Created in COMSOL Multiphysics 6.2

Polynomial Hyperelastic Model

Introduction

This example shows how you can implement a user-defined hyperelastic material using the strain energy density function. The implemented model is a general Mooney–Rivlin hyperelastic material model defined by a polynomial.

For such a material model, the strain energy density function has the following expression:

$$W = \sum_{i,j=0}^{n} C_{i,j}(\bar{I}_1 - 3)^i(\bar{I}_2 - 3)^j + \frac{1}{2}K(J_{\mathrm{el}} - 1)^2$$

Here $\bar{I}_1$ and $\bar{I}_2$ are the first and second invariant of the left isochoric Cauchy–Green deformation tensor, J_{el} is the elastic Jacobian, $C_{i,j}$ are coefficients in the polynomial, and K is the bulk modulus.

In this example, you implement two material models based on the above expression: a two-parameter equation and a five-parameter equation. The two-parameter Mooney–Rivlin material model implementation is then validated with the results obtained with the built-in Mooney–Rivlin hyperelastic material.

Model Definition

A simple geometry is used consisting of a single block of the hyperelastic material as shown in Figure 1. The block is fixed at one face and loaded with a uniform normal load of 1 MPa at the opposite face. Due to symmetry, only one quarter of the geometry is represented.

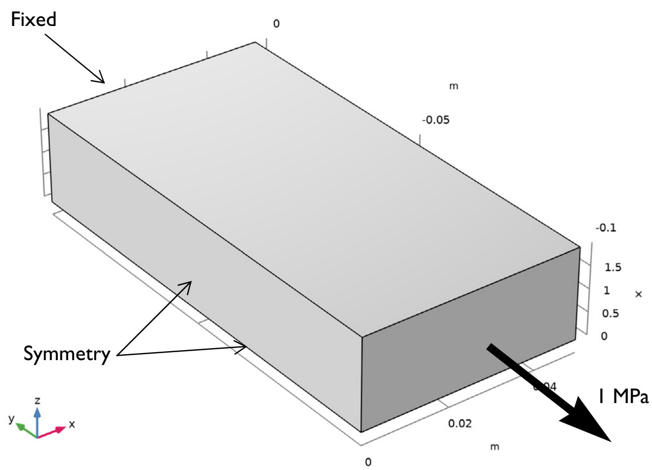

Figure 1: Model geometry with boundary conditions and loads.

The two-parameter Mooney–Rivlin material model is defined by the following strain energy density:

$$W_{\mathrm{siso}} = C_{1,0}(\bar{I}_1 - 3) + C_{0,1}(\bar{I}_2 - 3)$$

$$W_{\mathrm{svol}} = \frac{1}{2}\kappa(J_{\mathrm{el}} - 1)^2$$

The five-parameter Mooney–Rivlin material model is defined by the following strain energy density:

$$W_{\mathrm{siso}} = \begin{pmatrix} C_{1,0}(\bar{I}_1 - 3) + C_{0,1}(\bar{I}_2 - 3) + C_{2,0}(\bar{I}_1 - 3)^2 + \\ C_{0,2}(\bar{I}_2 - 3)^2 + C_{1,1}(\bar{I}_1 - 3)(\bar{I}_2 - 3) \end{pmatrix}$$

$$W_{\mathrm{svol}} = \frac{1}{2}\kappa(J_{\mathrm{el}} - 1)^2$$

Note: Both the two parameter and the five parameter Mooney-Rivlin material model are available in the **Hyperelastic Material** node.

Results and Discussion

Figure 2 shows the y-component of the second Piola–Kirchhoff stress along the center axis of the block. You can see that the results from the two parameter polynomial equation model perfectly matches the results of the built-in Mooney–Rivlin material.

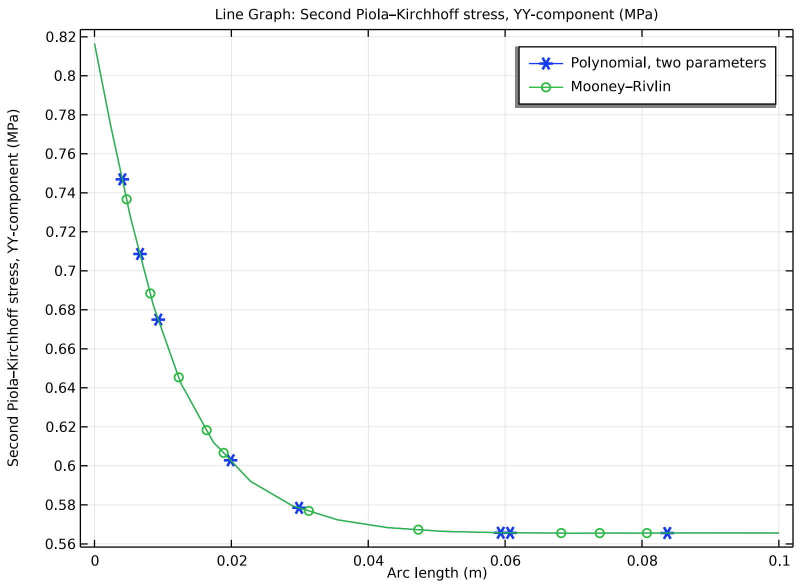

Figure 2: Stress plot (y-component of the second Piola–Kirchhoff stress) along the length of the block.

Figure 3 shows the von Mises stress distribution in the geometry obtained with the two parameter Mooney–Rivlin material. Figure 4 shows the von Mises stress distribution in the geometry with the five parameter Mooney–Rivlin material model. Note the difference in deformation: the five parameter polynomial model has a significantly smaller deformation than the two parameter model

For the five parameter material, you can see that the stress in the region far away from the fixed end is significantly lower than for the two parameter material. This is because the area reduction is much larger with the more flexible two parameter material, although, the total load is the same in both cases. The von Mises stress is computed from the Cauchy stress, which is based on force per current area.

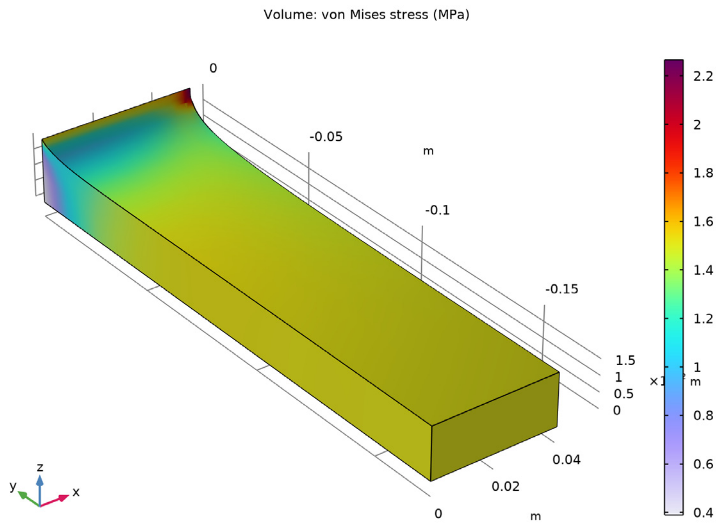

Figure 3: Distribution of the von Mises stress for the two-parameter polynomial hyperelastic material model.

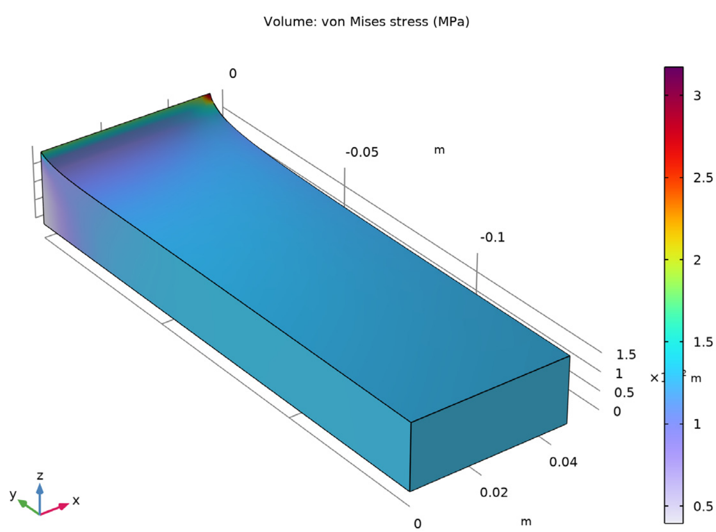

Figure 4: Distribution of the von Mises stress for the five-parameter polynomial hyperelastic material model.

Notes About the COMSOL Implementation

Instead of using the predefined hyperelastic material model, you manually define the material in the **Hyperelastic Material** node's **Settings** window. In the Hyperelastic Material section, select **User defined** from the **Material model** list.

For nearly incompressible materials, the strain energy density is defined using a separation of the isochoric strain energy density and the volumetric strain energy density.

When you use a hyperelastic material in your model, all studies automatically become geometrically nonlinear.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/polynomial_hyperelastic`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **3D**.
2. In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`kappa`	`3[MPa]`	3E6 Pa	`Bulk modulus`
`C01`	`0.5[MPa]`	5E5 Pa	`Polynomial coefficient C01`
`C10`	`0.1[MPa]`	1E5 Pa	`Polynomial coefficient C10`
`C11`	`0.15[MPa]`	1.5E5 Pa	`Polynomial coefficient C11`
`C20`	`0.2[MPa]`	2E5 Pa	`Polynomial coefficient C20`
`C02`	`-0.2[MPa]`	-2E5 Pa	`Polynomial coefficient C02`

GEOMETRY 1

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.1`.

4 In the **Depth** text field, type `0.05`.

5 In the **Height** text field, type `0.02`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

7 Click **Build All Objects**.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Wsiso_MR2	C10*(solid.I1CIel-3)+C01*(solid.I2CIel-3)	Pa	Isochoric strain energy density, Mooney-Rivlin two parameters
Wsiso_MR5	Wsiso_MR2+C20*(solid.I1CIel-3)^2+C02*(solid.I2CIel-3)^2+C11*(solid.I1CIel-3)*(solid.I2CIel-3)	Pa	Isochoric strain energy density, Mooney-Rivlin five parameters
Wsvol	0.5*kappa*(solid.Jel-1)^2	Pa	Volumetric strain energy density

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundary 5 only.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1 and 3 only.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
-1[MPa]	y
0	z

TWO PARAMETER POLYNOMIAL HYPERELASTIC MATERIAL MODEL

Polynomial, Two Parameters

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, type `Polynomial, Two Parameters` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **User defined**.

5 From the **Compressibility** list, choose **Nearly incompressible**.

6 In the W_{siso} text field, type `Wsiso_MR2`.

7 In the W_{svol} text field, type `Wsvol`.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 Select Boundary 5 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 6 and 12 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `4`.

6 In the **Element ratio** text field, type `5`.

7 Select the **Reverse direction** check box.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 7 and 8 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `6`.

6 In the **Element ratio** text field, type `5`.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `15`.

5 In the **Element ratio** text field, type `5`.

6 Click **Build All**.

MOONEY–RIVLIN HYPERELASTIC MATERIAL MODEL

Mooney–Rivlin

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, type `Mooney-Rivlin` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Mooney–Rivlin, two parameters**.

5 From the C_{10} list, choose **User defined**. In the associated text field, type `C10`.

6 From the C_{01} list, choose **User defined**. In the associated text field, type `C01`.

7 In the κ text field, type `kappa`.

Polynomial, Two Parameters

In the **Model Builder** window, right-click **Polynomial, Two Parameters** and choose **Duplicate**.

Polynomial, Five Parameters

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Polynomial, Two Parameters 1**.

2 In the **Settings** window for **Hyperelastic Material**, type `Polynomial, Five Parameters` in the **Label** text field.

3 Locate the **Hyperelastic Material** section. In the W_{siso} text field, type `Wsiso_MR5`.

The five parameter Mooney–Rivlin material is also available as predefined hyperelastic material.

First solve the two parameter polynomial model.

STUDY: POLYNOMIAL, TWO PARAMETERS

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study: Polynomial, Two Parameters` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Polynomial, Two Parameters** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 Select the **Modify model configuration for study step** check box.
4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Mooney–Rivlin** and **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Polynomial, Five Parameters**.
5 Click **Disable**.
6 In the **Home** toolbar, click **Compute**.

RESULTS

Stress (Polynomial, Two Parameters)

In the **Settings** window for **3D Plot Group**, type `Stress (Polynomial, Two Parameters)` in the **Label** text field.

Volume 1

1 In the **Model Builder** window, expand the **Stress (Polynomial, Two Parameters)** node, then click **Volume 1**.
2 In the **Settings** window for **Volume**, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.
4 In the **Stress (Polynomial, Two Parameters)** toolbar, click **Plot**.
5 Click the **Go to Default View** button in the **Graphics** toolbar.

Volume Maximum 1

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Maximum>Volume Maximum**.
2 In the **Settings** window for **Volume Maximum**, locate the **Selection** section.
3 From the **Selection** list, choose **All domains**.
4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>v - Displacement field, Y-component**.
5 Click to expand the **Configuration** section. From the **Find maximum of** list, choose **Absolute value**.
6 Click **Evaluate**.

Now solve the Mooney–Rivlin model.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** check box.

3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Polynomial, Five Parameters**.

4 Click **Disable**.

5 In the **Model Builder** window, click **Study 2**.

6 In the **Settings** window for **Study**, type `Study: Mooney-Rivlin` in the **Label** text field.

7 In the **Home** toolbar, click **Compute**.

RESULTS

Stress (Mooney–Rivlin)

In the **Settings** window for **3D Plot Group**, type `Stress (Mooney-Rivlin)` in the **Label** text field.

Volume 1

1 In the **Model Builder** window, expand the **Stress (Mooney–Rivlin)** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 In the **Stress (Mooney–Rivlin)** toolbar, click **Plot**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

Volume Maximum 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Volume Maximum 1**.

2 In the **Settings** window for **Volume Maximum**, locate the **Data** section.

3 From the **Dataset** list, choose **Study: Mooney–Rivlin/Solution 2 (sol2)**.

4 Click **Evaluate**.

Now solve the five parameter polynomial model.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: POLYNOMIAL, FIVE PARAMETERS

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study: Polynomial, Five Parameters` in the **Label** text field.

3 In the **Home** toolbar, click **Compute**.

RESULTS

Stress (Polynomial, Five Parameters)

In the **Settings** window for **3D Plot Group**, type `Stress (Polynomial, Five Parameters)` in the **Label** text field.

Volume 1

1 In the **Model Builder** window, expand the **Stress (Polynomial, Five Parameters)** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 In the **Stress (Polynomial, Five Parameters)** toolbar, click **Plot**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

Volume Maximum 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Volume Maximum 1**.

2 In the **Settings** window for **Volume Maximum**, locate the **Data** section.

3 From the **Dataset** list, choose **Study: Polynomial, Five Parameters/Solution 3 (sol3)**.

4 Click **Evaluate**.

To compare the results of the two parameter polynomial model with Mooney–Rivlin results, reproduce Figure 2.

Second Piola–Kirchhoff Stress

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Second Piola-Kirchhoff Stress` in the **Label** text field.

Line Graph 1

1 In the **Second Piola–Kirchhoff Stress** toolbar, click **Line Graph**.

2 Select Edge 2 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Second Piola–Kirchhoff stress (material and geometry frames) - N/m²>solid.SGpYY - Second Piola–Kirchhoff stress, YY-component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

5 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 From the **Positioning** list, choose **Interpolated**.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Polynomial, two parameters`

10 Right-click **Line Graph 1** and choose **Duplicate**.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Study: Mooney–Rivlin/Solution 2 (sol2)**.

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type `10`.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Mooney-Rivlin

6 Click to expand the **Title** section. From the **Title type** list, choose **None**.

7 In the **Second Piola–Kirchhoff Stress** toolbar, click **Plot**.