Created in COMSOL Multiphysics 6.2

Impact Between Two Soft Rings

Introduction

In this conceptual example, the soft impact between two elastic rings is modeled using the Solid Mechanics interface. The goal is to verify that the contact formulations conserve fundamental thermodynamic quantities such as momentum (linear and angular) and energy. The setup of the problem is based on a benchmark example from Ref. 1.

Model Definition

As illustrated in Figure 1, the 2D geometry consists of two thin rings. Both rings are 30 cm thick and have an outer diameter of 20 m.

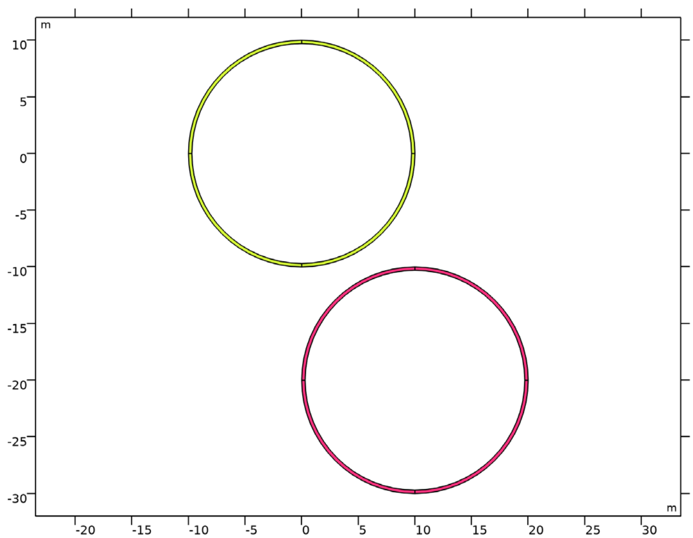

Figure 1: Model geometry.

The material of the rings is considered to be linear elastic, with material properties according to Table 1.

TABLE 1: MATERIAL PROPERTIES.

PROPERTY	VALUE
Young’s modulus	20 MPa
Poisson's ratio	1/6
Density	1000 kg/m^3

The pressure contact is modeled using two different formulations available in COMSOL Multiphysics: **Penalty, dynamic** and **Augmented Lagrangian, dynamic**. Both formulations are based on a viscous implementation, where the rate of the gap distance between the source and destination boundaries is constrained to zero once contact is detected. Such formulation assumes that the event dissipates energy, and it is therefore mainly suitable for short impact events.

The uppermost ring is given an initial vertical velocity of -4 m/s to initiate the impact event. No other constraints nor external forces are considered in the model.

Results and Discussion

Figure 2 shows snapshots of the deformed shape of the two rings for every 5 seconds. It can be seen that both rings deform significantly once ring 1 impacts ring 2, and that after the impact the ring 2 gained some of the momentum from ring 1. Also, both rings start to rotate after the impact. It can also be seen that the dynamic penalty method generates a significant overlap between the two geometries during contact. This is expected form the method, since constraints are only added in the step after contact is detected.

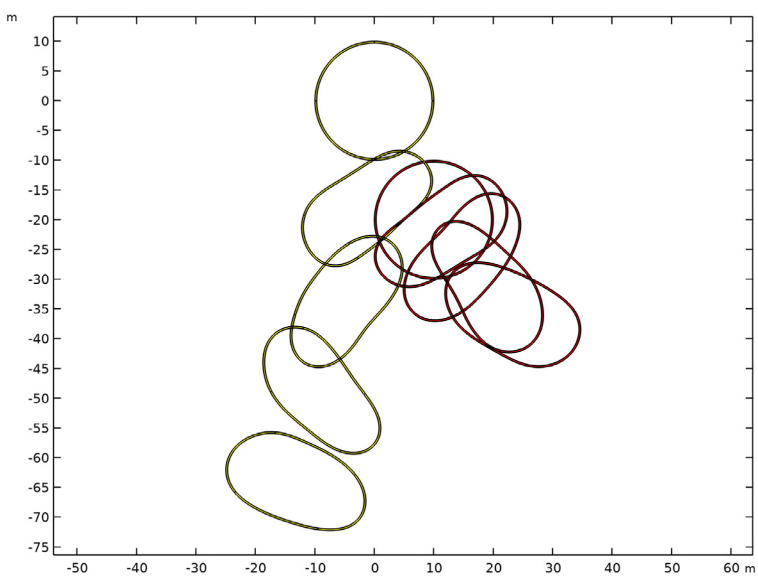

Figure 2: Computed deformation of the two rings using the dynamic penalty formulation plotted every 5 seconds.

The impact event can be studied in detail by examining different thermodynamic quantities. Figure 3 and Figure 4 show the linear momentum for each ring and for the total assembly. It can be observed how the initial linear momentum in the y direction of ring 1 is partially transferred to ring 2. During the impact, both rings also gain momentum in the x direction but in opposite directions. Importantly, it can be observed that the linear momentum of the assembly is conserved during the impact event. From Figure 5 it can be seen that both rings also gain angular momentum, which corresponds to the observed rotational movement of the rings. The total angular momentum is also properly conserved.

Figure 6 depicts the energy balance of the assembly and different components of the total energy. Parts of the initial kinetic energy of ring 1 is converted to elastic energy during and after the impact, as both rings deform. Also, the contact between the rings dissipate energy since a viscous formulation is used. The total energy is kept constant throughout the simulation, meaning that the model is consistent in terms of energy.

The conclusions obtained after inspecting the results in Figure 3 to Figure 6 apply to both the dynamic penalty and dynamic augmented Lagrangian formulations. However, the results differ slightly, given that the two formulations enforce the contact constraint in different ways.

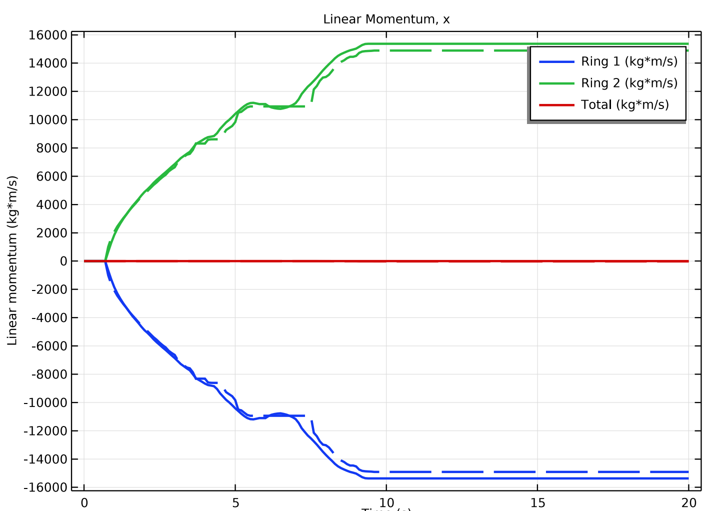

Figure 3: Variation of linear momentum in the x direction. Solid lines correspond to the penalty formulation and dashed lines to the augmented Lagrangian formulation.

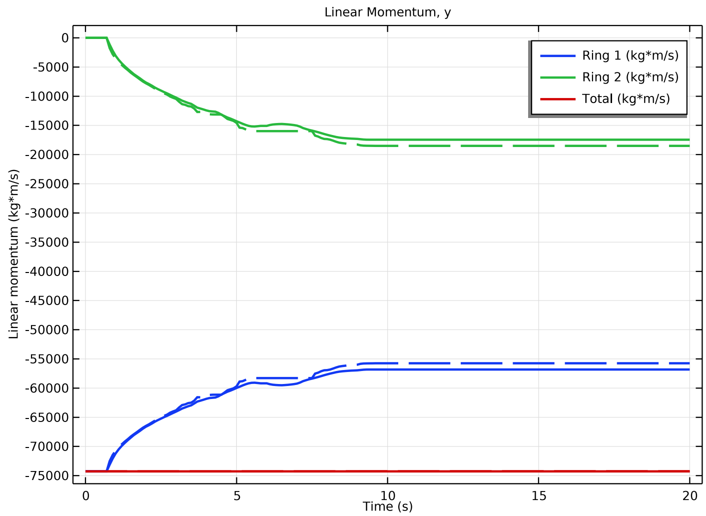

Figure 4: Variation of linear momentum in the y direction. Solid lines correspond to the penalty formulation and dashed lines to the augmented Lagrangian formulation.

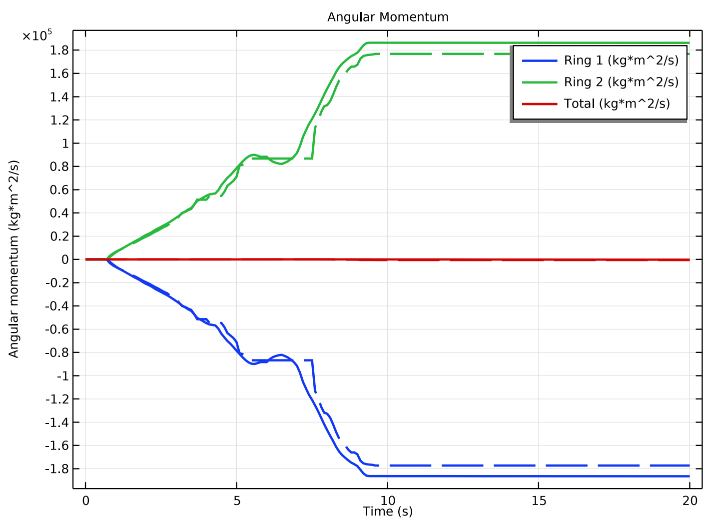

Figure 5: Variation of angular momentum. Solid lines correspond to the penalty formulation and dashed lines to the augmented Lagrangian formulation.

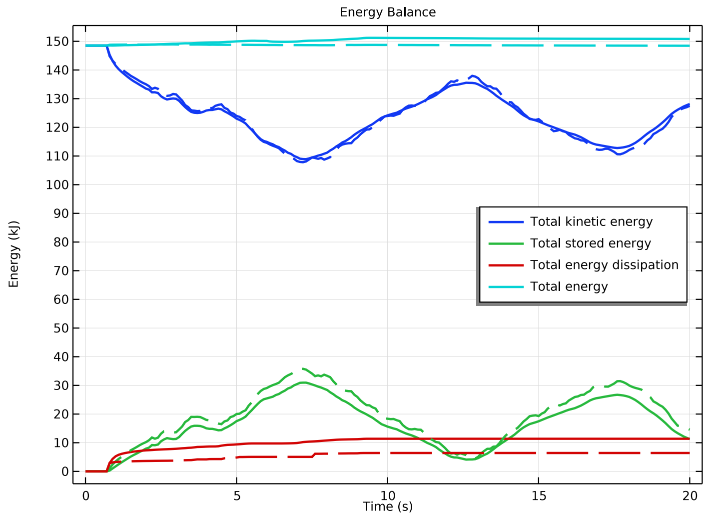

Figure 6: Variation of different energy quantities. Solid lines correspond to the penalty formulation and dashed lines to the augmented Lagrangian formulation.

Notes About the COMSOL Implementation

The two viscous formulations for contact use a penalty factor that is interpreted as an artificial viscosity. Hence, a time scale has to be given as a user input. This characteristic time can be related to the duration of the impact event, or used as a multiplier to find a stable definition of the contact model.

It is often a good practice to use a manual time-step control for this type of dynamic simulation, when the kinematics of the contacting bodies is the main interest. Otherwise, the error estimates used by the automatic time-step control will enforce unnecessary small time steps to resolve the wave propagation within the solid, which is not of primary interest.

Reference

1. T.A. Laursen, *Computational Contact and Impact Mechanics: Fundamentals of Modelling Interfacial Phenomena in Nonlinear Finite Element Analysis*, Springer-Verlag, 2002.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/ring_impact`

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**.

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>Time Dependent**.

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
radius	10[m]	10 m	Ring radius
thickness	0.3[m]	0.3 m	Ring thickness

GEOMETRY 1

Ring 1

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

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

3 In the **Radius** text field, type `radius`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`thickness`

5 In the **Label** text field, type `Ring 1`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Color** list, choose **Color 4**.

8 Click **Build All Objects**.

9 Right-click **Ring 1** and choose **Duplicate**.

Ring 2

1 In the **Model Builder** window, click **Ring 1.1 (c2)**.

2 In the **Settings** window for **Circle**, locate the **Position** section.

3 In the **x** text field, type `10`.

4 In the **y** text field, type `-20`.

5 In the **Label** text field, type `Ring 2`.

6 Locate the **Selections of Resulting Entities** section. From the **Color** list, choose **Color 12**.

Delete Entities 1 (del1)

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

2 Click in the **Graphics** window and then press Ctrl+D to clear all objects.

3 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

5 On the object **c1**, select Domain 5 only.

6 On the object **c2**, select Domain 5 only.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

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

DEFINITIONS

Contact Pair 1 (p1)

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

2 Select Boundaries 14, 19, 21, and 24 only.

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

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 9, 10, 15, and 18 only.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	20[MPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	1/6	1	Young's modulus and Poisson's ratio
Density	rho	1000	kg/m³	Basic

SOLID MECHANICS (SOLID)

Initial Values 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose the domain setting **More>Initial Values**.

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

3 From the **Selection** list, choose **Ring 1**.

4 Set the initial velocity to -4 [m/s] in the Y direction.

Contact 1

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

2 In the **Settings** window for **Contact**, locate the **Contact Method** section.

3 From the list, choose **Penalty, dynamic**.

For impact problems, a purely viscous penalty formulation is preferable.

4 Locate the **Contact Pressure Penalty Factor** section. From the **Penalty factor control** list, choose **Viscous only**.

5 In the τ_n text field, type `.1[ms]`.

Enable **Advanced Physics Options** to compute energy dissipation.

6 Click the **Show More Options** button in the **Model Builder** toolbar.

7 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

8 Click **OK**.

9 In the **Settings** window for **Contact**, click to expand the **Advanced** section.

10 Select the **Compute viscous contact dissipation** check box.

Also, solve the model using the dynamic augmented Lagrangian formulation.

11 Right-click **Contact 1** and choose **Duplicate**.

Contact 2

1 In the **Model Builder** window, click **Contact 2**.

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

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Method** section.

7 From the list, choose **Augmented Lagrangian, dynamic**.

8 Locate the **Contact Pressure Penalty Factor** section. In the τ_n text field, type `.01[ms]`.

MESH 1

Mapped 1

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

Distribution 1

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

2 Select Boundaries 3 and 7 only.

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

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

Distribution 2

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

2 Select Boundaries 9, 10, 13–15, 18, 21, and 24 only.

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

4 In the **Number of elements** text field, type `20`.

5 Click **Build All**.

STUDY 1: PENALTY

Use the dynamic penalty formulation in the first study.

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

2 In the **Settings** window for **Study**, type `Study 1: Penalty` in the **Label** text field.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1: Penalty** 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(0,0.1,20)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 2**.

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

Modify the time-dependent solver to use a manual time step control.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** 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 **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `0.05`.

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

RESULTS

Deformation (Penalty)

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

Surface 1

1 In the **Model Builder** window, expand the **Deformation (Penalty)** node, then click **Surface 1**.

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

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

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

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Yellow**.

Selection 1

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

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

3 From the **Selection** list, choose **Ring 1**.

Surface 1

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

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **Red**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Ring 2**.

4 In the **Deformation (Penalty)** toolbar, click **Plot**.

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

Deformation (Penalty)

In the **Model Builder** window, under **Results** click **Deformation (Penalty)**.

Animation 1

1 In the **Deformation (Penalty)** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 In the **Number of frames** text field, type 100.

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

5 Click the Play button in the **Graphics** toolbar.

ADD STUDY

Create a second study for the dynamic augmented Lagrangian formulation.

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

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

STUDY 2: AUGMENTED LAGRANGIAN

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

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

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2: Augmented Lagrangian** 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(0,0.1,20)`.

Solution 2 (sol2)

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

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**, locate the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `0.05`.

Increase the number of iterations allowed for the segregated solver.

6 In the **Model Builder** window, expand the **Study 2: Augmented Lagrangian> Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1** node, then click **Segregated 1**.

7 In the **Settings** window for **Segregated**, locate the **General** section.

8 In the **Maximum number of iterations** text field, type `50`.

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

RESULTS

Deformation (AugLag)

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

Surface 1

1 In the **Model Builder** window, expand the **Deformation (AugLag)** node, then click **Surface 1**.

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

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

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

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Yellow**.

Selection 1

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

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

3 From the **Selection** list, choose **Ring 1**.

Surface 1

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

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **Red**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Ring 2**.

4 In the **Deformation (AugLag)** toolbar, click **Plot**.

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

Deformation (AugLag)

In the **Model Builder** window, under **Results** click **Deformation (AugLag)**.

Animation 2

1 In the **Deformation (AugLag)** toolbar, click **Animation** and choose **Player**.
2 Click the **Zoom Extents** button in the **Graphics** toolbar.
3 In the **Settings** window for **Animation**, locate the **Frames** section.
4 In the **Number of frames** text field, type 100.
5 Click the **Play** button in the **Graphics** toolbar.

Energy Balance

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type Energy Balance in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Global 1

1 Right-click **Energy Balance** and choose **Global**.
2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Global>solid.Wk_tot - Total kinetic energy - J**.
3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Global>solid.Wh_tot - Total stored energy - J**.
4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Global>solid.Wd_tot - Total energy dissipation - J**.
5 In the **Energy Balance** toolbar, click **Plot**.
6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
solid.Wk_tot	kJ	Total kinetic energy
solid.Wh_tot	kJ	Total stored energy
solid.Wd_tot	kJ	Total energy dissipation
solid.Wk_tot+solid.Wh_tot+solid.Wd_tot	kJ	Total energy

7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

Plot the energy balance for the dynamic augmented Lagrangian formulation using dashed lines.

8 Right-click **Global 1** and choose **Duplicate**.

Global 2

1 In the **Model Builder** window, click **Global 2**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2: Augmented Lagrangian/Solution 2 (sol2)**.
4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
5 From the **Color** list, choose **Cycle (reset)**.
6 Click to expand the **Legends** section. Clear the **Show legends** check box.

Energy Balance

1 In the **Model Builder** window, click **Energy Balance**.
2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3 Select the **y-axis label** check box. In the associated text field, type `Energy (kJ)`.
4 Locate the **Legend** section. From the **Position** list, choose **Middle right**.
5 In the **Energy Balance** toolbar, click **Plot**.

Create evaluation groups to compute the linear and angular momentum of the rings.

Linear Momentum, x (Penalty)

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, type `Linear Momentum, x (Penalty)` in the **Label** text field.

Surface Integration 1

1 Right-click **Linear Momentum, x (Penalty)** and choose **Integration>Surface Integration**.
2 In the **Settings** window for **Surface Integration**, locate the **Selection** section.
3 From the **Selection** list, choose **Ring 1**.
4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
solid.rho*ut*solid.d	kg*m/s	Ring 1

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

Surface Integration 2

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

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

Expression	**Unit**	**Description**
solid.rho*ut*solid.d	kg*m/s	Ring 2

5 Right-click **Surface Integration 2** and choose **Duplicate**.

Surface Integration 3

1 In the **Model Builder** window, click **Surface Integration 3**.

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

3 From the **Selection** list, choose **All domains**.

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

Expression	**Unit**	**Description**
solid.rho*ut*solid.d	kg*m/s	Total

5 In the **Linear Momentum, x (Penalty)** toolbar, click **Evaluate**.

Linear Momentum, x (Penalty)

In the **Model Builder** window, right-click **Linear Momentum, x (Penalty)** and choose **Duplicate**.

Linear Momentum, x (AugLag)

1 In the **Model Builder** window, under **Results** click **Linear Momentum, x (Penalty) 1**.

2 In the **Settings** window for **Evaluation Group**, type `Linear Momentum, x (AugLag)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Augmented Lagrangian/ Solution 2 (sol2)**.

4 In the **Linear Momentum, x (AugLag)** toolbar, click **Evaluate**.

Linear Momentum, x (Penalty)

In the **Model Builder** window, right-click **Linear Momentum, x (Penalty)** and choose **Duplicate**.

Linear Momentum, y (Penalty)

1 In the **Model Builder** window, under **Results** click **Linear Momentum, x (Penalty) 1**.

2 In the **Settings** window for **Evaluation Group**, type `Linear Momentum, y (Penalty)` in the **Label** text field.

Surface Integration 1

1 In the **Model Builder** window, expand the **Linear Momentum, y (Penalty)** node, then click **Surface Integration 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
solid.rho*vt*solid.d	kg*m/s	Ring 1

Surface Integration 2

1 In the **Model Builder** window, click **Surface Integration 2**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
solid.rho*vt*solid.d	kg*m/s	Ring 2

Surface Integration 3

1 In the **Model Builder** window, click **Surface Integration 3**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
solid.rho*vt*solid.d	kg*m/s	Total

4 In the **Linear Momentum, y (Penalty)** toolbar, click **= Evaluate**.

Linear Momentum, y (Penalty)

In the **Model Builder** window, right-click **Linear Momentum, y (Penalty)** and choose **Duplicate**.

Linear Momentum, y (AugLag)

1 In the **Model Builder** window, under **Results** click **Linear Momentum, y (Penalty) 1**.

2 In the **Settings** window for **Evaluation Group**, type `Linear Momentum, y (AugLag)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Augmented Lagrangian/ Solution 2 (sol2)**.

4 In the **Linear Momentum, y (AugLag)** toolbar, click **= Evaluate**.

Linear Momentum, x (Penalty)

In the **Model Builder** window, right-click **Linear Momentum, x (Penalty)** and choose **Duplicate**.

Angular Momentum (Penalty)

1 In the **Model Builder** window, expand the **Results>Linear Momentum, x (Penalty) 1** node, then click **Linear Momentum, x (Penalty) 1**.

2 In the **Settings** window for **Evaluation Group**, type `Angular Momentum (Penalty)` in the **Label** text field.

Surface Integration 1

1 In the **Model Builder** window, click **Surface Integration 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`solid.rho*(x*vt-y*ut)*solid.d`	`kg*m^2/s`	`Ring 1`

Surface Integration 2

1 In the **Model Builder** window, click **Surface Integration 2**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`solid.rho*(x*vt-y*ut)*solid.d`	`kg*m^2/s`	`Ring 2`

Surface Integration 3

1 In the **Model Builder** window, click **Surface Integration 3**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`solid.rho*(x*vt-y*ut)*solid.d`	`kg*m^2/s`	`Total`

4 In the **Angular Momentum (Penalty)** toolbar, click **Evaluate**.

Angular Momentum (Penalty)

In the **Model Builder** window, right-click **Angular Momentum (Penalty)** and choose **Duplicate**.

Angular Momentum (AugLag)

1 In the **Model Builder** window, under **Results** click **Angular Momentum (Penalty) 1**.

2 In the **Settings** window for **Evaluation Group**, type `Angular Momentum (AugLag)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Augmented Lagrangian/ Solution 2 (sol2)**.

4 In the **Angular Momentum (AugLag)** toolbar, click **Evaluate**.

Linear Momentum, x

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

2 In the **Settings** window for **1D Plot Group**, type `Linear Momentum, x` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Linear momentum (kg*m/s)`.

Table Graph 1

1 Right-click **Linear Momentum, x** and choose **Table Graph**.

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

3 From the **Source** list, choose **Evaluation group**.

4 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

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

6 Right-click **Table Graph 1** and choose **Duplicate**.

Table Graph 2

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

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

3 From the **Evaluation group** list, choose **Linear Momentum, x (AugLag)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 From the **Color** list, choose **Cycle (reset)**.

6 Locate the **Legends** section. Clear the **Show legends** check box.

7 In the **Linear Momentum, x** toolbar, click **Plot**.

Linear Momentum, x

In the **Model Builder** window, right-click **Linear Momentum, x** and choose **Duplicate**.

Linear Momentum, y

1 In the **Model Builder** window, under **Results** click **Linear Momentum, x 1**.

2 In the **Settings** window for **1D Plot Group**, type `Linear Momentum, y` in the **Label** text field.

Table Graph 1

1 In the **Model Builder** window, expand the **Linear Momentum, y** node, then click **Table Graph 1**.

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

3 From the **Evaluation group** list, choose **Linear Momentum, y (Penalty)**.

Table Graph 2

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

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

3 From the **Evaluation group** list, choose **Linear Momentum, y (AugLag)**.

4 In the **Linear Momentum, y** toolbar, click **Plot**.

Linear Momentum, y

In the **Model Builder** window, right-click **Linear Momentum, y** and choose **Duplicate**.

Angular Momentum

1 In the **Model Builder** window, under **Results** click **Linear Momentum, y 1**.

2 In the **Settings** window for **1D Plot Group**, type `Angular Momentum` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Angular momentum (kg*m^2/s)`.

Table Graph 1

1 In the **Model Builder** window, expand the **Angular Momentum** node, then click **Table Graph 1**.

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

3 From the **Evaluation group** list, choose **Angular Momentum (Penalty)**.

Table Graph 2

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

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

3 From the **Evaluation group** list, choose **Angular Momentum (AugLag)**.

4 In the **Angular Momentum** toolbar, click **Plot**.