Hyperelastic finite element modeling of a rubber engine gasket in Abaqus

Jackson Sammartino
EGM6352
March 16, 2025

***Abstract* — Finite element analysis was performed, using Abaqus software, on a cross-section to simulate a gasket being compressed between the engine block and engine block head of a motor vehicle. Three sets of stress-strain data (uniaxial, planar shear, and biaxial) were used to compute the material properties of the rubber; the Ogden (N = 3) model fitted by Abaqus was chosen for its lowest error and reasonable stability under high compressive strains. The nonlinear hyperelastic simulation of the initial gasket geometry showed maximum stresses of 15.4 and 23.8 MPa at the top and corners of the gasket, respectively, and a gap of 9.63 mm between the gasket and the side of the engine block. Iterative methods were taken to reduce the concentration of pressure and the width of said gap; the improved geometry resulted in about 6% lower stress and the gap *d* was decreased to 2.47 mm.**

***Index Terms* — Abaqus software, hyperelastic materials, nonlinear finite element analysis, Ogden hyperelastic model.**

I. Introduction

The goal of this project was to model the 12.6 mm compression of a rubber engine gasket for use in an automobile (Fig. 1) and compute the pressure distribution and distance between the gasket and the engine block wall, *d* (Fig. 2). Then, improvements were sought to make the pressure distribution more uniform to reduce wear on the material, and to minimize *d* to prevent costly oil leakage.

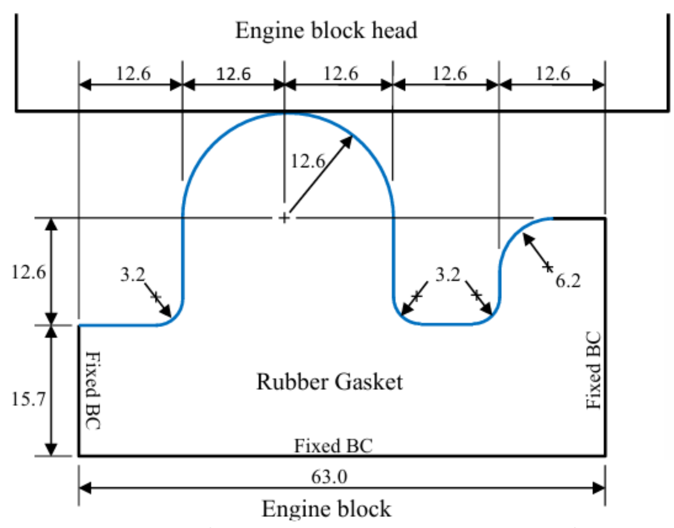

Fig. 1: planar view of original gasket geometry [1]. Note that a fixed boundary condition was imposed on all black edges. Units are in mm.

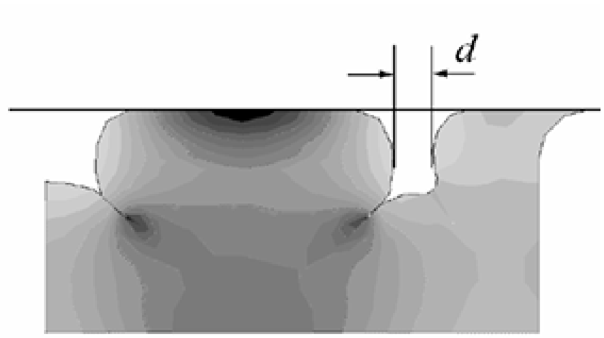

Fig. 2: Arbitrary pressure distribution of the compressed gasket showing gap *d* between gasket and engine block wall [1].

To accomplish these goals, a hyperelastic material model was used, material properties were estimated from experimental data, and Abaqus software was used for computation. What follows is a brief introduction to hyperelasticity.

Hyperelastic materials, like rubbers, foams, and soft/biological tissues, are frequently encountered in the real world and in the engineering practice. In contrast to typical linear elastic materials, i.e. ductile metals like steel, hyperelastic materials exhibit significant geometric nonlinearity- their deformation is not linearly proportional to applied loads. They deform greatly under even small forces, and can still return to their original shape after unloading. With these characteristics in mind, it is clear that analyzing the mechanics of hyperelastic materials requires quite a different formulation that that of linear elastic materials [2].

Mathematically, a hyperelastic material model is a type of constitutive relation for ideally elastic materials for which the stress–strain relationship is derived from a strain energy density function, W [2]. The St. Venant-Kirchoff material model is a ubiquitous case of hyperelasticity, and stress can be calculated as follows:

$$W = \frac{1}{2}E{:}D{:}E \rightarrow S = \frac{\partial W}{\partial E}$$

In the above equation, E is the Green-Lagrange strain tensor, D is the fourth-order constitutive tensor for isotropic materials (the same as it is for traditional linear elasticity), and S is the second Piola-Kirchoff stress [2]. Using these ideas, the materials properties of a hyperelastic material can be estimated from typical stress-strain testing data, which can then be implemented in a finite element analysis software, like Abaqus.

II. Procedure

Estimation of Material Properties

For this project, prior data were available for the cases of uniaxial tension, planar/pure shear, and biaxial tension. Table I displays the data below, where λ is the maximum principal stretch ratio. No volumetric stress data were available, so a nearly incompressible Poisson's ratio v of 0.495 was assumed. Three models were used to estimate material properties and compare their fits to the actual test data. In addition, the numerical stability of each model was assessed and the most favorable model was chosen to be used in the simulation.

Table I: Preliminary Testing Data for Rubber Material

Strain	λ	**Uniaxial**	**Shear**	**Biaxial**
0	1	0	0	0
0.1	1.1	4.2	5	6.4
0.2	1.2	6.4	7.6	9.4
0.3	1.3	8	9	11
0.4	1.4	9.2	10.2	13
0.5	1.5	10	11.2	15
0.6	1.6	11.2	12.4	17.6
0.7	1.7	12.6	14.8	21.4

Note: strain and λ are unitless, stress values are in MPa [1].

Bilinear Regression in MATLAB (Mooney-Rivlin Model)

The first method used the following formulation to compute material properties A_{10} and A_{01} for a hyperelastic material, where T is the nominal stress value [1].

$$T = \frac{\partial W}{\partial \lambda}$$

Applying this equation to the three stress cases yielded the following matrix equations, which are essentially the Mooney-Rivlin model:

$$T_{uniaxial} = [2(\lambda - \lambda^{-2}) \quad 2(1 - \lambda^{-3})] \begin{Bmatrix} A_{10} \\ A_{01} \end{Bmatrix}$$

$$T_{shear} = [2(\lambda - \lambda^{-3}) \quad 2(\lambda - \lambda^{-3})] \begin{Bmatrix} A_{10} \\ A_{01} \end{Bmatrix}$$

$$T_{biaxial} = [2(\lambda - \lambda^{-5}) \quad 2(\lambda^{3} - \lambda^{-3})] \begin{Bmatrix} A_{10} \\ A_{01} \end{Bmatrix}$$

$$(24 \times 2)\ coefficient\ matrix = [x_1 \quad x_2]$$

From this point, a complete matrix equation was created to encompass all three stress tests, and the MATLAB *fitlm* function was used to fit a bilinear model between the coefficient matrix and the T vector. The resulting coefficients of x_1 and x_2 were the material properties A_{10} and A_{01}, both in units of MPa.

Mooney-Rivlin and Ogden (N = 3) Models in Abaqus

Abaqus software also has the ability to fit various hyperelastic material models to user-inputted data. Curve fits were done for the Mooney-Rivlin model (and compared to the MATLAB bilinear regression results), as well as for the Ogden model with N = 3. Both formulations are shown below in order, where W is the strain energy density function. For the Mooney-Rivlin model, A_{10} and A_{01} are material properties, and I_1 and I_2 are the invariants of the left Cauchy-Green deformation tensor C [2]. For the Ogden model (N = 3), μ_i and α_i are material properties, and λ are the principal stretch ratios in the deformed state.

$$W = A_{10}(I_1 - 3) + A_{01}(I_2 - 3)$$

$$W = \sum_{i=1}^{N} \frac{\mu_i}{\alpha_i}(\lambda_1{}^{\alpha_i} + \lambda_2{}^{\alpha_i} + \lambda_3{}^{\alpha_i} - 3)$$

Finite Element Modeling in Abaqus

For the Abaqus Model, the engine gasket was created as a 2D planar, deformable Part, sketched in 2D and dimensioned according to Fig. 1. The lower left corner was constrained (fixed) to fully define the Section Sketch. The Ogden (N = 3) material model was selected for Materials because of its low error and reasonable stability limits. A solid, homogeneous Section was created (no thickness specified) and assigned to the Part.

The engine block head was created as a 2D planar, analytical rigid Part, sketched in 2D above the gasket Part and dimensioned at 63 mm in length as shown in Fig. 1. No Materials nor Sections were necessary. However, a reference point was added to the Part, and a rigid body Constraint was imposed.

In the Initial Step, a fixed boundary condition (BC) was applied to all the black edges of the gasket shown in Fig. 1. To model friction and contact, an Interaction Property was created with the following details: "hard" contact, penalty friction formulation, friction coefficient μ of 0.05. A surface-to-surface contact Interaction was then created between the engine block head and the blue edges of the gasket (Fig. 1).

In the displacement Step (after Initial Step), the nonlinear geometry feature was used with automatic incrementation (initial 0.05, minimum 10^{-5}, maximum 0.1). Another Interaction was created to model the gasket contacting itself- self contact with the same Interaction Property applied to the blue edges of the gasket (Fig. 1). Finally, a displacement boundary condition was applied to the engine block head (U1 = 0, U2 = -12.6, UR3 = 0).

For meshing the gasket Part, plane stress elements (CPS4) were used to model the planar compression of the gasket. Reduced integration was turned off to more accurately model the multiple contacting surfaces. Incompatible modes were not used because shear stress was expected in this loading scenario, especially due to friction and contacting surfaces. For the initial simulation, the Part was seeded based on Abaqus recommended defaults (approximate global size 3.2 with curvature and minimum size control) and the mesh was generated. Subsequent simulations were performed with finer mesh densities and part seeds to determine the most accurate results.

The Job was created, input file written, data check passed, and analysis report generated. This process was repeated with different meshings, and modified Section Sketch geometries to optimize the previously discussed performance of the gasket.

III. Results and Discussion

Material Models

Analysis of the experimental stress-strain data (Table I) gave the following results for each material model (Tables II-IV).

Table II: MATLAB Mooney-Rivlin Bilinear Regression

Intercept **(MPa)**	A_{10} **(MPa)**	A_{01} **(MPa)**	***RMSE*** **(MPa)**	R^2 **(unitless)**
1.04	3.8215	0.71385	0.778	0.979

Table III: Abaqus Mooney-Rivlin Results

Intercept **(MPa)**	A_{10} **(MPa)**	A_{01} **(MPa)**
1.8963×10^{-3}	4.8398	0.45122

Table IV: Abaqus Ogden (N = 3) Results

i	μ_i	α_i	***Intercept***
1	75.415	3.1381	1.6498×10^{-3}
2	-63.253	3.4452	0
3	1.5737×10^{-3}	-9.9244	0

The errors of each model, as well as their stability, were considered before implementation in the Abaqus Model. Table V highlights these criteria for the two Abaqus models.

Table V: Selection Criteria for Abaqus Material Models

	Uniaxial	**Biaxial**	**Shear**	**Stability**
Mooney-Rivlin	0.1655	0.05879	0.1409	Perfect
Ogden (N = 3)	0.06867	0.02062	0.07319	Fair

Note: numerical values are the RMSE (MPa) of the models compared to experimental data, and stability was inspected at high strain values.

Based on these results, the Ogden (N = 3) material model was selected for its lowest error- one order of magnitude lower than the other models. The stability of the model was in question, but the ranges for which instability occurred were very extreme, and graphical inspection revealed an excellent fit to experimental data.

Initial Geometry and Default Meshing Simulation

The desired results of the simulation were the von Mises (Fig. 3) and pressure distributions (Fig. 4) within the gasket, the load-displacement curve of the engine block head (Fig. 5), and the gap *d* (Fig. 1) which, for this initial trial, was approximately 9.777 mm.

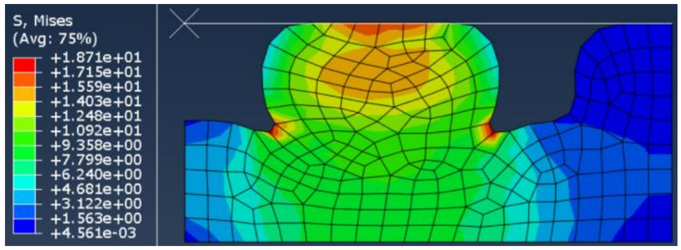

Fig. 3: Abaqus plot of von Mises stress (MPa) for the initial gasket Model.

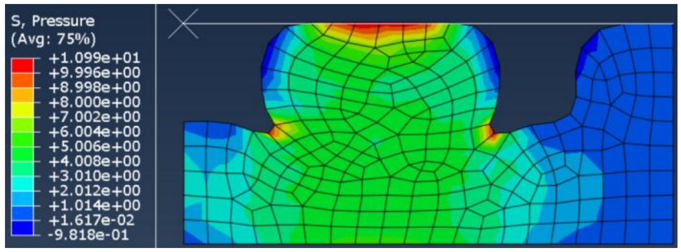

Fig. 4: Abaqus plot of pressure (MPa) for the initial gasket Model.

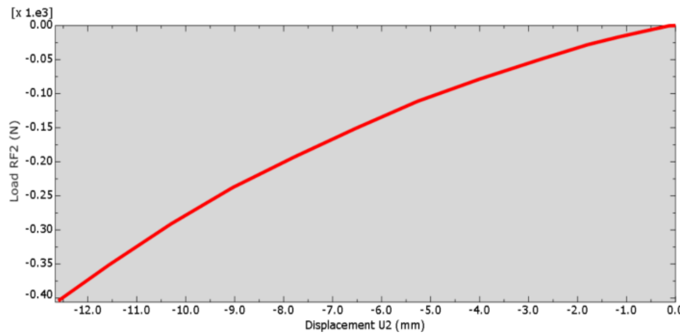

Fig. 5: Abaqus load-displacement curve for the initial gasket Model.

Mesh Convergence (Initial Geometry)

To determine an appropriate element size for the most accurate results, mesh convergence was performed under various global element size settings. Table VI summarizes the results, in order of which they were tested.

Table VI: Summary of Mesh Density Analysis

Size	**Elements**	**Stress at Top (MPa)**	**Max Stress (MPa)**	**Gap *d* (mm)**
3.2	236	17.023	18.711	9.7770
1.6	850	15.558	21.345	9.6366
0.8	3206	NO	CONVER	GENCE
1.2	**1392**	**15.441**	**23.800**	**9.6328**

As shown in Table VI, refining the mesh density yielded lower values of stress along the top of the gasket, but higher maximum stresses at the fillets. Because these fillet locations are effectively stress concentrators, the convergence of the top stress was prioritized.

Modified Geometry

As previously stated, the geometry of the gasket contour (blue edges, Fig. 1) was modified to improve performance. The goals were to make the pressure distribution more uniform to reduce wear on the material, and to minimize *d* to prevent costly oil leakage. An iterative trial-and-error approach was taken, and the best mesh settings were chosen based on Table VI. Figures 6-9 show the final geometry, stress plot, pressure plot, and load-displacement curve, respectively (1710 elements). The gap *d* was reduced to 2.469 mm.

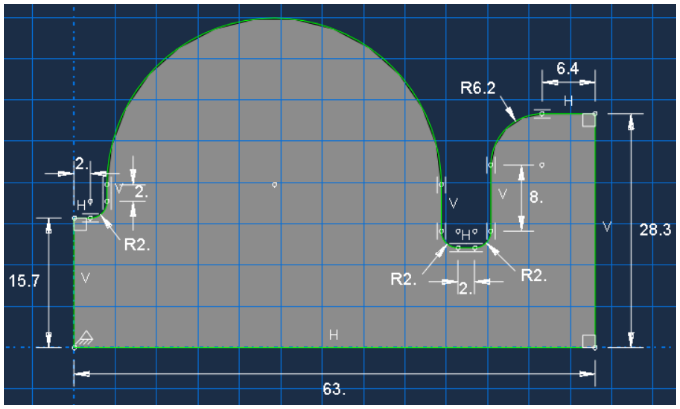

Fig. 6: Modified gasket geometry (mm).

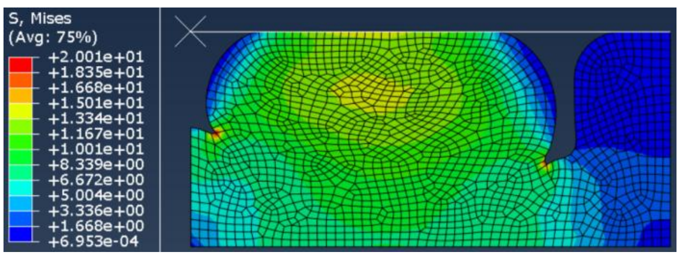

Fig. 7: Abaqus plot of von Mises stress (MPa) for the final gasket Model.

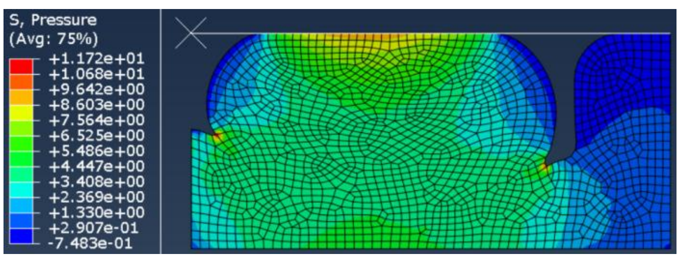

Fig. 8: Abaqus plot of pressure (MPa) for the final gasket Model.

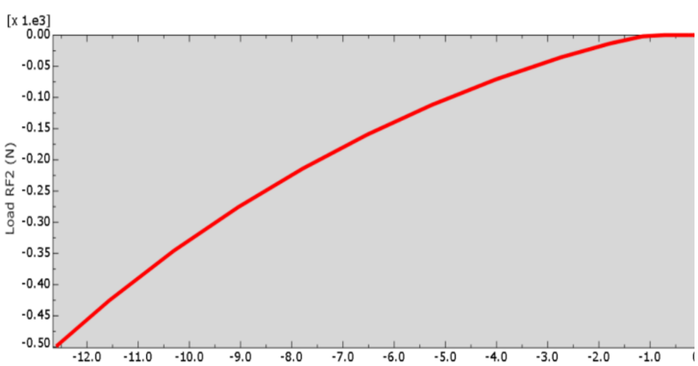

Fig. 9: Abaqus load-displacement curve for the final gasket Model.

Comparison of Initial to Final Geometry

In addition to Figures 6-9, Table VII below summarizes the improvements made to the gasket.

	Stress at Top (MPa)	**Max Stress (MPa)**	**Gap *d* (mm)**	**Load Capacity (N)**
Initial	15.441	23.800	9.6328	394.56
Final	14.583	20.014	2.4686	498.94

The improved gasket geometry resulted in decreased stress levels- about 5.88% lower along the top of the gasket, and 18.9% lower at the fillets. Comparing Figures 4 and 8, the pressure distribution along the compressed surface is slightly more uniform, and clearly lower in magnitude. The gap *d* was drastically reduced by 74.4%. The load-bearing capacity was increased by 26.5%. Comparing Figures 5 and 9, the load-displacement curve of the modified gasket increased more steeply, and converged very quickly at low displacements. In summary, the new gasket geometry was successful and offers great benefits over the initial geometry.

IV. Conclusion

The goal of this project was to model a hyperelastic rubber material in Abaqus, simulating the compression of a rubber gasket in an automobile. Based on curve fits to experimental data, the Ogden material model (N = 3) was used. Results for computed for stress and pressure distributions, load-displacement relations, and the gap *d* was measured. The geometry of the gasket was modified, decreasing the stress and making the pressure distribution marginally more uniform. The gap *d* was drastically reduced. The updated gasket was successfully implemented and should last longer than the initial gasket, with less oil leakage.

Appendix

MATLAB Code for Mooney-Rivlin Model

```
clc; clear;
l = [1 1.1 1.2 1.3 1.4 1.5 1.6 1.7];
T = [0; 4.2; 6.4; 8; 9.2; 10; 11.2; 12.6;
    0; 5; 7.6; 9; 10.2; 11.2; 12.4; 14.8;
    0; 6.4; 9.4; 11; 13; 15; 17.6; 21.4];
x = zeros(24, 2);
for ii = 1:1:8
    for jj = 1:1:2
        lamb = l(ii);
        if jj == 1
            x(ii, jj) = 2*(lamb - lamb^-2);
        else
            x(ii, jj) = 2*(1 - lamb^-3);
        end
    end
end
for ii = 9:1:16
    for jj = 1:1:2
        lamb = l(ii-8);
        x(ii, jj) = 2*(lamb - lamb^-3);
    end
end
for ii = 17:1:24
    for jj = 1:1:2
        lamb = l(ii-16);
        if jj == 1
            x(ii, jj) = 2*(lamb - lamb^-5);
        else
            x(ii, jj) = 2*(lamb^3 - lamb^-3);
        end
    end
end
model = fitlm(x, T);
disp(model);
```

MATLAB Code Output

Linear regression model:
y ~ 1 + x1 + x2

Estimated Coefficients:

	Estimate	SE	tStat	pValue
(Intercept)	1.04	0.3332	3.1214	0.0051624
x1	3.8215	0.24574	15.551	5.3537e-13
x2	0.71385	0.10233	6.9762	6.8748e-07

Number of observations: 24, Error degrees of freedom: 21
Root Mean Squared Error: 0.778
R-squared: 0.98, Adjusted R-Squared: 0.979
F-statistic vs. constant model: 525, p-value = 1.18e-18
>>

References

[1] Kim, Nam-Ho, “Project 1,” assignment description, course EGM6352, University of Florida, Spring 2025.

[2] Kim, Nam-Ho, “Chapter 3: FEA for Nonlinear Elastic Problems,” class notes, course EGM6352, University of Florida, Spring 2025.