

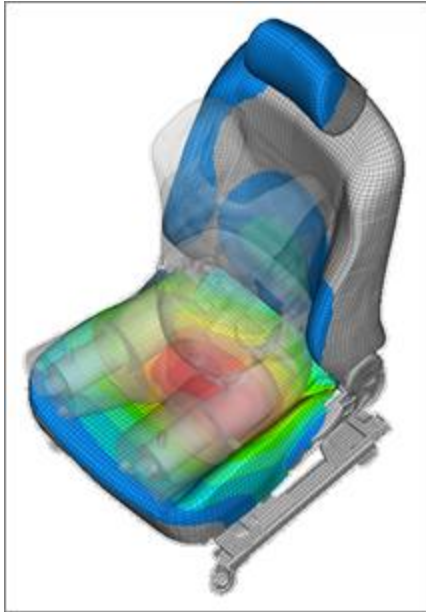


Altair

HyperWorks

HM-4650: Seat Deformer with LS-DYNA Pre-Simulation

In this tutorial, you will learn how to setup a LS-DYNA model to simulate the seat deformation under the dummy.



Model Files

The driver seat and environment used in this tutorial are based on the LS-DYNA Toyota Yaris model, provided on the National Crash Analysis Center (NCAC) website.

The LS-DYNA dummy model used in this tutorial is a release version of the THOR-Mod Kit/Metric crash test dummy with SD3 shoulder posted on the NHTSA website.

This tutorial uses the `seat_deformer.hm` file, which can be found in `<hm.zip>/interfaces/lsdyna/`. Copy the file(s) from this directory to your working directory.

Exercise

Step 1: Open HyperMesh with the LS-DYNA User Profile

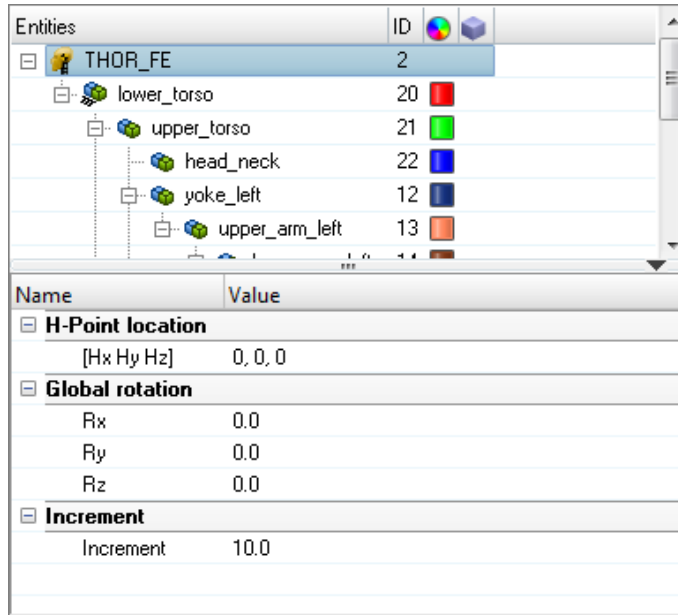
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna, Keyword971_R8.0**.


Step 2: Load the Model File

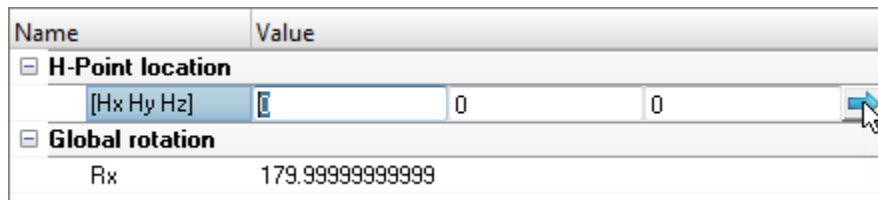
1. From the menu bar, click **File > Open > Model**.
2. In the **Open Model** dialog, navigate to your working directory and open the file `seat_deformer.hm`.

Step 3: Position the Dummy to its H-Point Location

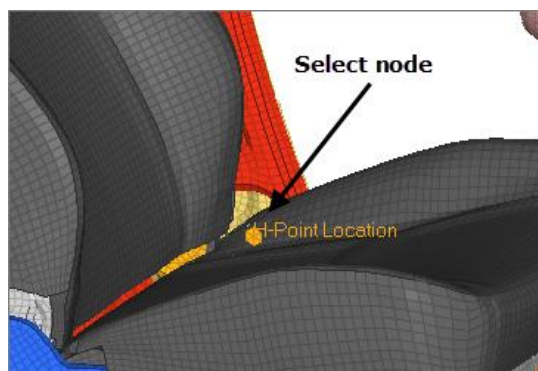
1. From the menu bar, click **Tools > Dummy** to open the **Dummy** browser.
2. In the **Dummy** browser, click on the **THOR_FE** dummy entity to activate the global positioning parameters in the **Entity Editor**.



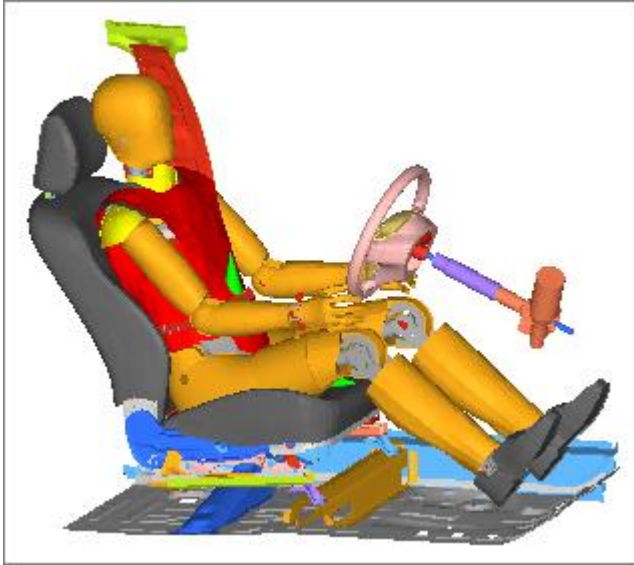
3. In the **Entity Editor**, for the **Global rotation** attribute **Rx**, enter 180. The whole dummy rotates.
4. Under **H-Point location**, click on the coordinates area and then click .



5. In the graphics area, select the node tagged as **H-Point Location** on the seat.



6. In the panel area, click **proceed**. The dummy automatically moves to the selected H-Point location.



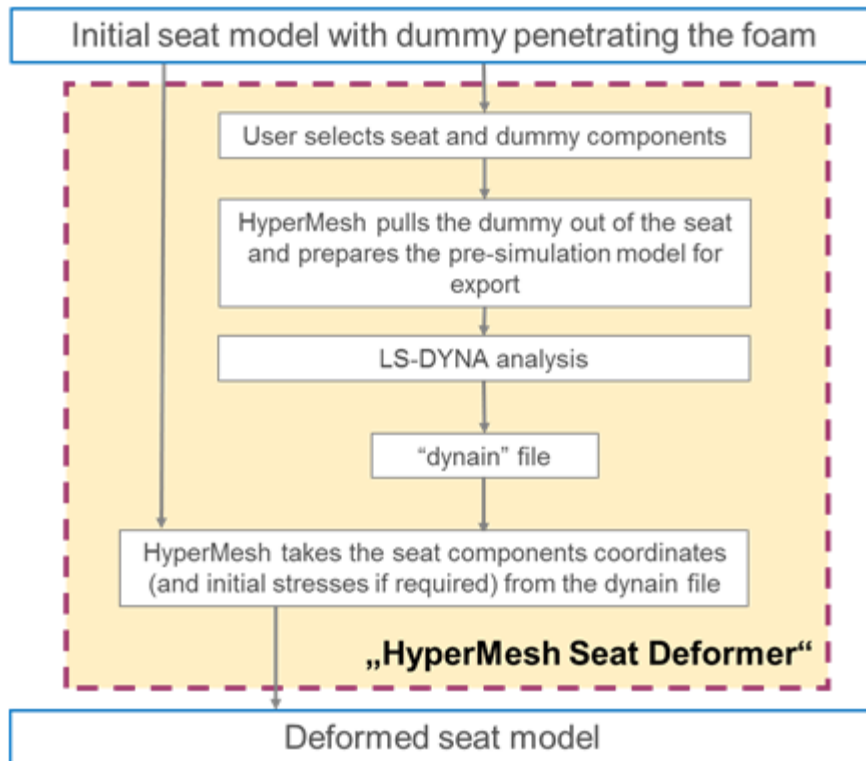
Step 4: Generation of the Seat Deformer Model for the LS-DYNA Simulation

After positioning the dummy to its H-Point location on the seat, intersections between the dummy and seat foam parts occur.

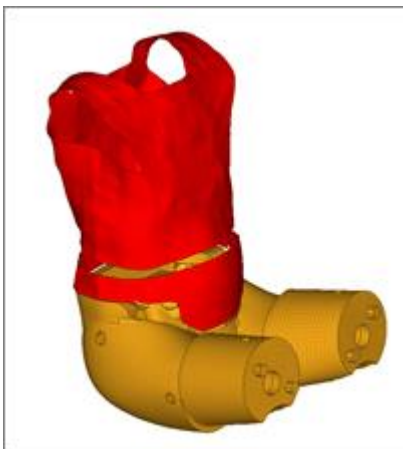


Cutting plane showing intersection between the dummy and seat components.

The **Seat Deformer** tool enables you to export an input deck, for the appropriate solver, in order to simulate the deformation of the seat under the dummy. The simulation result files are then imported to update the initial FE model, and thus remove the intersections and penetrations between the dummy and the seat.



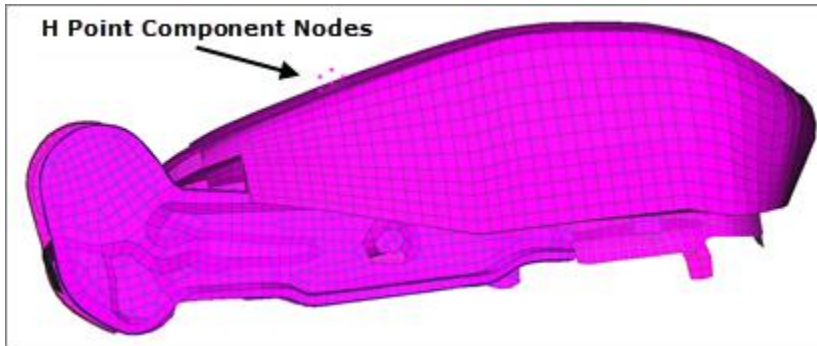
1. Open the **Seat Deformer** tool, from the menu bar, by clicking **Tools > Pre-Simulation(Seat Deformer)**.
2. In the **PreSimulation** dialog, under **PreSimulation Set-Up**, set **Unit System** to **mm,ms,kg**.
3. Select the dummy components penetrating the seat foam.
 - a. In **Dummy** field, click **0 Components > Components**.
 - b. In the **Select Components** dialog, select the components shown in the image below and click **OK**.



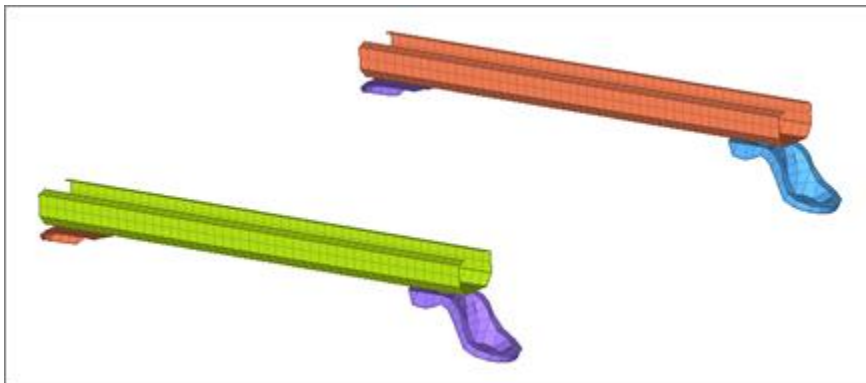
4. Select the seat mechanism.
 - a. In **Seat** field, switch the selector to **Mechanisms**.

- b. Click **0 Mechanisms > Mechanisms**.
- c. In the **Select Mechanisms** dialog, select **Driver_Seat** and click **OK**.

Note: By selecting a mechanism, only the components defined in the bodies will be taken into account. The nodes will not be taken into account. In this example, the Seat_Cushion body is composed of a set of components and a set of nodes. The nodes will not be taken into account in the process.



5. Select the fixed nodes of the seat.
 - a. In the **Fixed Nodes Of Seat** field, click **0 Nodes > Nodes**
 - b. In the panel area, click **nodes >> by collector**.
 - c. Select following components.



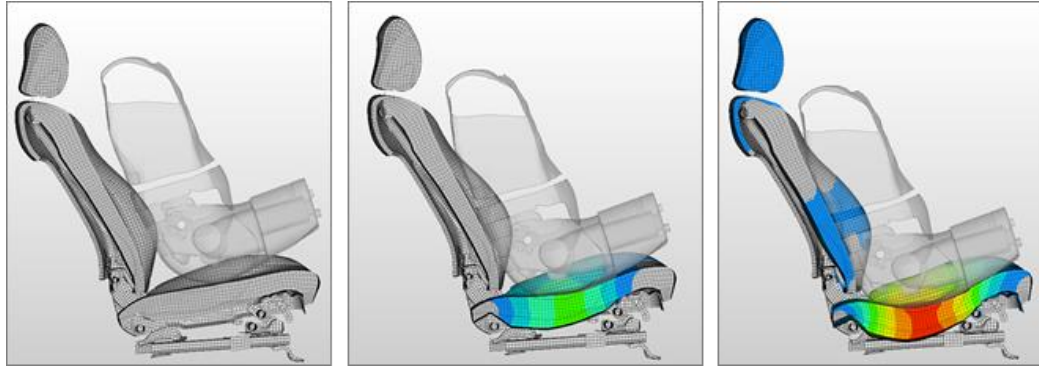
- d. Click **proceed** when finished.
6. In the **Dummy Displacement Direction** field, enter $-0.3, 0, -0.7$.
7. In the **Export File** field, specify the file path and name of your working directory.
8. In the **Dummy Velocity** field, enter 1.0 mm/ms.
9. In the **Dummy Displacement Step** field, enter 20.0 mm.
10. In the **Imposed Contact Thickness** field, enter 1.0 mm.
11. Click **Export** to start the export process of the LS-DYNA seat deformer pre-simulation input deck.

The LS-DYNA model is exported to the given location and ready to be solved by the solver.

Step 5: Review Seat Deformer Simulation Results and Update the Initial Model

When the seat deformer model has been solved by the solver, you should find the d3plot files to visualize the animations in HyperView.

The displacement fields on the seat should look similar to the images below.



Check also that LS-DYNA has written out the dynain file, which contains the new coordinates of the seat nodes at the end of the simulation, as well as the *INITIAL_STRESS cards for the different type of elements present in the seat model (beam/shell/solid).

To update the initial model with the simulation results, proceed as follow:

1. If you have closed the **Pre-Simulation(Seat Deformer)** tool, re-open it.
2. Verify that the Import *INITIAL_STRESS_SOLID checkbox is enabled. By default, this checkbox is enabled in order to update the solid elements with the initial stress values coming from the simulation results.
3. Click **Import dynain File**, then select the resulting dynain file of the seat deformer simulation.

Default updates the node coordinates of the seat and the solid elements with their corresponding initial stress card.

The resulting updated model will look similar to the image below.



Initial model



Updated model