.39, and 2.05 MPa, which correspond to the pressure levels of the RET test data. The single thermal cycle analyses were defined with a peak temperature of 800ºC for the whole gasket elements in Step 2, then cool down to room temperature in Step 3. The peak temperature during Step 2 for two thermal cycle analyses was 600ºC, then cool down to room temperature in Step 3. And the gasket elements are heated up again to 800ºC in Step 4, and cooled down to room temperature in Step 5.

The pressure loading in Step 1 was applied with several speeds, 0.005 mm/sec, 0.4 mm/sec, and 1.6 mm/sec, to verify the loading rate dependence of the mat material modeling.

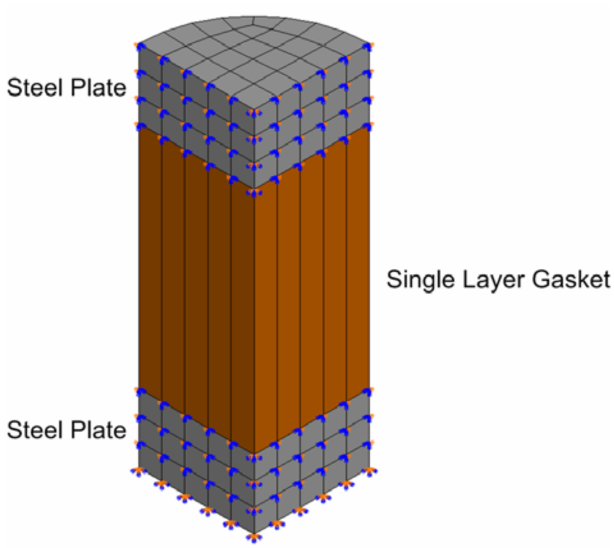


Figure 9. Simple model with 1 layer of gasket elements.

4.2 Simulation results and correlation

The simulation results are shown in Figure 10 and Figure 11 and compared with the test data. Figure 10 shows the one thermal cycle analyses results and Figure 11 shows the two thermal cycle analyses results. The developed model reproduces the test data very well.

The loading rate dependence of the developed model is shown in Figure 12, and compared with the test data. The loading rate dependence of test data was also correlated very well with the gasket element representation of this paper.

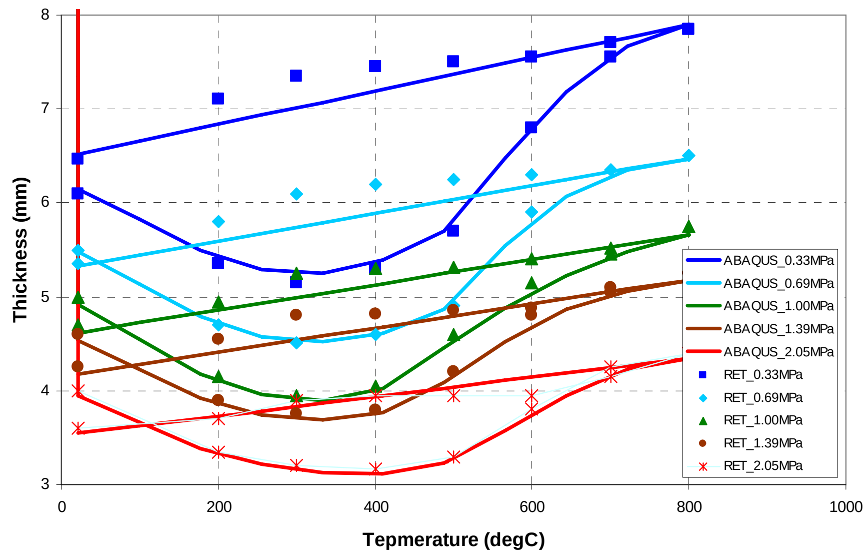


Figure 10. Correlation of the simple model simulation results with RET data.

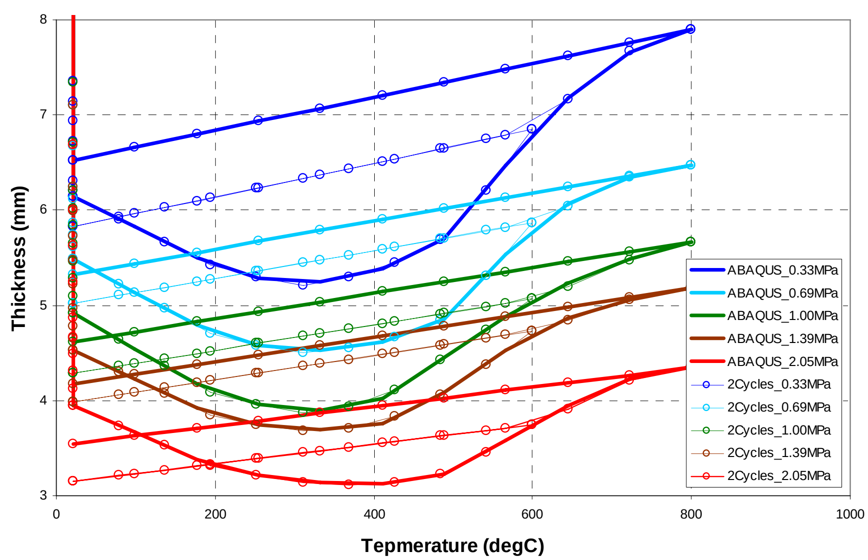


Figure 11. Comparison of 2 cycle thermal loading analyses results with single cycle results.

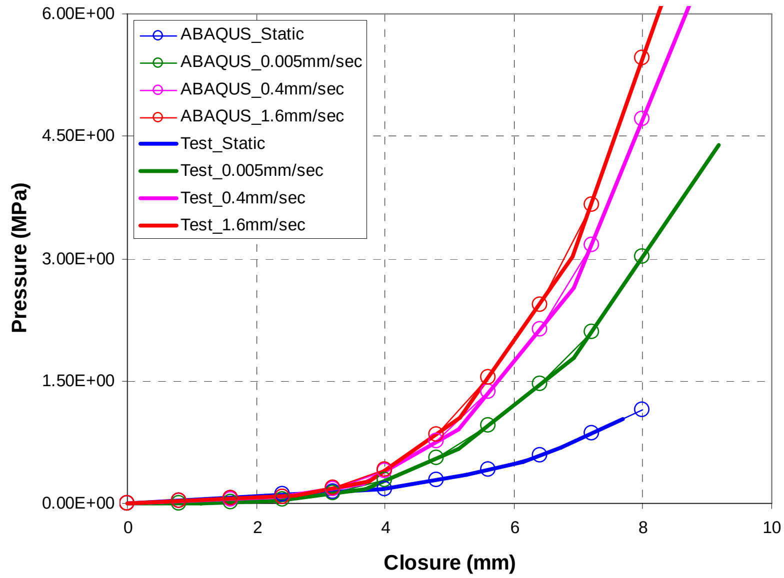


Figure 12. Validation of loading rate dependency.

5. 3D catalytic converter model analysis

5.1 3D model definition

A 3D catalytic converter model (Figure 13) was used for the developed model validation. This model was originally built for a stuffing analysis with the conventional gasket element definition. The original gasket element definition was replaced with the gasket element representation of an intumescent mat material developed in this paper.

The original analysis procedure definition for assembly analysis was used without change; (i) deforming the 'Shell' ends to match the geometry with 'Stuff Cone', (ii) pushing in the 'Mat' part through the 'Stuff Cone' and 'Shell' to locate the 'Mat' in the assembled position. The additional analysis steps of defining thermal cycles for the operation conditions were added to the assembly analysis steps.

5.2 3D model results

Figure 14 compares the pressure on the mat (a) at the end of stuffing and (b) at the end of relaxation of the Static and Dynamic analyses. The Static analysis was performed by enforcing the first field variable value to be zero. While there is no pressure level change between stuffing and relaxation steps in the Static analysis, the peak pressure due to the loading rate dependence was monitored in the Dynamic analysis. It is also observed that the relaxed pressure levels of the Static and Dynamic analyses are almost identical as expected.

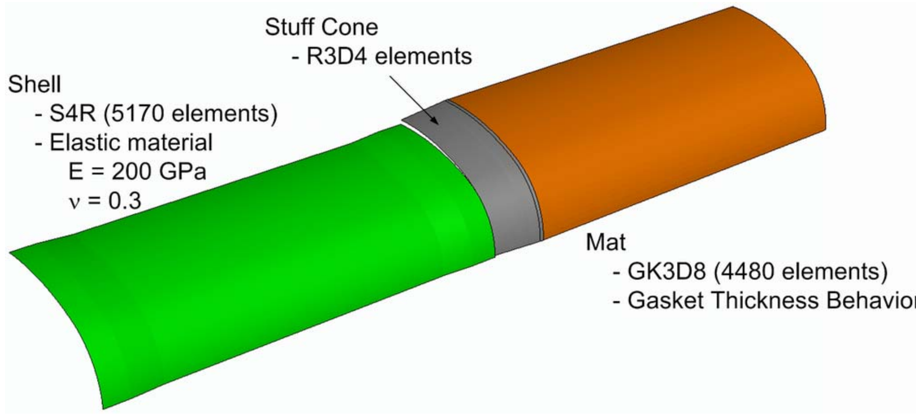
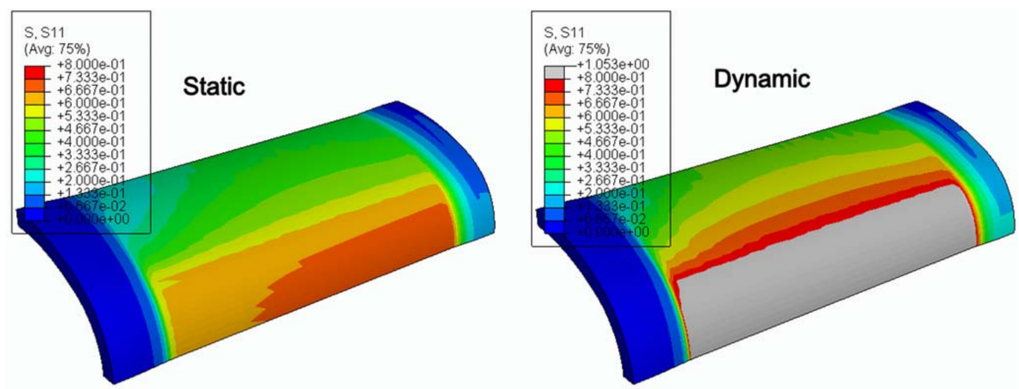
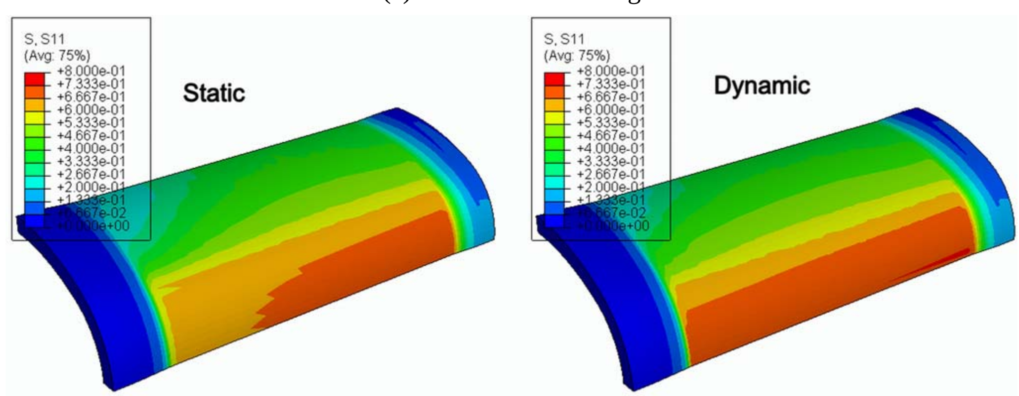


Figure 13. 3D Catalytic Converter Model.



(a) at the end of stuffing



(b) at the end of relaxation

Figure 14. Comparison of pressure on a mat (Static vs. Dynamic).

6. Concluding remarks

The complex and unique behavior of an intumescent mat material was modeled by using gasket elements in ABAQUS with the developed empirical model and two user subroutines to communicate with ABAQUS solvers. The concept of the model is simple and well understood, so that the application to the other intumescent mat materials is straightforward. The analyses of an automotive catalytic converter for both assembly and operation stages became possible with this development.

A complete ABAQUS input file preparation from the partial input file, with node and element definition, is automated by a python script. This python script reads the partial ABAQUS input file and Microsoft™ Excel spreadsheets containing RET and DCT test data, and converts the partial input file into the complete and ready to run ABAQUS input file.

References

1. Chen, D.K.S., “Integrity of an Automotive Catalytic Converter during the Assembly Process,” ASME, 90-WA/DE-6AC, 1990
2. Fuehne, J., Taylor, W., Kim, J., and Lee, J.K., “Characterization of Catalytic Converter Mat Material for Predictive Analysis,” SAE 2000-01-0219, 2000
3. Kim, H.J., Kim, J.S., Walter, M.E., and Lee, J.K., “The Thermomechanical Stress – Strain Response of Intumescent Mat Materials Through Experiments and Strain Decomposition,” Journal of Strain Energy, Vol. 37, No. 3, 2002
4. Kim, J.S., Walter, M.E., and Lee, J.K., “Modeling Mechanical Response of Intumescent Mat Material at Room Temperature,” International Journal of Mechanical Sciences, Vol. 44, 2002
5. Muju, S., Sager, R., and Snider, B., “Catalytic Converter Durability Analysis,” IMECE’97, ASME International Congress and Exposition, Nov. 16, 1996
6. Taylor, W., Fuehne, J., Lyon, R., Kim, J., and Lee, J.K., “FEA Simulation and Experimental Validation of Catalytic Converter Structural Integrity,” SAE 2000-01-0219, 2000