HyperWorks

Altair MotionView 2019 Tutorials

MV-1028: Modeling Point-to-Deformable-Surface (PTdSF) Higher-Pair Constraint

altairhyperworks.com

MV-1028: Modeling Point-to-Deformable-Surface (PTdSF) Higher-Pair Constraint

In this tutorial, you will learn how to:

- Model a PTdSF (point-to-deformable-surface) joint

A PTdSF (point-to-deformable-surface) joint is a higher pair constraint. 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 PTdSF constraint.

Exercise

Copy the files `membrane.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. Click the **Project Browser** tab, 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** = `0.0`.

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`.

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

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

4. From the **Properties** tab, check the *Deformable* box.

5. Click on the *Graphic file* browser icon 📂 and select `membrane.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* to define a second 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	4000.00	4000.00	4000.00	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.

13. For the **Ball** body, from the **Initial Conditions** tab - check the *Vx* box under *Translational velocity* and enter a value of `1` into the text box.
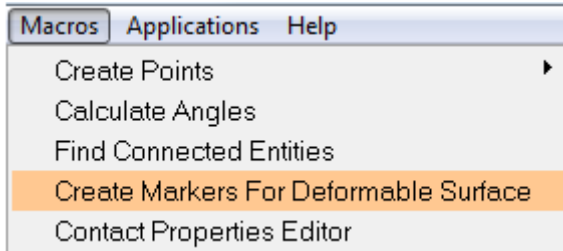
    This sets a value of 1 for the translational velocity of the ball in the X direction.

14. Repeat the same for *Vy*.

△ Altair

Step 3: Creating Markers and a Deformable Surface.
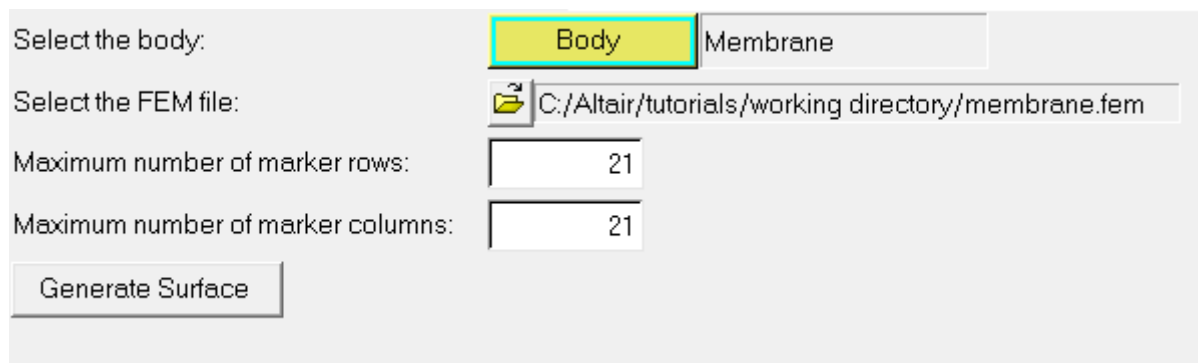
Now, we will define some markers required for the membrane.

1. From the **Macros** menu, select *Create Markers For Deformable Surface*.

    ```
    Macros   Applications   Help
      Create Points                        ▶
      Calculate Angles
      Find Connected Entities
      Create Markers For Deformable Surface
      Contact Properties Editor
    ```

    The **Create Markers For Deformable Surface** utility is displayed at the bottom of the screen.

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

Select the body:	Body	Membrane
Select the FEM file:		C:/Altair/tutorials/working directory/membrane.fem
Maximum number of marker rows:		21
Maximum number of marker columns:		21
Generate Surface		

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

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

5. Click *Generate Surface*.

    The Markers and Deformable Surface are created.

Step 4: Creating Joints.

Here, we will define all the necessary joints except the PTdSF joint, which will be defined as an advanced joint later. 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`.

△ Altair

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 Advanced Joints.

Now we will define the advanced PTdSF joint.

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

    The **Add AdvJoint** dialog is displayed.

2. For **Label**, enter `AdvancedJoint 0`.

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

4. From the **Connectivity** tab, select:

    – *PointToDeformableSurface Joint*

    – *Ball* for **Body**

    – *BallCM* for **Point**

    – *DeformableSurface 1* for **DeformableSurface**.

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. Under the **Connectivity** tab, double-click on *Body*.

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

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

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

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

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

8. Under the **Properties** tab, enter `1.0` as the radius of the **Ball**.

9. Under 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. For the membrane which has already been defined, click on the *Nodes* button.

    The **Nodes** dialog is displayed.

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

4. Close the the **Nodes** dialog.

    At the end of these steps your model should look like the one shown in the figure below:
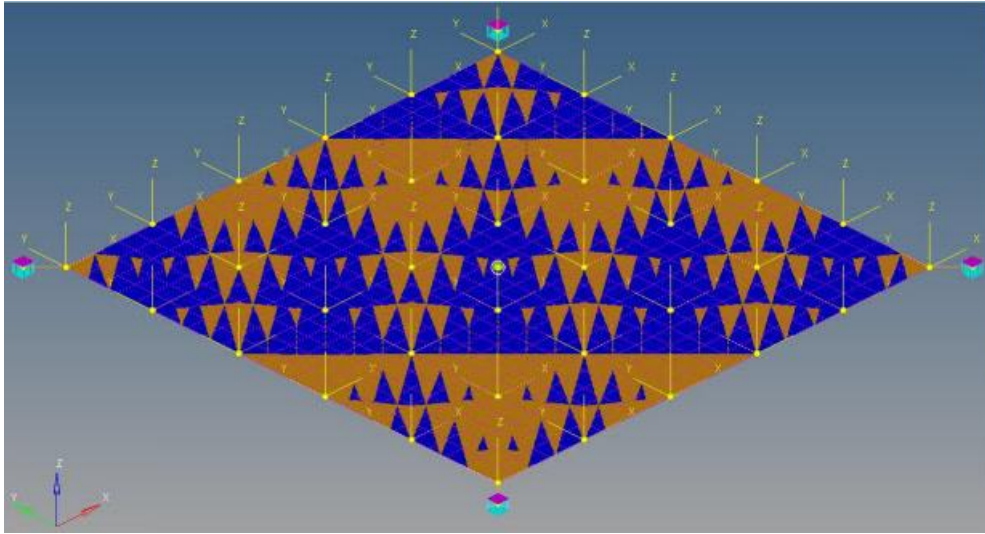
One final comment before running the model:

This type of constraint does not ensure that the contact point will stay within the range of data specified for the surface. Additional forces at the end need to be defined by the user to satisfy this requirement. If the contact point goes out of range of the data specified for this curve, the solver encounters an error (unless additional forces are defined to satisfy this). In that case, one has to change the initial velocities for the ball, or increase the range of data specified for the curve, or run the simulation for a shorter interval of time.

Step 8: Running the Model.

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

1. Click the *Run* icon 🔵 on the **Model-Main** toolbar.

    The **Run** panel is displayed.

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

    – **End time** = 2.000

    – **Print interval** = 0.0100

△ Altair
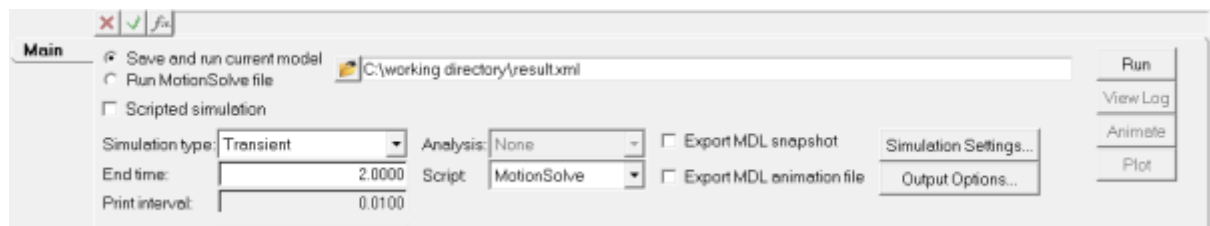
3. Click on the ***Simulation Settings*** button.

   The **Simulation Settings** dialog is displayed.

4. Click the ***Transient*** tab and as specify the **Maximum step size** as `0.001` (as the solution is not converged for the default step size of 0.01):
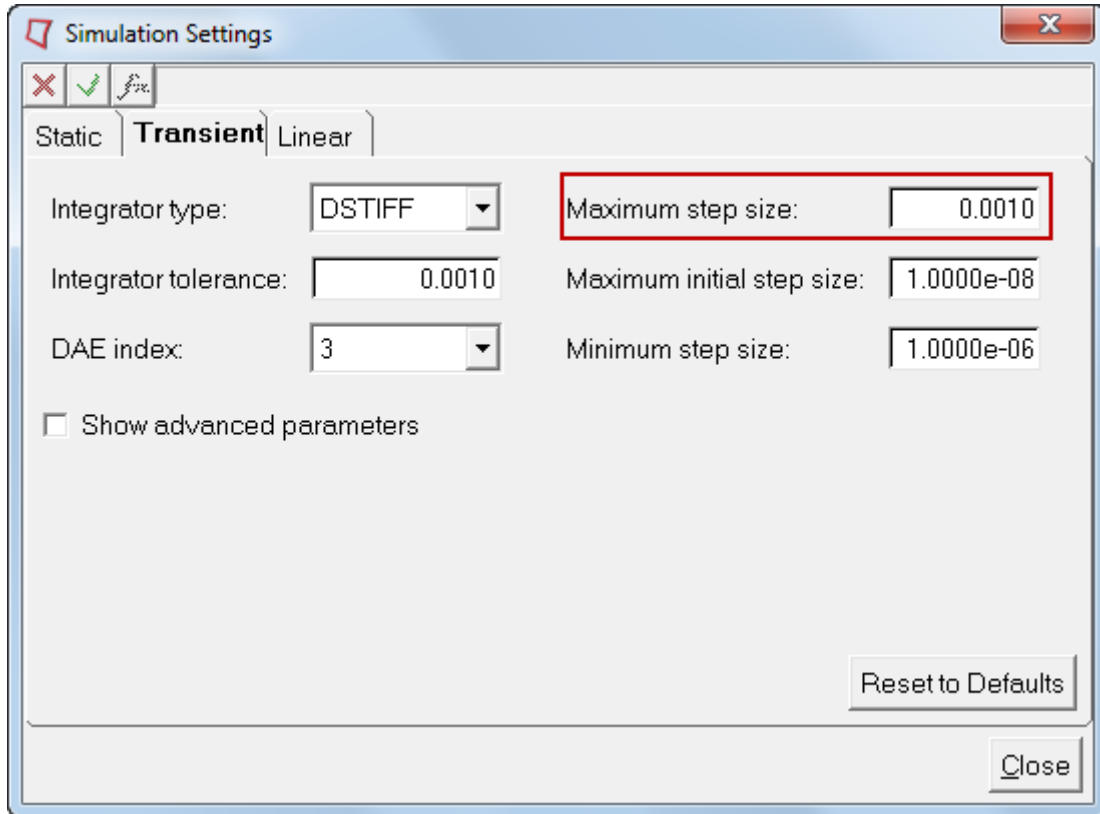
5. Click ***Close*** to close the dialog.

6. Verify that the ***Save and run current model*** radio button is selected.

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

8. Click ***Save***.

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

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

   The solver will get invoked here.

Step 9: Viewing the Results.

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

   The 🔵 icon 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 beam and the ball. For this, we will plot the Z position of the center of mass of the ball.

2. Click on the ***Add Page*** icon and add a new page.

3. Use the **Select application** drop-down menu to change the application from ***MotionView*** 🔵 to ***HyperGraph 2D*** 📊.

4. Click the ***Build Plots*** 📖 icon on the ***Curves*** toolbar.

5. Click on the browser icon 📂 and load the `result.abf` file from the `<working directory>`.

6. Make selections for the plot as shown below:
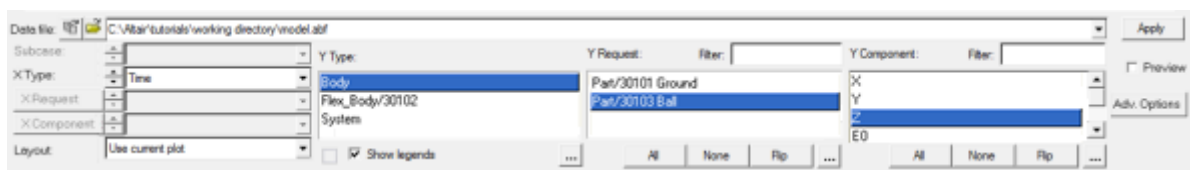
   We are plotting the Z position of the center of mass of the ball.

7. Click ***Apply***.

   The profile for the Z-displacement of the ball should look like the one shown below: