Model created in COMSOL Multiphysics 6.4

Diaphragm Accumulator

Introduction

Diaphragm accumulators are pressure vessels used in hydraulic systems. A rubber diaphragm separates the hydraulic fluid from an inert, compressible gas, typically nitrogen (Figure 1). Accumulators can serve various purposes. For example, they can provide a temporary energy storage for the hydraulic fluid and compensate for pressure and volume fluctuations, thereby dampening pulsations and shocks.

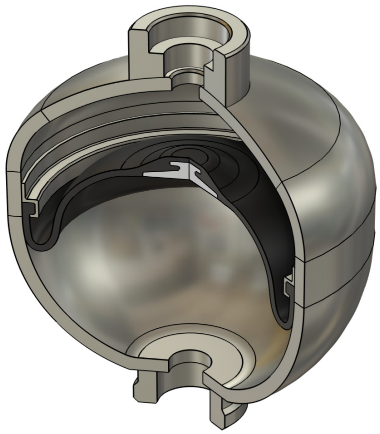

Figure 1: A diaphragm accumulator consisting of a steel pressure vessel with connections for nitrogen (top) and the hydraulic fluid (bottom). The fluids are separated by a rubber membrane.

The analysis of these accumulators involves solving a fluid–structure interaction (FSI) problem, where two types of fluids exert pressure on both sides of the diaphragm and the surrounding metallic walls.

If the pressure changes in the fluids are slow enough so that momentum and energy transfer remain minimal, it is not necessary to explicitly solve for the fluid domains. Instead, pressure changes can be computed through the equation of state.

This approach eliminates the need to mesh the fluid domain, significantly simplifying the modeling of diaphragm accumulators, especially when contact problems are involved.

Model Definition

Diaphragm accumulators can be manufactured in different ways (screwed, forged, or welded). In this example, the geometry is given for a welded design, where the diaphragm is pressed into the bottom part of the shell with a clamp ring before the accumulator is welded together.

The pressure vessel has a gas connection on the top side, and a hydraulic connection on the bottom side, where the hydraulic fluid (for example, mineral oil) enters the accumulator. The rubber diaphragm mounted in the clamp ring separates the fluids. A valve seat (also poppet or button) made from a thermoset is vulcanized into the membrane. When the accumulator is precharged, the valve seat covers the fluid connection.

The geometry of a small accumulator is illustrated in Figure 2. Some geometric details such as welds, threads, or valves are not modeled explicitly. The geometry is assumed to be axially symmetric.

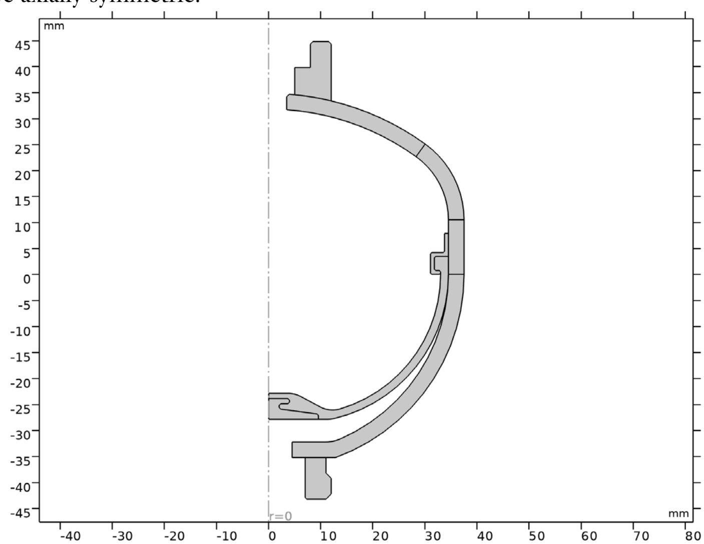

Figure 2: Diaphragm accumulator geometry.

The accumulator walls and the valve seat are modeled as linear elastic materials with material properties listed in Table 1.

TABLE 1: MATERIAL PROPERTIES.

PROPERTY	SHELL	VALVE SEAT
Young’s modulus	200 GPa	1.1 GPa
Poisson's ratio	0.3	0.35
Density	7850 kg/m^3	1300 kg/m^3

The diaphragm is made from *nitrile butadiene rubber* (NBR). It is modeled as an incompressible Neo-Hookean hyperelastic material with Lamé parameter $\mu = 0.7$ MPa and a density $\rho = 1100\ \mathrm{kg/m^3}$.

Diaphragm accumulators have different operating conditions. Initially, neither the gas nor hydraulic fluid are pressurized. Before operation, nitrogen is precharged to an operating pressure, p_0, which is determined by the operating requirements of the hydraulic system. Typically, the precharge pressure is set between 70% and 90% of the hydraulic fluid's minimum operating pressure.

During operation, the hydraulic fluid flows into the accumulator when the system pressure exceeds p_0. The system's minimum and maximum operating pressures are denoted p_1 and p_2, respectively. All pressures, p_i $(i = 0, 1, 2)$, are relative pressures with respect to the atmospheric pressure.

In this example, the accumulator is first precharged to a pressure equal to p_0 in a stationary study. In a second time-dependent study, a temporary increase of the hydraulic pressure is considered, and the response of the diaphragm and gas cavities are analyzed. The hydraulic pressure is increased to a maximum pressure p_2 and subsequently decreased to p_1. The time-dependent pressure variation is shown in Figure 3.

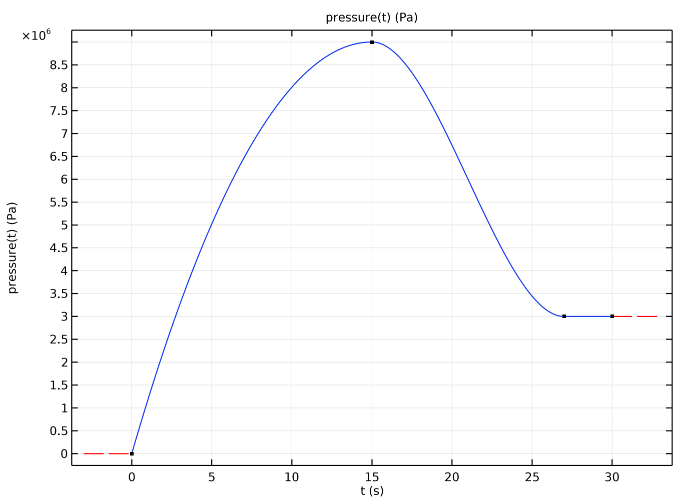

Figure 3: Pressure variation in the hydraulic fluid.

Nitrogen is modeled as an ideal gas. As the compression and expansion of the gas take place relatively fast, *adiabatic* conditions are assumed; that is, heat transfer between the gas and the surrounding walls is assumed negligible. For an adiabatic process, the deformed volume of the cavity, V, is related to its initial volume, V_{ref}, by the equation of state

$$pV^{\gamma} = p_{ref}V_{ref}^{\gamma}$$

Here, p_{ref} is the reference fluid pressure inside the undeformed cavity, p is the absolute pressure after deformation, and γ is the ratio of specific heats. For nitrogen $\gamma = 1.4$. Since $p = p_{ref} + p_{rel}$, the equation of state is equivalently written as

$$p_{rel} = p_{ref}\left[\left(\frac{V_{ref}}{V}\right)^{\gamma} - 1\right]$$

where p_{rel} is the relative pressure acting on all cavity walls. The volume in the reference and deformed configurations can be obtained using the divergence theorem (or Gauss's theorem). More information on the volume computation can be found in the section *Enclosed Fluids* in the *Structural Mechanics Theory* chapter of the *Structural Mechanics Module User's Guide*.

Results and Discussion

In the first study, the accumulator is precharged with nitrogen. There is no relative pressure acting on the hydraulic fluid side. Figure 4 shows the deformation of the diaphragm and the von Mises stress distribution after the precharge step.

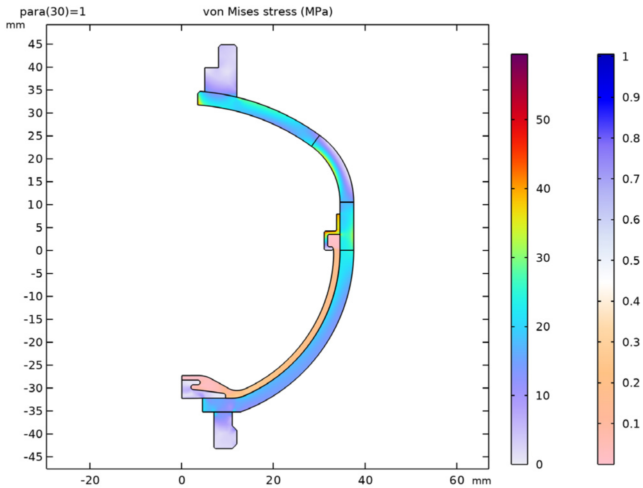

Figure 4: von Mises stress distribution after precharging the accumulator with nitrogen gas.

In the second study, the effect of a time-dependent pressure change in the hydraulic fluid is analyzed. The pressure in the hydraulic fluid increases to the maximum pressure, p_2, and the rubber membrane deforms significantly. The diaphragm bulges into the gas cavity, thereby compressing the nitrogen. The von Mises stress at $p(t = 15\ \mathrm{s}) = p_2$ is shown in Figure 5.

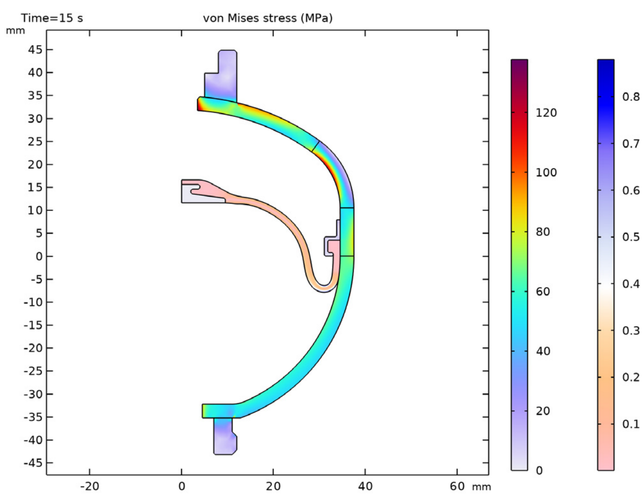

Figure 5: von Mises stress at the maximum hydraulic pressure.

Once the hydraulic pressure exceeds the precharge pressure, there is a pressure equilibrium; that is, the gas pressure and fluid pressures balance each other as shown in Figure 6.

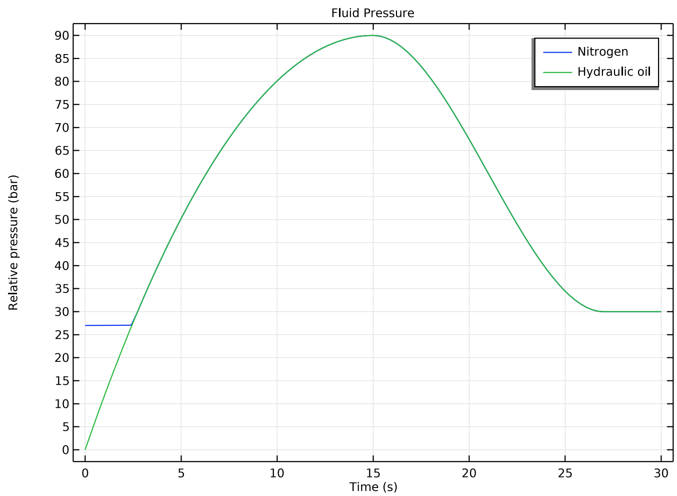

Figure 6: Pressure variation of the nitrogen gas and hydraulic fluid.

Figure 7 shows the volume of the hydraulic fluid that enters the accumulator. At a pressure $p = p_2$, a volume of 94.4 ml has entered the accumulator. During the adiabatic compression of the nitrogen chamber, the gas temperature increases significantly as shown in Figure 8.

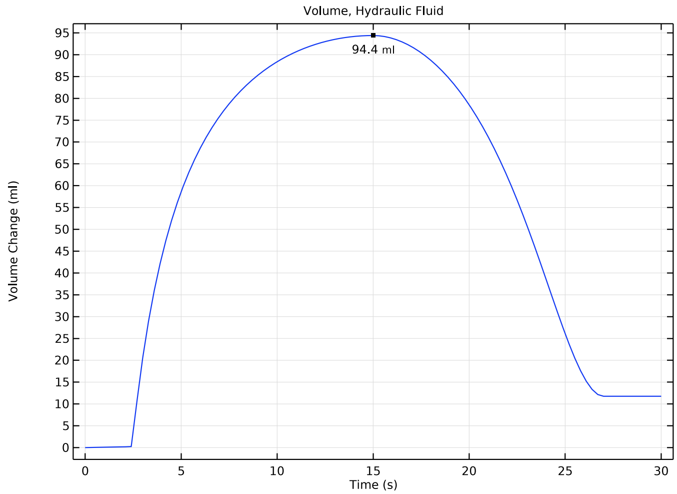

Figure 7: Volume of hydraulic fluid entering the accumulator.

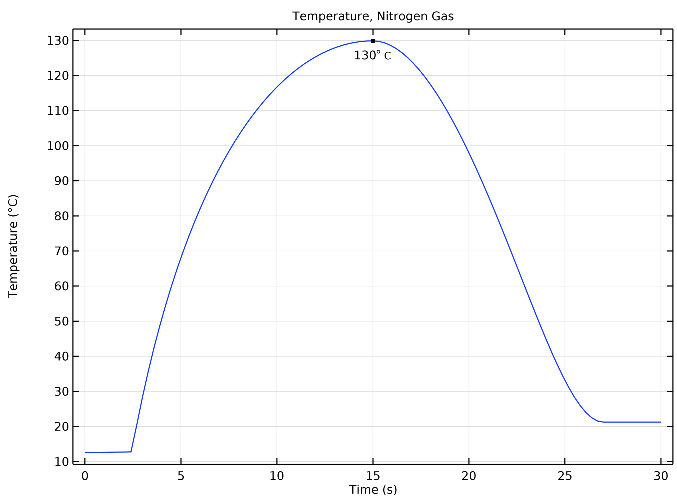

Figure 8: Temperature change of the nitrogen gas.

Notes About the COMSOL Implementation

To account for the initial strain state due to press fitting the clamp ring, an **Initial Stress and Strain** node is added. Here, the circumferential strain resulting from the press fit is entered.

The cavities for the nitrogen gas and hydraulic fluid are modeled with the built-in **Enclosed Cavity** node in the *Solid Mechanics* interface.

The **Enclosed Cavity** feature automatically computes the volume of the undeformed and deformed configurations. The advantage of this approach is such that these domains do not need to be meshed.

The pressure load is known for the hydraulic fluid. It can be applied with an ordinary **Boundary Load** node. However, it is also possible to apply this load with the **Prescribed Pressure** subnode under **Enclosed Cavity**. The advantage with this approach is that the volume change is automatically tracked.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/diaphragm_accumulator`

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 **2D Axisymmetric**.

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**.

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `diaphragm_accumulator_geom_sequence.mph`.

GLOBAL DEFINITIONS

Import the geometry sequence for the diaphragm accumulator.

Geometry Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Geometry Parameters` in the **Label** text field.

Operating Parameters

Next, add auxiliary parameters for the operating pressures.

1 In the **Home** toolbar, click **Parameters** and choose **Add** > **Parameters**.

2 In the **Settings** window for **Parameters**, type `Operating Parameters` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
para	1	1	Stepping parameter
p0	0.9*p1	2.7E6 Pa	Precharge pressure
p1	30[bar]	3E6 Pa	Minimum operating pressure
p2	3*p1	9E6 Pa	Maximum operating pressure
t_end	30[s]	30 s	Time

Interpolation 1 (int1)

Add an interpolation function for the prescribed hydraulic fluid pressure.

1 In the **Home** toolbar, click **Functions** and choose **Global** > **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `pressure`.

4 In the table, enter the following settings:

t	f(t)
0	0
t_end*0.5	p2

t	f(t)
t_end*0.9	p1
t_end	p1

5 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

6 Locate the **Units** section. In the **Function** table, enter the following settings:

Function	Unit
pressure	Pa

7 In the **Argument** table, enter the following settings:

Argument	Unit
t	s

8 Click **Plot**.

DEFINITIONS

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `13, 60, 65, 73` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

7 Click **Paste Selection**.

8 In the **Paste Selection** dialog, type `2, 25, 40, 61, 63` in the **Selection** text field.

9 Click **OK**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

As the clamp ring has been pressfitted, add an approximate initial strain.

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 8 only.

5 Locate the **Initial Stress and Strain** section. Specify the ε_0 matrix as

0	0	0
0	0.01[mm]/R	0
0	0	0

Contact 1

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

2 In the **Settings** window for **Contact**, locate the **Contact Pressure Penalty Factor** section.

3 From the **Penalty factor control** list, choose **Manual tuning**.

4 In the f_p text field, type 5.

Hyperelastic Material 1

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

2 In the **Settings** window for **Hyperelastic Material**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Diaphragm**.

4 Locate the **Hyperelastic Material** section. From the **Compressibility** list, choose **Incompressible**.

Fixed Constraint 1

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

2 Select Boundary 31 only.

Enclosed Cavity, Nitrogen Gas

Add an **Enclosed Fluid** node to model the compressible nitrogen gas. Select the boundaries, which are enclosing the gas in order to compute the volume change, and thus compute the pressure change.

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

2 In the **Settings** window for **Enclosed Cavity**, type `Enclosed Cavity, Nitrogen Gas` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `5, 9, 10, 16, 34-36, 40-42, 47, 52, 55, 59, 66, 67, 69, 70, 73, 75` in the **Selection** text field.

5 Click **OK**.

As the selected boundaries do not form a watertight volume, mark the selection as open and select a reference point. The reference point defines a virtual cut plane parallel to the R-axis. In axisymmetry, only the Z-coordinate of the chosen point is taken into account.

6 In the **Settings** window for **Enclosed Cavity**, locate the **Volume Definition** section.

7 From the **Volume type** list, choose **Open surface**.

8 Locate the **Reference Point** section. Click to select the **Activate Selection** toggle button.

9 Select Point 13 only.

Prescribed Pressure 1

Precharging the accumulator involves two study steps. First, the known precharge pressure is applied with a **Prescribed Pressure** subnode. The precharge pressure and the computed volume then serve as the new reference configuration for the equation of state when the hydraulic pressure is applied.

1 In the **Physics** toolbar, click **Attributes** and choose **Prescribed Pressure**.

2 In the **Settings** window for **Prescribed Pressure**, locate the **Prescribed Pressure** section.

3 In the *p* text field, type `p0*para`.

Fluid 1

The compression and expansion of the nitrogen gas is assumed to be adiabatic. Change the reference pressure and reference volume in order to set the new reference state for the analysis involving the hydraulic fluid.

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

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 From the **Reference pressure** list, choose **From prescribed pressure**.

4 Click to expand the **Advanced** section. From the **Reference volume** list, choose **User defined**.

Enclosed Cavity, Hydraulic Fluid

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

2 In the **Settings** window for **Enclosed Cavity**, type `Enclosed Cavity, Hydraulic Fluid` in the **Label** text field.

3 Locate the **Volume Definition** section. From the **Volume type** list, choose **Open surface**.

4 Locate the **Reference Point** section. Click to select the **Activate Selection** toggle button.

5 Select Point 15 only.

6 Locate the **Boundary Selection** section. Click **Paste Selection**.

7 In the **Paste Selection** dialog, type `2, 11, 13, 25, 60, 61, 63, 65` in the **Selection** text field.

8 Click **OK**.

Fluid 1

In the **Model Builder** window, right-click **Fluid 1** and choose **Delete**.

Prescribed Pressure 1

1 In the **Physics** toolbar, click **Attributes** and choose **Prescribed Pressure**.

2 In the **Settings** window for **Prescribed Pressure**, locate the **Prescribed Pressure** section.

3 In the *p* text field, type `pressure(t)`.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Structural steel**.

4 Right-click and choose **Add to Component 1 (comp1)**.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Structural steel (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Steel Vessel**.

Nitrile butadiene rubber (NBR)

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Nitrile butadiene rubber (NBR)` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Diaphragm**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Lamé parameter μ	muLame	`0.7[MPa]`	N/m^2	Lamé parameters
Density	rho	`1100`	kg/m^3	Basic

Thermoset

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Materials** > **Nitrile butadiene rubber (NBR) (mat2)** node.

2 Right-click **Materials** and choose **Blank Material**.

3 In the **Settings** window for **Material**, type `Thermoset` in the **Label** text field.

4 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Diaphragm Plate**.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`1.1[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.35`	1	Young's modulus and Poisson's ratio
Density	rho	`1300`	kg/m^3	Basic

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 5 and 6 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely coarse**.

Size 2

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 3, 4, 7, and 9 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `shell.th/2`.

8 Select the **Minimum element size** checkbox. In the associated text field, type `shell.th/2`.

Size Expression 1

1 Right-click **Free Triangular 1** and choose **Size Expression**.

2 In the **Settings** window for **Size Expression**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1 and 2 only.

5 Locate the **Element Size Expression** section. In the **Size expression** text field, type `if(R>10[mm], 0.22, 0.5)*dia.th`.

Distribution 1

1 Right-click **Free Triangular 1** and choose **Distribution**.

2 Select Boundaries 50, 51, 53, 54, and 58 only.

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

4 In the **Number of elements** text field, type `3`.

5 Click **Build All**.

PRECHARGE

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

2 In the **Settings** window for **Study**, type `Precharge` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Precharge** 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** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid), Controls spatial frame** > **Enclosed Cavity, Nitrogen Gas** > **Fluid 1** and **Component 1 (comp1)** > **Solid Mechanics (solid), Controls spatial frame** > **Enclosed Cavity, Hydraulic Fluid** > **Prescribed Pressure 1**.

5 Right-click and choose **Disable**.

6 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Stepping parameter)	`range(0, 5e-4, 8e-3), range(0.01, 0.01, 0.1), 0.5, 0.7, 1`	

9 In the **Study** toolbar, click **Compute**.

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Click **Apply**.

Stress (Precharge)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Stress (Precharge)` in the **Label** text field.

Surface 2

1 In the **Model Builder** window, expand the **Stress (Precharge)** node.

2 Right-click **Results** > **Stress (Precharge)** > **Surface 1** and choose **Duplicate**.

3 In the **Settings** window for **Surface**, click to expand the **Title** section.

4 From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Twilight**.

Selection 1

1. Right-click **Surface 2** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Diaphragm**.
4. In the **Stress (Precharge)** toolbar, click **Plot**.

Stress, 3D (Precharge)

1. In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.
2. In the **Settings** window for **3D Plot Group**, type `Stress, 3D (Precharge)` in the **Label** text field.

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** > **Time Dependent**.
4. Right-click and choose **Add Study**.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

HYDRAULIC FLUID

In the **Settings** window for **Study**, type `Hydraulic Fluid` in the **Label** text field.

Step 1: Time Dependent

1. In the **Model Builder** window, under **Hydraulic Fluid** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. In the **Output times** text field, type `range(0,t_end/100,t_end)`.
4. Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
5. From the **Method** list, choose **Solution**.
6. From the **Study** list, choose **Precharge, Stationary**.
7. From the **Parameter value (para)** list, choose **Last**.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

The diaphragm deformation involves a local snap-through problem. Change to the BDF method, which handles this problem better as it introduces more numerical damping.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Method** list, choose **BDF**.

5 From the **Steps taken by solver** list, choose **Strict**.

6 In the **Model Builder** window, expand the **Hydraulic Fluid** > **Solver Configurations** > **Solution 2 (sol2)** > **Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

7 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

8 From the **Nonlinear method** list, choose **Constant (Newton)**.

9 From the **Jacobian update** list, choose **On every iteration**.

10 Click **Run**.

RESULTS

Stress (Hydraulic Fluid)

1 In the **Settings** window for **2D Plot Group**, type `Stress (Hydraulic Fluid)` in the **Label** text field.

2 Locate the **Data** section. From the **Time (s)** list, choose **15**.

Surface 2

1 In the **Model Builder** window, expand the **Stress (Hydraulic Fluid)** node.

2 Right-click **Results** > **Stress (Hydraulic Fluid)** > **Surface 1** and choose **Duplicate**.

3 In the **Settings** window for **Surface**, locate the **Title** section.

4 From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Twilight**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

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

3 From the **Selection** list, choose **Diaphragm**.

4 In the **Stress (Hydraulic Fluid)** toolbar, click **Plot**.

Stress, 3D (Hydraulic Fluid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Stress, 3D (Hydraulic Fluid)` in the **Label** text field.

RESULTS

Fluid Pressure

Add a plot showing the pressure in the nitrogen gas and the hydraulic fluid.

1 In the **Model Builder** window, expand the **Hydraulic Fluid** > **Solver Configurations** > **Solution 2 (sol2)** > **Time-Dependent Solver 1** node.

2 Right-click **Results** and choose **1D Plot Group**.

3 In the **Settings** window for **1D Plot Group**, type `Fluid Pressure` in the **Label** text field.

4 Click to expand the **Title** section. Locate the **Data** section. From the **Dataset** list, choose **Hydraulic Fluid/Solution 2 (sol2)**.

5 Locate the **Title** section. From the **Title type** list, choose **Label**.

6 Locate the **Plot Settings** section.

7 Select the **y-axis label** checkbox. In the associated text field, type `Relative pressure (bar)`.

Global 1

1 Right-click **Fluid Pressure** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
solid.enc1.p_rel	bar	Nitrogen
solid.enc2.p_rel	bar	Hydraulic oil

4 In the **Fluid Pressure** toolbar, click **Plot**.

Volume, Hydraulic Fluid

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Volume, Hydraulic Fluid` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Hydraulic Fluid/Solution 2 (sol2)**.

4 Locate the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

1 Right-click **Volume, Hydraulic Fluid** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`solid.enc2.DeltaV-at(0[s], solid.enc2.DeltaV)`	`ml`	`Volume Change`

4 In the **Volume, Hydraulic Fluid** toolbar, click **Plot**.

Graph Marker 1

1 Right-click **Global 1** and choose **Graph Marker**.

2 In the **Settings** window for **Graph Marker**, locate the **Display** section.

3 From the **Display** list, choose **Max**.

4 Locate the **Text Format** section. Select the **Include unit** checkbox.

5 In the **Precision** text field, type `3`.

6 Click to expand the **Coloring and Style** section. From the **Anchor point** list, choose **Upper middle**.

7 In the **Volume, Hydraulic Fluid** toolbar, click **Plot**.

Temperature, Nitrogen Gas

1 In the **Model Builder** window, right-click **Volume, Hydraulic Fluid** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Temperature, Nitrogen Gas` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Temperature, Nitrogen Gas** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 Click **Clear Table**.

4 In the table, enter the following settings:

Expression	Unit	Description
solid.enc1.fl1.T	°C	Temperature

5 In the **Temperature, Nitrogen Gas** toolbar, click **Plot**.

Diaphragm Accumulator, 3D

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Diaphragm Accumulator, 3D` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Revolution 2D 2**.

4 From the **Time (s)** list, choose **15**.

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

6 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (r, phi, z)**.

Surface 1

1 Right-click **Diaphragm Accumulator, 3D** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

Material Appearance 1

1 Right-click **Surface 1** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Steel**.

Deformation 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 Select Domains 3–9 only.

Surface 2

Right-click **Surface 1** and choose **Duplicate**.

Material Appearance 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Material Appearance 1**.
2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.
3 From the **Material type** list, choose **Rubber**.
4 From the **Color** list, choose **Black**.

Selection 1

1 In the **Model Builder** window, click **Selection 1**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 From the **Selection** list, choose **Diaphragm**.

Surface 3

In the **Model Builder** window, under **Results** > **Diaphragm Accumulator, 3D** right-click **Surface 2** and choose **Duplicate**.

Material Appearance 1

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Material Appearance 1**.
2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.
3 From the **Material type** list, choose **Plastic**.
4 From the **Color** list, choose **Gray**.

Selection 1

1 In the **Model Builder** window, click **Selection 1**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 From the **Selection** list, choose **Diaphragm Plate**.

Diaphragm Accumulator, 3D

1 Click the **Zoom Extents** button in the **Graphics** toolbar.
2 Click the **Go to Default View** button in the **Graphics** toolbar.
3 Click the **Show Grid** button in the **Graphics** toolbar.
4 In the **Model Builder** window, under **Results** click **Diaphragm Accumulator, 3D**.
5 In the **Settings** window for **3D Plot Group**, in the **Graphics** window toolbar, click ▼ next to **Scene Light**, then choose **Ambient Occlusion**.
6 In the **Graphics** window toolbar, click ▼ next to **Scene Light**, then choose **Indoor**.

7 In the **Graphics** window toolbar, click ▾ next to **Scene Light**, then choose **Gamma Correction**.

8 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

9 In the **Diaphragm Accumulator, 3D** toolbar, click **Plot**.