Altair HyperMesh 2019 Tutorials

HM-1010: Opening and Saving Files

altairhyperworks.com

HM-1010: Opening and Saving Files

In this tutorial, you will learn how to:

- Open a HyperMesh file
- Import a file into a current HyperMesh session
- Save the HyperMesh session as a HyperMesh model file
- Export all of the geometry to an IGES file
- Export all of the mesh data to an OptiStruct input file
- Delete all of the data from the current HyperMesh session
- Import an IGES file
- Import an OptiStruct file to the current HyperMesh session

Model Files

This exercise uses the following model files: `bumper_cen_mid1.hm`, `bumper_mid.hm`, `bumper_end.igs`, and `bumper_end_rgd.fem`. Each model file contains different sections of the bumper, that when combined make up the whole bumper model. Copy the model file(s) from the `hm.zip` file to your working directory.

Exercise

Step 1: Open the HyperMesh model file, `bumper_cen_mid1.hm`.

1. Start HyperMesh Desktop.
2. To open a HyperMesh model file, click *File* > *Open* > **Model** from the menu bar, or click on the **Standard** toolbar.
2. In the **Open File** dialog, open the `bumper_cen_mid1.hm` model file. HyperMesh loads a model containing mesh and geometry data.
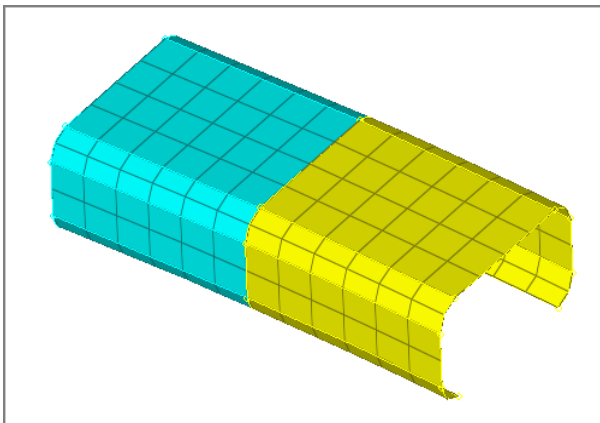
Step 2: Import the HyperMesh model file, `bumper_mid.hm`, into the current HyperMesh session.

1. To open the **Import** tab, click *File* > *Import* > *Model* from the menu bar, or click ![icon] on the **Standard** toolbar.

2. In the **Import** tab, click ![icon] .

3. In the **Open** dialog, open the `bumper_mid.hm` model file.

4. Click *Import*. HyperMesh imports the `bumper_mid.hm` model file on top of the existing data.
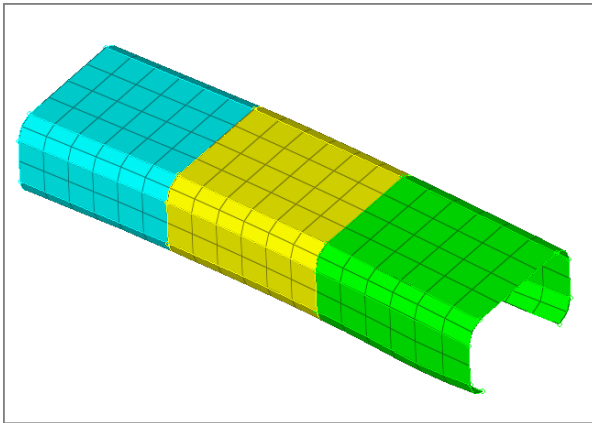

Step 3: Import the IGES geometry file, `bumper_end.iges`, into the current HyperMesh session.

1. In the **Import** tab, click ![icon] .

2. From the **File type** list, select *IGES*.

3. Click ![icon] .

4. In the **Select IGES file** dialog, open the `bumper_end.iges` geometry file.

5. Click *Import*. HyperMesh imports the IGES geometry file's data.

△ Altair
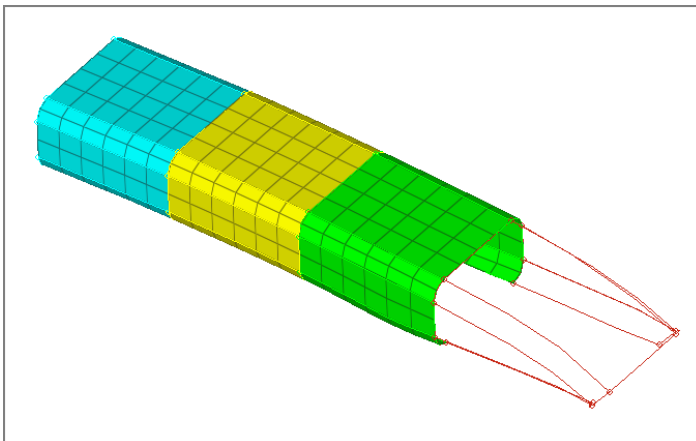
Step 4: Import the OptiStruct input file, `bumper_end_rgd.fem`, into the current HyperMesh session.

1.   In the **Import** tab, click .

2.   From the **File type** list, select *OptiStruct*.

3.   In the **File** field, open the `bumper_end_rgd.fem` input file.

4.   Click *Import*. HyperMesh imports a mesh for the bumper's end portion to the geometry representing the bumper's end.
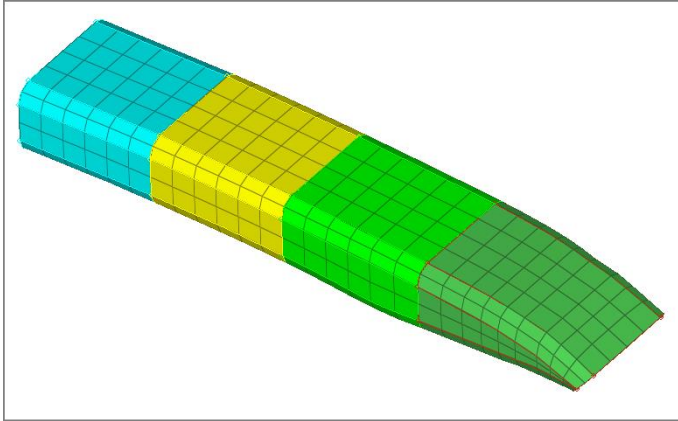
Step 5: Save the HyperMesh session as a HyperMesh model file called `practice.hm`.

1.   From the menu bar, click *File* > *Save As* > *Model*.

2.   In the **Save Model As** dialog, navigate to your working directory and save the data in your current session as a binary data file labeled `practice.hm`.

Step 6: Export the model's geometry data to an IGES file called `practice.iges`.

1.   To access the **Export** tab, click *File* > *Export* > *Geometry* from the menu bar, or click on the **Standard** toolbar.

2.   From the **File type** list, select *IGES*.

3.   In the **File** field, navigate to your working directory and save the file as `practice.igs`.

4.   Click *Export*. HyperMesh exports all of the geometry data (points, lines, surfaces) loaded in your current session as a `.iges` file.

Step 7: Export the model's mesh data to an OptiStruct input file called `practice.fem`.

1.   In the **Export** tab, click .

2.   From the **File type** list, select *OptiStruct*.

△ Altair

3. In the **File** field, navigate to your working directory and save the file as `practice.fem`.

4. Click **Export**. HyperMesh exports all of the finite element data (nodes, elements, loads) loaded in your current session as a `.fem` file.

Step 8 (Optional): Delete all data from the current HyperMesh session by starting a new session.

1. To open a new HyperMesh model, click **File** > **New** > **Model** from the menu bar, or click on the **Standard** toolbar.

2. In the **HyperMesh** dialog, click **Yes** if you would like to discard all current model data.

Step 9 (Optional): Import the IGES geometry file you created, `practice.igs.`

1. To import the `practice.igs` file into your current session repeat step 3. HyperMesh imports the geometry data in the file to the existing data.

Step 10 (Optional): Import the Optistruct input file you created, `practice.fem`, into the current HyperMesh session.

1. To import the `practice.fem` file into the current session, repeat step 4. HyperMesh imports the data in the file to the existing data.

With the completion of Steps 8, 9, and 10, your current HyperMesh session should contain all of the geometry and mesh data that existed in the HyperMesh session that you saved to a HyperMesh file in Step 5.

Step 11 (Optional): Save your work.

1. From the menu bar, click **File** > **Save** > **Model**.