Altair

HyperWorks

HM-4910: Setting up a MADYMO Occupant Safety Analysis in HyperMesh

In this tutorial, you will:

- Load the MADYMO user profile
- Import XML files

The MADYMO HyperMesh interface supports both the import and export of MADYMO files. A MADYMO XML file can be read using the *Import* tab. The model will be displayed. The model can be altered and the finished model can be written using the *Export* tab.

Model Files

This tutorial uses the following files, which can be found in `<hm.zip>/interfaces/madymo/`. Copy the file(s) from this directory to your working directory.
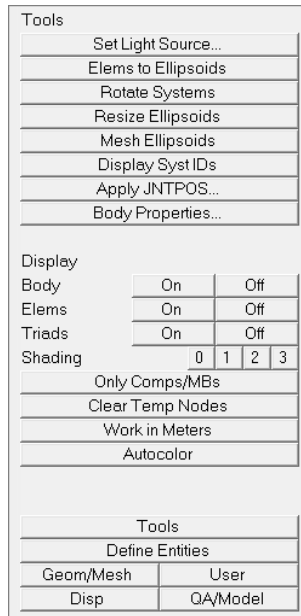
- `truck_model.xml`
- `steering_column.xml`
- `seatbelt_system.xml`
- `dummy_pulse.dat`

Exercise 1: Position the dummy

Step 1: Load the MADYMO user profile

The MADYMO user profile configures the user interface to be MADYMO specific. The user profile also loads the XML import reader and MADYMO template file. The template file defines MADYMO specific entity types and attributes versus generic entities.

1. Upon opening, HyperMesh prompts you to select a user profile. Select the *MADYMO* profile.

2. Once the **MADYMO** profile is selected, the **template** drop down menu becomes activated. Select the *MADYMO 70* template.

3. Click *OK*.

4. If the **Utility Menu** is not already displayed, click *View* > *Browsers* > *HyperMesh* > *Utility* to display it.

Step 2: Import the file

The `truck_model.xml` file was created by converting an **NCAC LS-DYNA truck** model to **MADYMO** within the HyperMesh interface.

1. Click *File* > *Import* > *Solver Deck*.

2. In the **File:** field, browse to the file `truck_model.xml` and click *Open*.

3. Click *Import Options* to expand the panel and display the advanced importing options.

4. Click on the *Display Import Errors* checkbox to deactivate this function.

5. Leave all other default settings as they are and click *Import* to import the model.

6. Remain in the **Import** tab.

Step 3: Import the steering column file

The `steering_column.xml` file was created from scratch within the HyperMesh interface.

1. In the **File:** field, browse to select the `steering_column.xml` file and click *Open*.

2. Leave all other default settings as they are and click *Import* to import the model.

Step 4: Position the steering column

The **Set Joints** panel contains functionality for defining joint rotations and joint orientations for all joints plus global positioning for the highest joint in any system of bodies.

1. From the **Analysis** page, click *safety* and then select the *set joints* subpanel.

2. Select the **H-point** subpanel. The **H-point** subpanel is used for defining global positioning for the highest joint in the system of bodies.

3. Make sure the yellow entity selection is set to **multibody**. To change, click on the triangle switch button next to the yellow entity selection box and select ***multibodies*** from the pop-up list. **Comp** is used to position finite element and facet dummy models. **Multibody** is used for positioning ellipsoidal rigid body models.

4. Click on any ellipsoid in the steering column.

5. In the **Position** fields, enter the following:

   - **x =** `3.30`

   - **y =** `0.41`

   - **z =** `0.98`

6. Click the green ***position*** button to set the position of the steering column.

7. Enter `180.00` in the increment field.

8. Click the triangle switch next to ***N1, N2, N3*** and select ***z-axis*** from the pop-up window.

9. Click the green ***rotate+*** button to rotate the **steering** column.

10. Click ***return*** to exit the panel.

Step 5: Import `d_hyb350el_usr.xml`

The `d_hyb350el_usr.xml` file is taken directly from your MADYMO installation. These files can be found in the path `...\share\dbs\dummies\3d` folder in typical MADYMO installation.

1. Click *File* > *Import* > *Solver Deck*.

2. In the **File:** field, browse to select the `d_hyb350el_usr.xml` file and click **Open**.

3. Click ***Import*** to import the model.

Step 6: Position the dummy H-point

1. From the **Analysis** page, click ***safety*** and then select the ***dummy*** subpanel.

2. Select the ***H-point*** subpanel.

3. Click on any **ellipsoid** in the dummy.

4. In the **Position** fields, enter the following:

   - **x =** `2.16`

   - **y =** `0.41`

   - **z =** `0.88`

5. Click the green *position* button to set the position of the dummy.

6. Enter 18 in the **increment** field.

7. Click the triangle switch under **rotate:** and select *y-axis* from the pop-up window.

8. Click the green *rotate-* button to rotate the dummy back against the seat.

9. Remain in the **Dummy Positioning** panel for the next exercise.

Steps 7-10: Position the dummy limbs

You can quickly find the dummy limbs used in the steps below by using the **Model Browser**. Right-click on any entity and select Isolate to display only the part being manipulated.

Use the **Dummy Positioning** panel for the next series of steps.

Step 7: Rotate the femur ellipsoids

1. Select the *incremental* subpanel.

2. Enter 10 in the **increment** field.

3. Click on a femur ellipsoid *(FemurR_bod)*. HyperMesh automatically searches up to the next joint in the hierarchy.

4. Rotate the hip child joint coordinate system about the x, y, and z axes of the hip parent joint coordinate system by using y rot >.

5. Click the *reset* button ◄ next to the **multibody selection** option.

6. Repeat the same for other femur ellipsoid **(FemurL_bod)**.

Step 8: Rotate the upper tibia

1. Select the right upper tibia ellipsoid **(tibiaUpR_bod)**.

2. In the **y rot** row, set the current field to –60 and click *enter*.

3. Repeat steps one and two for left upper tibia ellipsoid (**tibiaUpL_bod**).

Step 9: Rotate the upper arm

1. Select left upper arm ellipsoid **(ArmUpL_bod)**.

2. In the **x rot** row, set the current field to –30 and click *enter*.

3. Repeat steps one and two for the right upper arm ellipsoid **(ArmUpR_bod)**.

Step 10: Rotate the lower arm

1. Select the left lower arm ellipsoid (**ArmLowL_bod**).

2. In the **x rot** row, set the **current** field to –5 and click *enter*.

3. In the **y rot** row, set the **current** field to `-70` and click ***enter***.

4. Select the **right lower arm ellipsoid** (**ArmLowR_bod**).

5. In the **x rot** row, set the **current** field to `5` and click ***enter***.

6. In the **y rot** row, set the **current** field to `-70` and click ***enter***.

7. Remain in the **Dummy Positioning** panel for the next exercise.

Step 11: Set the steering wheel tilt

While still in the **Dummy** panel, complete the following steps:

1. Activate the **incremental** radio button.

2. Enter `10` in the **increment** field.

3. Select an ellipsoid in the steering wheel. HyperMesh automatically searches up to the next joint in the hierarchy.

4. Click the ***rotate:*** toggle to change **joint dof** to **joint orientation***.*

   joint dof rotates the child joint coordinate system locally about its axes and **joint orientation** rotates the parent joint coordinate system. Rotating the child joint coordinate system instead of the parent joint coordinate system to set the tilt will initiate an undesired initial torque. Rotation the parent joint coordinate system will temporarily "unlock" the joint.

5. Use the **<** button next to **x rot** to set the tilt.

6. Set the **rotate:** toggle back to **joint dof**.

7. Click ***return*** to exit the panel.

Step 12: Define the MADYMO/SYSTEM/FE MODEL hierarchy

Since three running MADYMO models were imported, extra INERTIAL_SPACE and MADYMO_MODEL blocks were defined.

1. Click on the **Model Browser**.
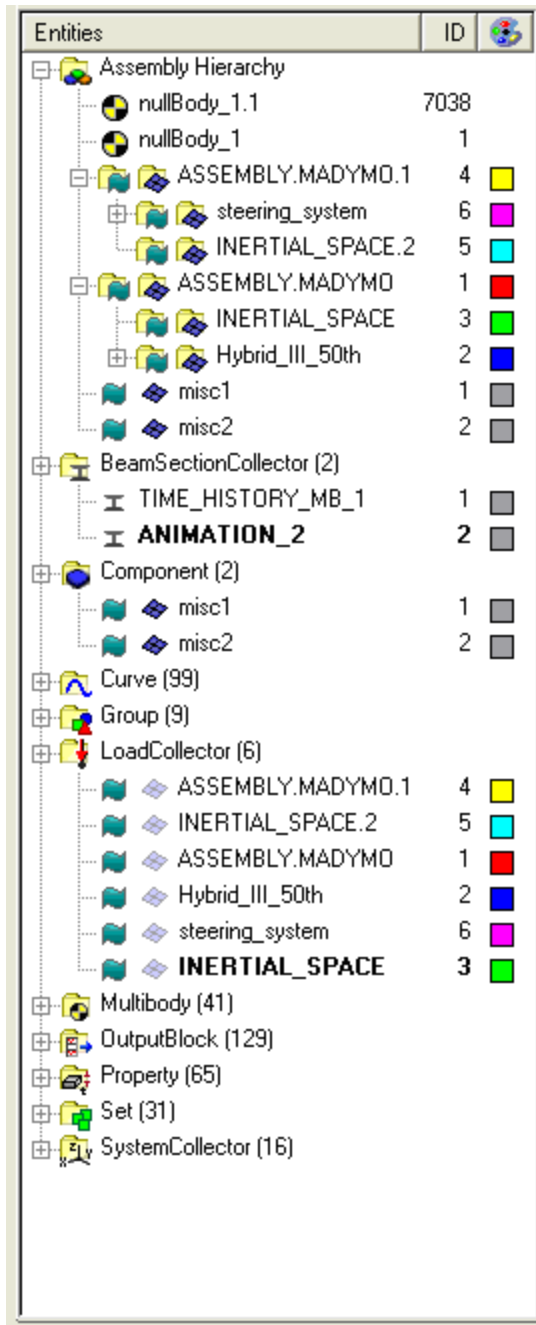
2. Drag and drop **truck_system** and **steering_system** from **ASSEMBLY.MADYMO** and **ASSEMBLY.MADYMO.1** respectively into the **ASSEMBLY.MADYMO.2** folder.

3. Right-click and select delete on **ASSEMBLY.MADYMO** and **ASSEMBLY.MADYMO.1**.

Steps 13 - 14: Applying a pulse function

A pulse function is applied by reading vector data from an external file source and applying the vector to a system of bodies in the card editor.

Step 13: Read the pulse data from an external file

HyperMesh can read vector data from the same file formats supported by HyperGraph. HyperMesh recognizes file types based on the file's contents and automatically selects the correct reader to import the data.

1. Click *XYPlots* > *Create* > *Plots*.

2. In the **plot=** field, enter *acceleration_data*.

3. Click *create plot* to create an empty plot window.

4. Click *return* to leave the **Plots** panel.

5. Click *XYPlots > Edit > Plots.*

6. Click on *create* to open the **create** subpanel.

7. In the **file** field, click *load.* Browse to the `/hm/interfaces/madymo` directory and select the file `dummy_pulse.dat`.

8. Activate the **y=** radio button.

9. Click the **+** button next to **comp=** to set the y component to `Column 2`.

10. Click *create*.

11. Remain in the **Edit Curves** panel for the next exercises.

Step 14: Apply the pulse vector to the dummy

The pulse vector is applied by card editing the HyperMesh assembly which represents the dummy system of bodies.

1. Right click *Hybrid_III_50th* in the **Model Browser** and click *Card Edit*.

2. Activate the **LOAD.SYSTEM_ACC** checkbox.

3. Click once on the yellow *AX_FUNC* curve selection box.

4. Click on the pulse curve (**curve1**) in the plot window.

5. Click *return* to save the changes and leave the card image.

Steps 15 - 16: Applying gravity

A gravity vector is applied by defining the vector in HyperMesh then applying the vector to a system of bodies in the card editor.

Step 15: Create vector data in HyperMesh

1. From the **XY Plots** menu, click on *Curve Editor...*.

2. Click on *New...* at the bottom of the window.

3. In the **Name=** field, enter `gravity`.

4. Click *proceed*.

5. In the **Curve Editor** window, select *gravity* on the top left corner.

6. In the table, in the first row, enter:

    -   In the **X** column, enter `0`

    -   In the **Y** column, enter `-9.81`

7. In the second row of the table, enter

    -   In the **X** column, enter 2

    -   In the **Y** column, enter `-9.81`

8. Select a color, and activate the **Display** checkbox.

9. Set the **Symbol** field to **Triangle***.*

10. Set the **Line style** field to **Solid***,* and activate the **Thick line** checkbox.

11. Click *Update.*

12. Click *Close* to close the **Curve Editor**.
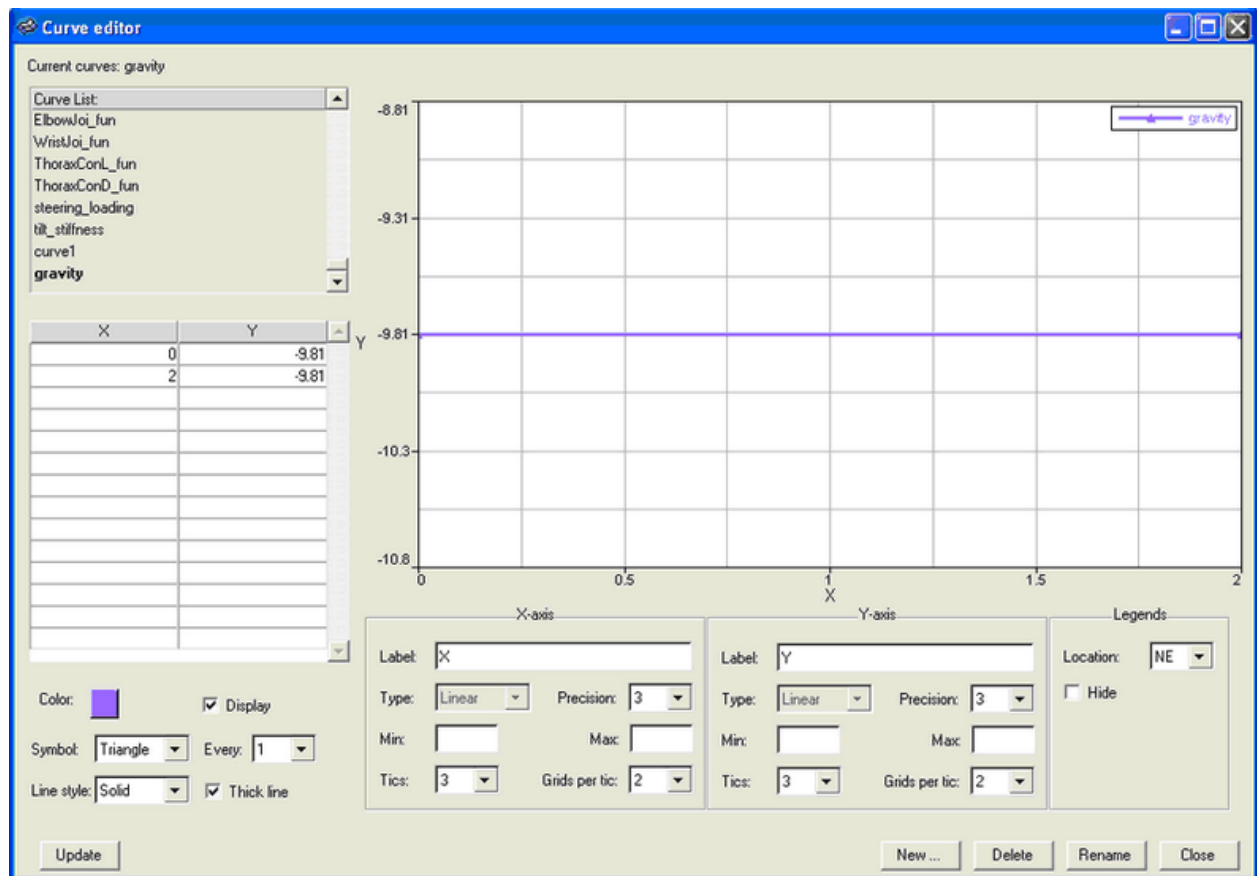
Step 16: Apply the gravity vector to the dummy

The gravity vector is applied by card editing the HyperMesh assembly which represents the dummy system of bodies.

1. Right click the *Hybrid_III_50th* assembly in the **Model Browser** and click *Card E*dit.

2. Click once on the *yellow AZ_FUNC curve selection* box.

3. Click on the *gravity curve (gravity)* in the plot window.

4. Click *return* to save the changes and leave the card image.

Steps 17 - 19: Defining contacts

Contacts are created in the *contacts* panel. A MADYMO GROUP consists of a single entity LIST or multiple LIST cards. LIST and GROUP cards are both defined in the HyperMesh **Entity Sets** panel.

Step 17: Create a contact entity

1. Click *BCs* > *Create* > *Interfaces*.

2. In the **name=** field, enter `seat_to_dummy`.

3. Click *type=* and select **MB_MB** from the pop-up list.

4. Click *create/edit* to create the entity and bring up the **card editor**.

Step 18: Define contact attributes

1. In the **FRIC_COEF** field, enter `0.1`.

2. Click on the box under **CONTACT_TYPE** and select **MASTER** from the pop-up list.

3. Click *return* to close to save the changes and close the **card editor**.

Step 19: Add master and slave surfaces

1. Click *add* to open the **add** subpanel.

2. Next to the **master:** field, click the *yellow sets entity selection* box twice to bring up a list of sets.

3. Activate the check box next to **seat_gmb** to make that the master surface.

4. Click *select*.

5. Click *update* to the right of **master:** to update the master surface definition.

6. Next to the **slave:** field, click the *yellow sets entity selection* box twice to bring up a list of sets.

7. Activate the checkbox next to *thorax_gmb and pelvis_gmb* to use these as the slave surface.

8. Click *select*.

9. Click *update* to the right of **slave:** to update the slave surface definition.

10. Repeat for other contacts.

Steps 20 - 23: Adding Seatbelts

A seatbelt system can be created by scratch, or by importing and modifying and existing system. This exercise focuses on creating a seatbelt system in the seatbelt routing panel. When creating seatbelts from scratch, create a component of type BELT for each BELT_TYING or RETRACTOR point then connect the belt points in the *seatbelt* routing panel.

Step 20: Import belt_system.xml

The `seatbelt_system.xml` contains a running model of a **single belt** system.

1. Click *File* > *Import* > *Solver Deck*.
2. In the **File:** field, browse to select the `seatbelt_system.xml` file and click *Open*.
3. Click *Import* to import the model.

Step 21: Delete existing belt elements

The `seatbelt_system.xml` contains a running model of a single belt system.

1. Press the *F2* key to enter the **Delete** panel.
2. Set the yellow entity selector to **elements**.
3. Click *elements* and select *by collector* from the pop-up list.
4. Activate the check boxes next to **fe_shoulder_belts** and **fe_lap_belts** and click *select*.
5. Right click on the last lap belt LINE2 element attached to the constraint in the floor (**left end of pelvis**) to deselect it.
6. Click *delete entity* to delete the selected entities.
7. Click *return* to exit the panel.

Step 22: Route the shoulder belt

1. Enter the **Seatbelt** panel from the **Safety** panel.
2. Click *G* to open the **Global** panel.
3. Click *component=* and click *fe_shoulder_belts* in the component list.
4. Click *return* to leave the **Global** panel.
5. Select the node at the end of the BELT_SEGMENT element near the left shoulder.
6. Select the node at the end of the higher BELT_SEGMENT element near the buckle.
7. Click the *ellipsoids* entity selection button to make it active.
8. Click the ellipsoids in the thorax. Alternatively click the *ellipsoids* button again to display the ellipsoid list, select ellipsoids *by collector*, and click on all the bodies in the thorax.
9. Click *orient* to display the belt routing line segments.

10. Click on one of the red, end segments to activate it.

11. While holding the left mouse button, drag the mouse up and down to adjust the location of the belt segment.

12. Repeat the previous two steps for the other end segment and then the middle segment.

13. Click *mesh* to create the mesh.

14. Set **using size=** to `0.03`.

15. Click *return*.

16. Remain in the **Seatbelt** panel for the next exercise

Step 23: Route the lap belt

1. Click *G* to open the **Global** panel.

2. Click *component=* and click *fe_lap_belts* in the component list.

3. Click *return* to close the **Global** panel.

4. In the **Seatbelt** panel, select the node at the end of the LINE2 element near the floor constraint.

5. Select the node at the end of the lower BELT_SEGMENT element near the buckle.

6. Select the ellipsoids in the pelvis and abdomen.

7. Click *orient* to display the belt routing line segments.

8. Click on one of the red end segments to activate it.

9. While holding the left mouse button, drag the mouse up and down to adjust the location of the belt segment.

10. Repeat the previous two steps for the other end segment and then the middle segment.

11. Click *mesh* to create the mesh.

12. Set **using size=** to `0.03`.

13. Click *return* to close the **Seatbelt** panel.

Steps 24 - 27: Equivalence duplicate nodes

Many times it is preferred to use a duplicate node as the to or from node to define BELT_TYING. However, some times this feature is not preferred. Because inserting a coincident node is more difficult than equivalencing nodes, the *seatbelt* panel always uses a duplicate node. This training exercise does not use the duplicate node option in BELT_TYING so the duplicated nodes must be equivalenced.

Step 24: Replace nodes

1. Click *O* to open the **Options** panel.

2. Click on the *graphics* subpanel.

3. Activate the *coincident picking* checkbox.

4. Click *return* to leave the **Options** panel.

5. Press *F3* to enter the **Replace** panel.

6. Click a node selected as a to or from node in the **Seatbelt** panel.

7. In the **selection** circle, select the node with the lower ID.

8. Click on the same node location and select the other node from the coincident selection circle.

9. Repeat for the other three to and from nodes.

10. When finished, click *return* to close the panel.

Step 25: Create a group multi-body for contacts

1. Click *Tolls* > *Create* > *Sets*.

2. Set the yellow entity selector to *nodes*.

3. In the **name=** field, enter `seatbelt_node_list`.

4. Click on the *nodes* selection box and *by collector* from the pop-up list.

5. Activate the **fe_shoulder_belts** and **fe_lap_belts** check boxes and click *select*.

6. Click *create* to create the entity set.

7. Set the yellow entity selector to *sets*.

8. In the **name=** field, enter `seatbelt_group`.

9. Click on the *sets* selection box.

10. Activate the **seatbelt_node_list** check box and click *select*.

11. Click *create* to create the entity set.

12. Click *return* to leave the **Entity Sets** panel.

Step 26: Define the contact

Applying MB_FE contacts is very similar to applying MB_MB contacts.

1. Click *BCs* > *Create* > *Interfaces*.

2. In the **name=** field, enter `dummy_to_belts`.

3. Click *type=* and select *MB_FE* from the pop-up list.

4. Click *create/edit* to create the contact definition and bring it up in the card editor.

5. Set **CONTACT_FORCE** to **KINEMATIC**.

6. Under **FRIC_COEF**., enter `0.1`.

7. Click *return* to save the changes and close the card editor.

8. Click the **add** subpanel.

9. Click the master *sets* selection box.

10. Activate the **Shoulders_gmb**, **Thorax_gmb**, **Abdomen_gmb**, and **Pelvis_gmb** check boxes and click *select*.

11. Click the master *update* button.

12. Click the slave *sets* selection box.

13. Activate the **seatbelt_group** check box and click *select*.

14. Click the slave *update* button.

15. Click *return* to close the **Contacts** panel.

Step 27: Update the MADYMO/SYSTEM/FE MODEL hierarchy

Since three running MADYMO models were imported, extra INERTIAL_SPACE and MADYMO_MODEL blocks were defined.

1. Click on the **Model Browser**.

2. Click and drag the **Vehicle** folder from its **ASSEMBLY.MADYMO** folder to the top **ASSEMBLY.MADYMO.2** folder.

3. Click and drag **slipring_seg** and **buckle_seg** from their **ASSEMBLY.MADYMO** folder to the top **ASSEMBLY.MADYMO.2** folder.

4. Right-click and select *delete* on **ASSEMBLY_MADYMO**.

Step 28: Export and run for finite time

1. Right click *ASSEMBLY.MADYMO.2* in the **Model Browser** and click *Card Edit*.

2. In the **card image**, enter `0.05` in the **TIME_END** field.

3. Click *return* to save the changes and close the card image.

4. Click *File* > *Export* > *Solver Deck*.

5. In the **file:** field, enter an `XML` file name.

6. Click *Export*.