Altair MotionView 2019 Tutorials

MV-1029: Modeling Point-to-Deformable-Surface Force (PTdSF)

MV-1029: Modeling Point-to-Deformable-Surface Force (PTdSF)

In this tutorial, you will learn how to:

- Model a PTdSFforce (point-to-deformable-surface) joint with a contact force

A PTdSFforce (point-to-deformable-surface) joint is a higher pair constraint with an added contact force. The force is either modeled as a linear one or a Poisson type here. This constraint restricts a specified point on a body to move along a specified deformable surface on another body. The point may belong to a rigid, flexible or point body. The deformable surface for this tutorial is defined on a rigidly supported plate. A mass is constrained to move on the surface with a PTdSFforce constraint. The added feature here is that a flexible contact force acts at the center of mass of the ball between it and the deformable surface. In this tutorial we will take up the case of the linear force model.

Exercise

Copy the following files `Plate.h3d` and `membrane.fem`, located in the `mbd_modeling\interactive` folder, to your `<working directory>`.

Step 1: Creating points.

Let's start with creating points that will help us locate the bodies and joints as required. We will define points for center of mass of the bodies and joint locations.

1. Start a new MotionView Session. We will work with the default units (kg, mm, s, N).

2. From the **Project Browser** right-click on *Model* and select *Add Reference Entity > Point* (or right-click the *Points* icon 🟡 on the **Model-Reference** toolbar).

   The **Add Point or PointPair** dialog is displayed.

3. For **Label**, enter `BallCM`.

4. Accept the default variable name and click *OK*.

5. Click on the *Properties* tab and specify the coordinates as **X** = `0.0`, **Y** = `0.0`, and **Z** = `50.0`.

6. Follow the same procedure for the other points specified in the following table:

Point	X	Y	Z
PointMembInterface39	-55.00	-55.00	0.0
PointMembInterface40	55.00	-55.00	0.0
PointMembInterface41	55.00	55.00	0.0
PointMembInterface42	-55.00	55.00	0.0

△ Altair

Step 2: Creating Bodies.

We will have two bodies apart from the ground body in our model visualization: the membrane and the ball. Pre-specified inertia properties will be used to define the ball.

1. From the **Project Browser** right-click on *Model* and select *Add Reference Entity > Body* (or right-click the *Body* icon 🌐 on the **Model-Reference** toolbar).

   The **Add Body or BodyPair** dialog is displayed.

2. For **Label**, enter `Membrane` and click *OK*.

3. Accept the default variable name and click *OK*.

   For the remainder of this tutorial - accept the default names that are provided for the rest of the variables that you will be asked for.

4. From the **Properties** tab, check the *Flex Body (CMS)* box.

5. Click on the *Graphic file* browser icon 📂 and select `Plate.h3d` from the `<working directory>`.

   The same path will automatically appear next to the **H3D file** browser icon 📂.

6. Right-click on *Bodies* in the **Project Browser** and select *Add Body*.

   The **Add Body or BodyPair** dialog is displayed.

7. For **Label**, enter `Ball` and click *OK*.

8. From the **Properties** tab, specify the following for the **Ball**:

Body	Mass	Ixx	Iyy	Izz	Ixy	Iyz	Izx
Ball	1.00	40000.00	40000.00	40000.00	0.0	0.0	0.0

9. For the **Ball** body, under the **CM Coordinates** tab, check the **Use center of mass coordinate system** box.

10. Double click on *Point*.

    The **Select a Point** dialog is displayed.

11. Choose *BallCM* and click *OK*.

12. Accept defaults for axes orientation properties.

△ Altair

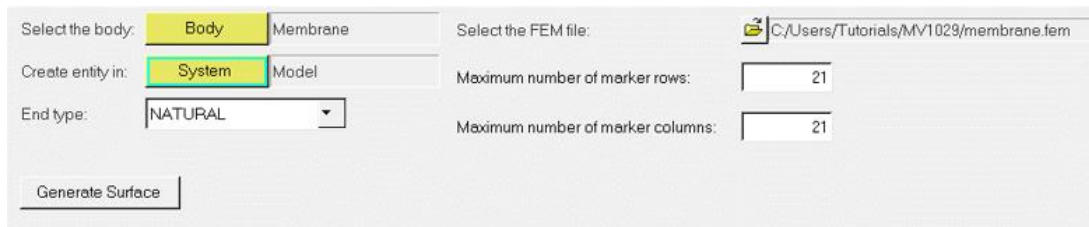### Step 3: Creating Markers and a Deformable Surface.

Now, we will define some markers required for the membrane.

1. From the **Macros** menu, select *Create Deformable Surface From FEM*.
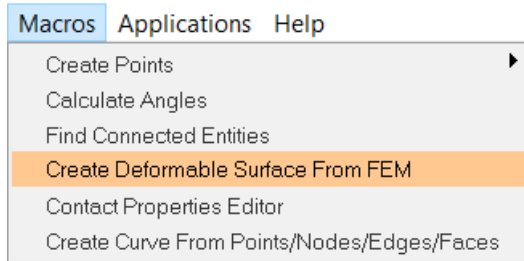
   The **Deformable Surface From FEM** panel is displayed.

2. For **Select the Body**, use the **Body** input collector to select *Membrane*.

3. Click on the *Select the FEM file* file browser icon and select the `membrane.fem` file.

4. For **Create entity in**, double click on the *System* collector and select the *Model* system.

5. Use the default values for the **Maximum number of marker rows** and **Maximum number of marker columns**.

6. Click *Generate Surface*.

   The Markers and Deformable Surface are created.

Step 4: Creating Joints.

Here, we will define all the necessary joints. We require four joints for the model, all of them being fixed joints between the membrane and the ground.

1. From the **Project Browser** right-click on *Model* and select *Add Constraint > Joint* (or right-click the *Joints* icon on the **Model-Constraint** toolbar).

   The **Add Joint or JointPair** dialog is displayed.

2. For **Label**, enter `Joint 1`.

3. Select *Fixed Joint* as the type and click *OK*.

4. From the **Connectivity** tab, double-click on *Body 1*.

   The **Select a Body** dialog is displayed.

5. Choose *Membrane* and click *OK*.

6. From the **Connectivity** tab, double-click on *Body 2*.

   The **Select a Body** dialog is displayed.

7. Choose *Ground Body* and click *OK*.

8. From the **Connectivity** tab, double-click on *Point*.

   The **Select a Point** dialog is displayed.

9. Choose *PointMembInterface39* and click *OK*.

10. Repeat the same procedure for the other three joints.

    A table is provided below for your convenience:

Label	Type of Joint	Body 1	Body 2	Point
Joint 2	**Fixed**	**Membrane**	**Ground Body**	**PointMembInterface40**
Joint 3	**Fixed**	**Membrane**	**Ground Body**	**PointMembInterface41**
Joint 4	**Fixed**	**Membrane**	**Ground Body**	**PointMembInterface42**

Step 5: Creating Contacts.

Here we will define the contact force between the deformable membrane and the ball.

1. From the **Project Browser** right-click on *Model* and select *Add Force Entity >
   Contact* (or right-click the *Contacts* icon on the **Model-Force** toolbar).

   The **Add Contact** dialog is displayed.

2. For **Label**, enter Contact 0.

3. Select *PointToDeformableSurfaceContact* for the type of contact and click *OK*.

4. From the **Connectivity** tab; select *Ball* for **Body**, *BallCM* for **Point**, and
   DeformableSurface 1 for **DeformableSurface**.

5. Click on the *Properties* tab and enter 10 for Radius, 1000 for *Stiffness*, and 0.2 for
   Damping.

6. Uncheck the **Flip normal** check box.

Step 6: Creating Graphics.

Graphics for the ball will now be built here.

1. From the **Project Browser** right-click on *Model* and select *Add Reference Entity
   > Graphic* (or right-click the *Graphics* icon on the **Model-Reference** toolbar).

   The **Add Graphics or GraphicPair** dialog is displayed.

2. For **Label**, enter Ball.

3. For **Type**, choose *Sphere* from the drop-down menu and click *OK*.

4. From the **Connectivity** tab, double-click on *Body*.

   The **Select a Body** dialog is displayed.

5. Choose *Ball* and click *OK*.

6. Again from the **Connectivity** tab, double-click on *Point*.

   The **Select a Point** dialog is displayed.

7. Choose *BallCM* and click *OK*.

8. From the **Properties** tab, enter 10 as the radius of the **Ball**.

9. From the **Visualization** tab, select a color for the **Ball**.

Step 7: Return to the Bodies Panel.

1. Click the *Body* icon 🌐 on the **Model-Reference** toolbar.

2. Select the *Membrane Body* from the graphics area.

   The panel is displayed.

3. Click on the *Nodes* button.

   The **Nodes** dialog is displayed.

4. Uncheck the *Only search interface nodes* box and then click on *Find All*.

5. Close the **Nodes** dialog.

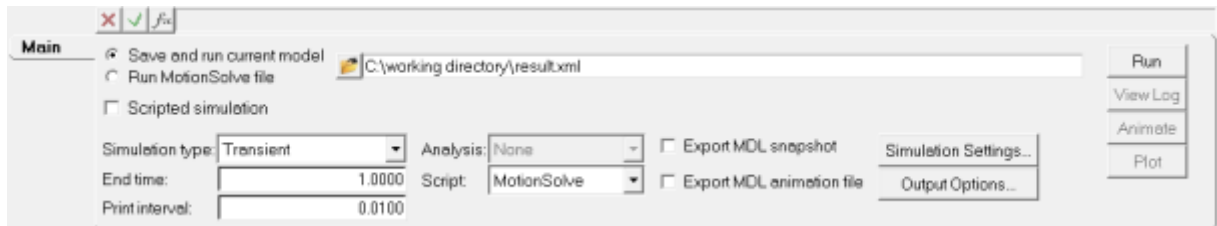   At the end of these steps your model should look like the one shown in the figure below:
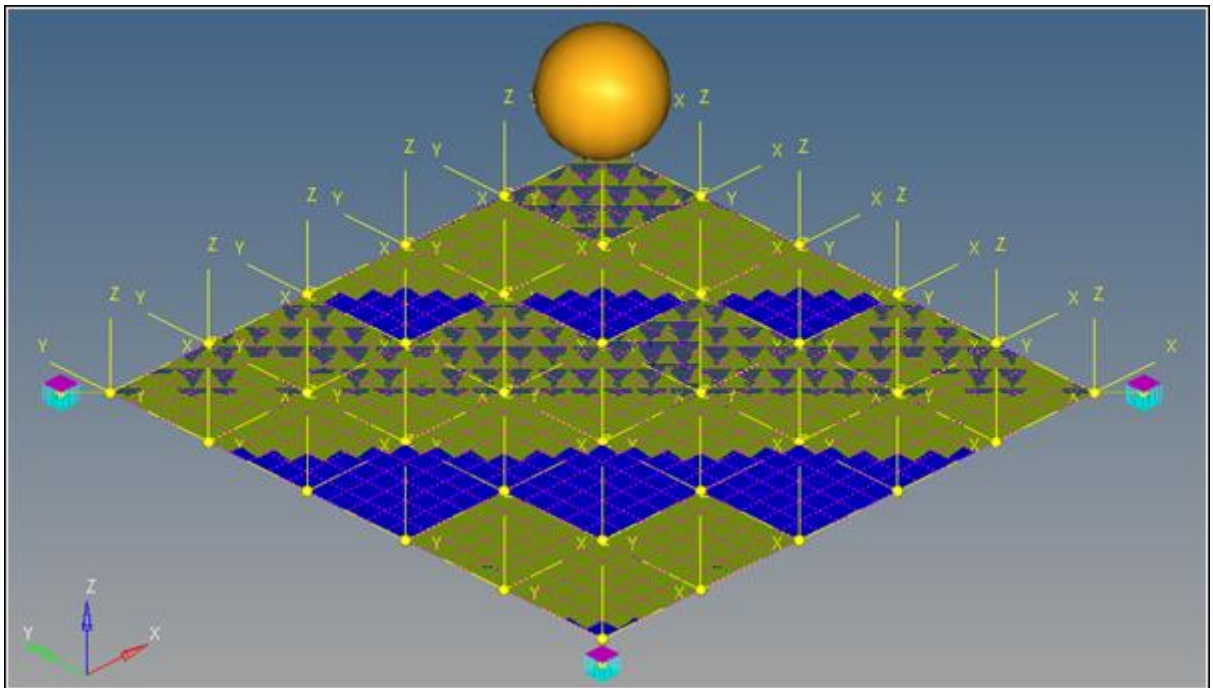
6. Save the model as `ptdsf_contact.mdl`.

Step 8: Running the Model.

Now we have the model defined completely and it is ready to run.

1. Click the ***Run*** icon (RUN) on the **Model-Main** toolbar.

   The **Run** panel gets displayed.

2. From the **Main** tab, specify values as shown below:

3. Choose the ***Save and run current model*** radio button.

4. Click on the browser icon 📂 and save the file as `result.xml` in the `<working directory>`.

5. Click ***Save***.

6. Click the ***Check Model*** button ✔ on the **Model Check** toolbar to check the model for errors.

7. To run the model, click the ***Run*** button on the panel.

   The solver is invoked.

8. Once the run is complete, ***Close*** the solver window and ***Clear*** the Message Log.

Step 9: Viewing the Results.

1. Once the solver has finished its job, the **Animate** button will be active. Click on ***Animate***.

   The ▶ can be used to start the animation, and the ⏸ icon can be used to stop/pause the animation.

   One would also like to inspect the displacement profile of the membrane and the ball. For this, we will plot the Z position of the center of mass of the ball.

2. Return to the MotionView window and click on the ***Plot*** button in the **Run** panel.

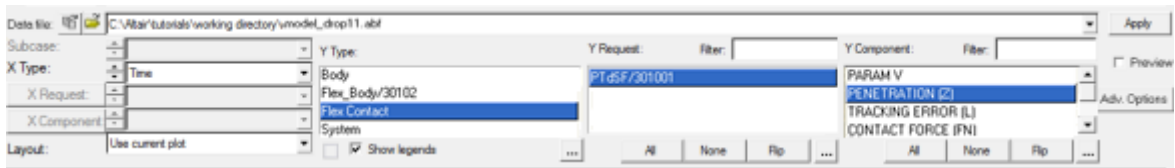   A HyperGraph window is added and the result file is loaded.

3. Click *Apply*.

    The profile for the Z-displacement of the ball should look like the one shown below:
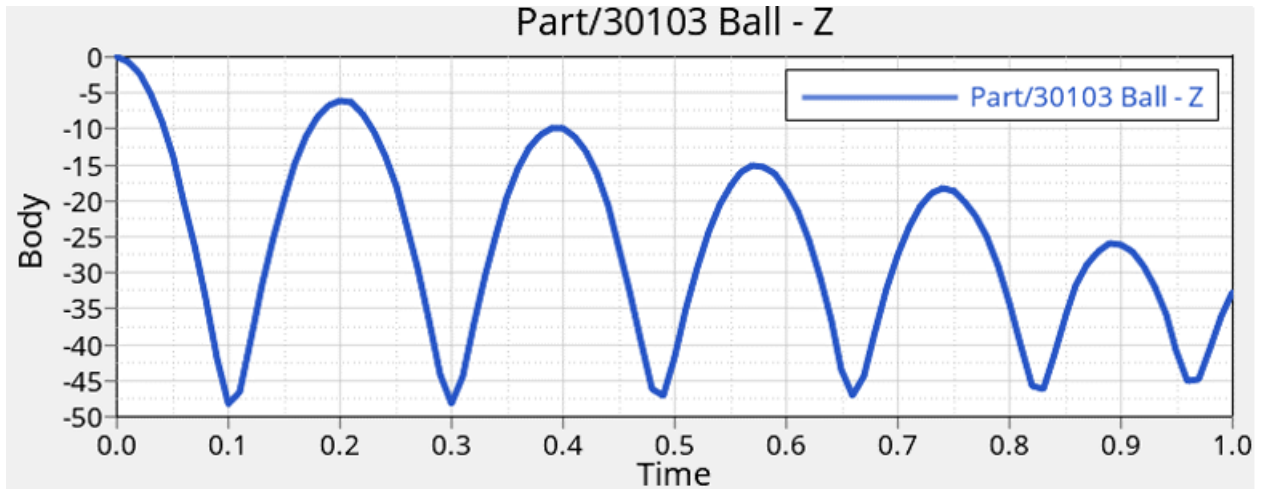
We can also plot the penetration distance for this flexible contact.

4. Add a page to the session .

5. Make selections for the plot as shown below:

6. Click *Apply*.

7. The penetration profile as a function of time looks like the one shown below: