Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-4350: Pre-Processing for Crashing Tubes Analysis using Abaqus

HM-4350: Pre-Processing for Crashing Tubes Analysis using Abaqus

For this tutorial it is recommended that you complete the introductory tutorial, HM-1000: Getting Started with HyperMesh as well as the tutorial HM-4310: Defining Abaqus Contacts for 2-D Models in HyperMesh. Working knowledge of the creation and editing of collectors and card images are a pre-requisite.

In this tutorial you will learn how to setup an Abaqus input file in HyperMesh, which will be used to obtain the dynamic response of multiple tubes with one tube fully constrained, and gravity applied on the other tubes. The modeling steps that are covered are:

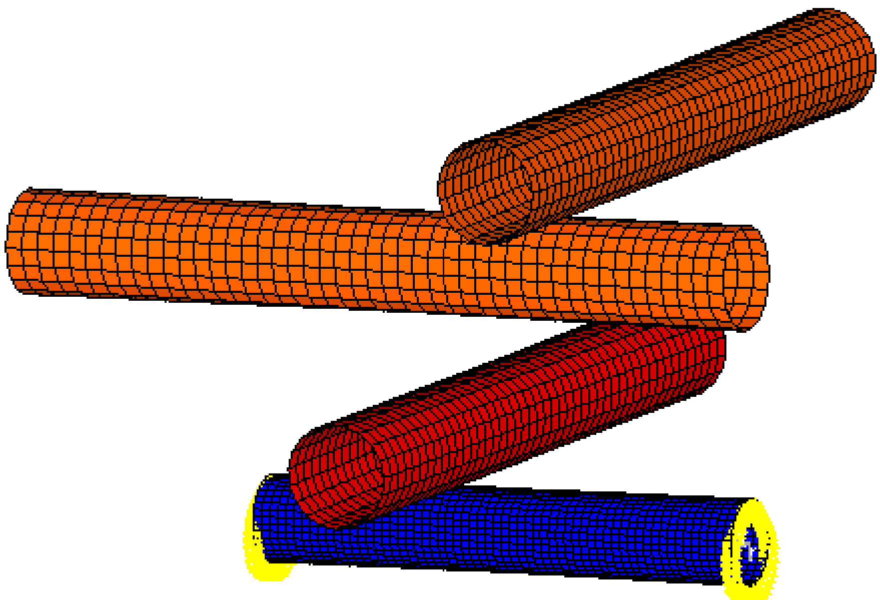
- Create *ORIENTATION system
- Create contact between shell elements
- Create a step with *AMPLITUDE associated to *DLOAD

The units used in this tutorial are Milliseconds, Millimeters, Kilograms, and Kilonewtons (ms, mm, kg, kN), and the tutorial is based on Abaqus 6.9-EF1.

For more information regarding the panels used in this tutorial, please refer to the Panels section of the on-line help, or click the ***h*** key while in the panel to bring up its context sensitive help. For detailed information on the HyperMesh Abaqus interface, refer to the External Interfacing section of the on-line help.

This tutorial requires about 30 minutes to completed.

The model used is composed of four tubes (see image below).



Crashing tubes

Model Files

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

Exercise

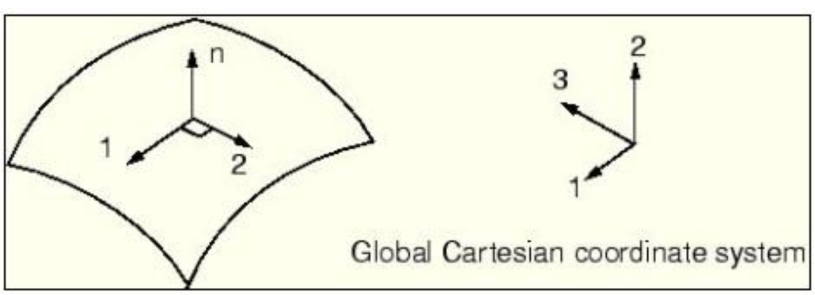
Step 1: Load the Abaqus Explicit user profile and retrieve the model

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to ***Abaqus***, ***Explicit***.
3. Open a model file by clicking ***File*** > ***Open*** > ***Model*** from the menu bar, or clicking  on the **Standard** toolbar.
4. In the **Open Model** dialog, open the `crash_tubes.hm` file. The model contains the following Abaqus model and history data:
    - Four tubes with shell (S4R) elements. The corresponding ELSETs are named **FixTube**, **MovTube**, **MovTube2** and **MovTube3**.
    - A *SHELL SECTION property for each tube. Each property is associated with one of two materials.
    - *BOUNDARY constraints on the ELSET named **FixTube**.
    - A HyperMesh system.

Defining the Abaqus *ORIENTATION in HyperMesh

*ORIENTATION specifies a local system defining local material directions for elements.

In Abaqus, shell and membrane elements have default local directions. They are not the global system directions. The default local 1-direction is the projection of the global x axis direction onto the shell surface. If the global x axis is normal to the shell surface, the local 1-direction is the projection of the global z axis onto the shell surface. The local 2-direction is perpendicular to the local 1-direction in the surface of the shell. Refer to the figure below.



Default local shell directions

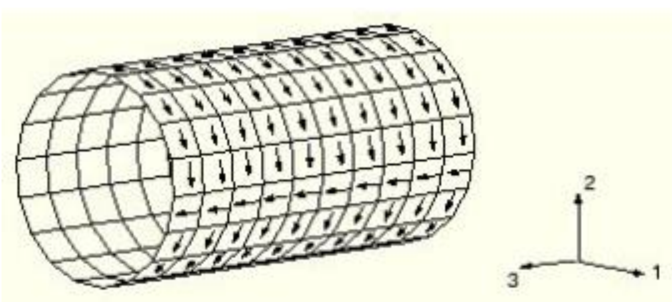
The general steps outlined below can help you understand the process followed in this tutorial.

1. Create a **System Collector** with no card image and give it a name as per your preference.

   Note: Any number of systems can be collected in a system collector.
2. Create a system by clicking ***Geometry*** > ***Create*** > ***System*** from the menu bar.
3. Create the ***ORIENTATION** using the **Card** panel.
4. In the **Card** panel, select the ***HyperMesh system (systs)*** and click ***edit***.
5. Activate the **ORIENTATION** option to create the ***ORIENTATION** keyword.
6. If the ***ORIENTATION** system is for solid elements, do not activate the locdir_alpha option. If this ***ORIENTATION** system is for shell and membrane elements, activate the locdir_alpha option. By default, the local axis closest to being normal to the elements' 1 and 2 material directions is the local 1-axis. Also by default, the additional rotation about the local normal axis is 0. You can change these values by editing the [locdir] and [alpha] fields in the pop-up card image.
7. Associate the ***ORIENTATION** to the desired sectional properties.

Local directions for this model

The default set of local material directions can sometimes cause problems; a case in point is the model's fixed tube pictured below. For most of the elements in the tube, the local 1-direction is circumferential. However, there is a line of elements normal to the global x axis. For these elements the local 1-direction is the projection of the global z axis onto the shell, making the local 1-direction axial instead of circumferential. A contour plot of the direct stress in the local 1-direction will look strange, since for most elements, it is the circumferential stress, whereas for some elements it is the axial stress. In this case, use the *ORIENTATION option for the fixed tube to define more appropriate local directions.

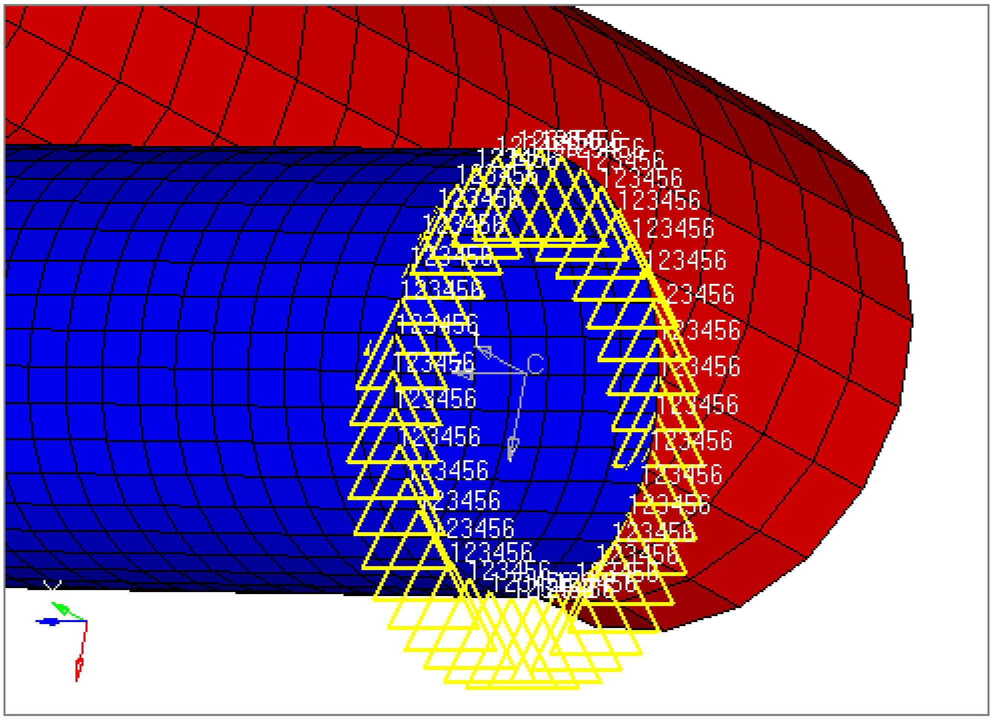


Default local 1-direction in the fixed tube

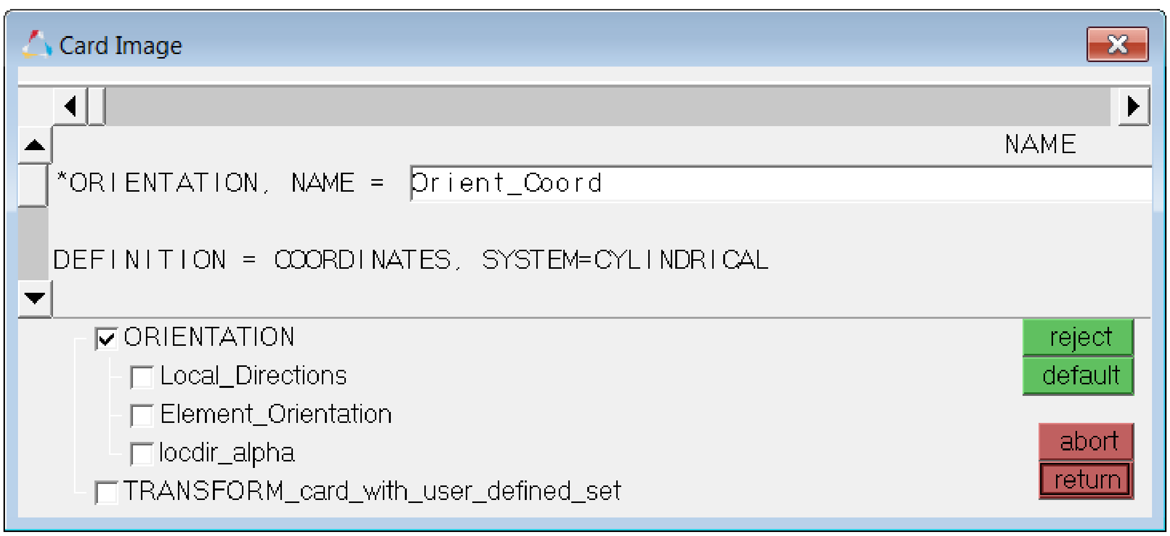
Step 2: Create the *ORIENTATION for the fixed tube

In this step, you will use the approach described in the previous section to create an *ORIENTATION for the fixed tube. Use the pre-defined cylindrical coordinate system for this tube and define the card using the **Card** panel.

1. Use your mouse to position the model to the view shown below. This system is located at one end of the fixed tube and is organized in the system collector.

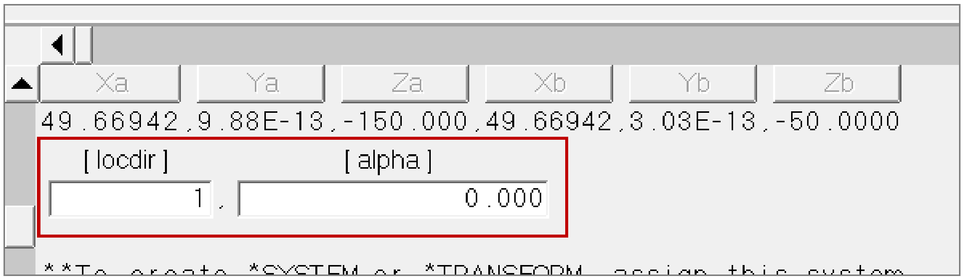


2. From the menu bar, click ***Geometry*** > ***Card Edit*** > ***Systems*** to edit the system's card and define the *ORIENTATION option.
3. Using the **systs** selector, select the local system.
4. Click ***edit***.
5. In the **Card Image**, select the **ORIENTATION** checkbox.
6. In the ***ORIENTATION, NAME** field, enter `Orient_Coord`.

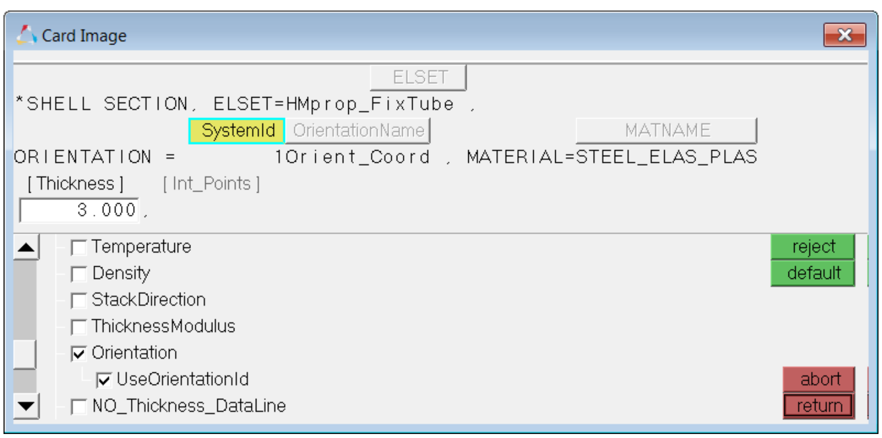


7. Select the **locdir_alpha** checkbox. The **locdir** and **alpha** fields display under *ORIENTATION in the card image.

   Tip: Use the vertical scroll bar to display the **locdir** and **alpha** parameters if they are not visible.



8. Leave the **locdir** field set to `1` to specify the radial axis as the axis closest to being normal to the shells' 1 and 2 material directions.
9. Leave the **alpha** field set to `0` for the additional rotation of the local normal axis.
10. Click ***return*** to close the card image.
11. In the card editor, set the **entity selector** to ***props***.
12. Click ***props***.
13. Select the property, ***FixTube***.
14. Click ***select***. *ORIENTATION is now associated with the fixed tube's sectional property.
15. Click ***edit***. The **Card Image** opens, and displays *SHELL SECTION, ELSET = FixTube.
16. Select the **Orientation** checkbox.
17. Select the **UseOrientationId** checkbox.
18. Click the ***SystemId*** selector and graphically select the system. This method also assigns the system name to the card image.



19. Click ***return*** to close the card image.
20. Click ***return*** to exit the panel.

*ORIENTATION has now been defined for ELSET FixTube.

Step 3: Define contact between the tubes as Abaqus general contact

General contact is usually easier to set up than common contact between two surfaces. Follow the steps below to set up a general contact.

The following is the simplest definition of general contact:

*CONTACT

*CONTACT INCLUSIONS, ALL EXTERIOR

You can assign other contact properties within a general contact using the following option.

*CONTACT PROPERTY ASSIGNMENT

surf_1, surf_2, prop_1

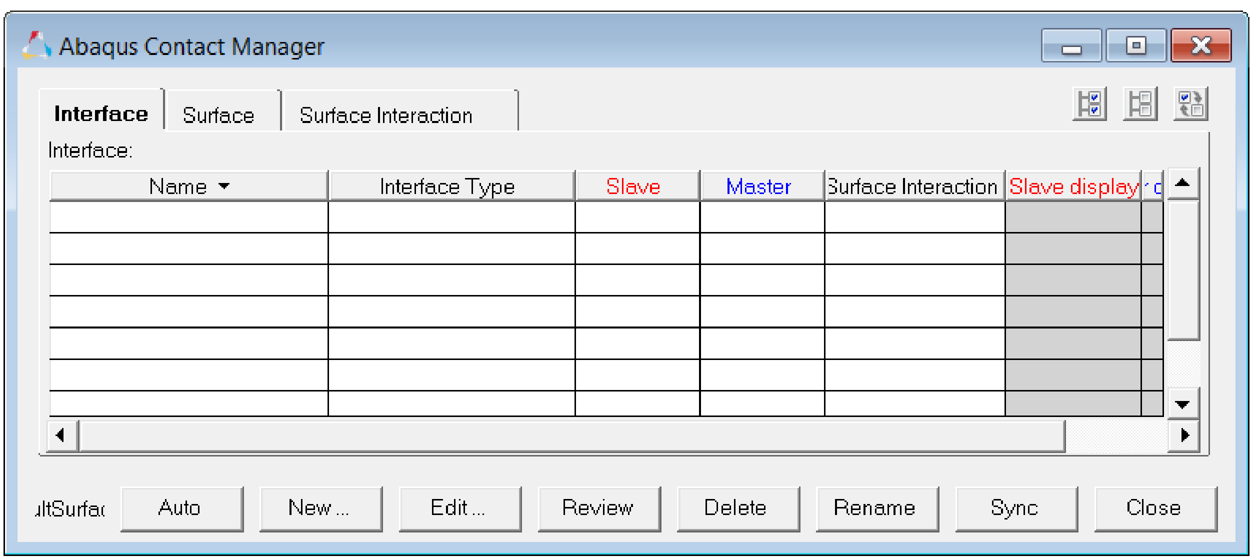
In this section, you will use the **Contact Manager** to define a contact pair property between the FixTube and the MoveTube (the closest tube to the fixed tube). Then you will define a general contact for the entire model and assign the contact pair property to it.

The general contact algorithm is used to define contacts between the tubes. A contact pair property is assigned to the general contact to define a different type of contact algorithm between the FixTube and the MoveTube. This contact pair property is not required. However, it is created here for the purpose of demonstrating how it is specified in a general contact using HyperMesh.

In a model like this, where both components have similar geometry (mesh) and material properties, either the fixed or moving tube can be chosen for the slave or master surface. Here use the ELSET FixTube for the slave surface of the contact pair property.

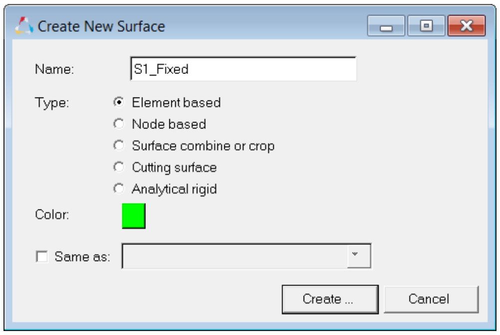
Complete the steps below to create a slave *SURFACE on FixTube by selecting elements in the **Contact Manager**:

1. From the menu bar, click ***Tools*** > ***Contact Manager***. The **Contact Manager** opens.

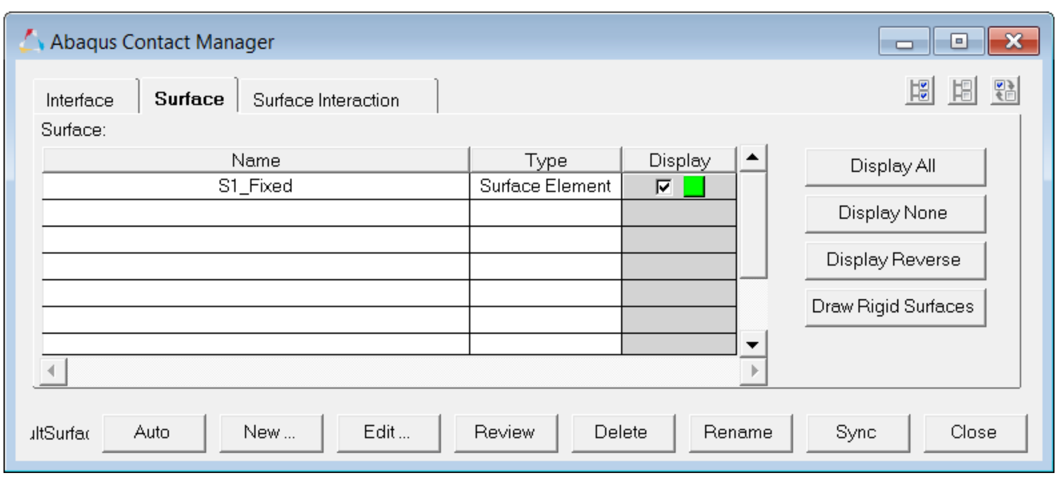


2. Click the ***Surface*** tab.
3. Click ***New*** to define a surface.

4. In the **Create New Surface** dialog, **Name** field, enter `S1_Fixed`.
5. Leave the surface **Type** set to **Element based**.
6. Optional. Choose a **Color** for visualizing the surface.

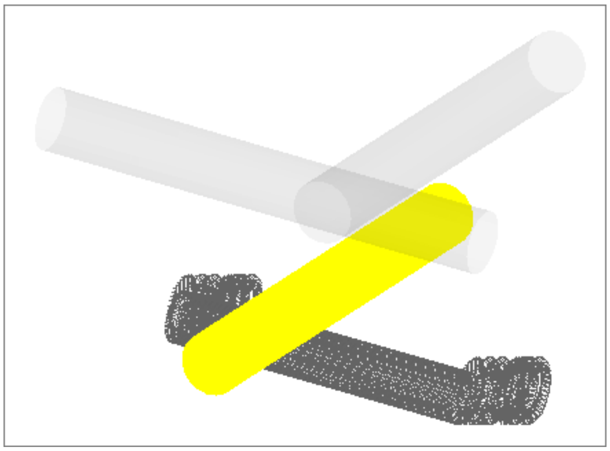


7. Click ***Create***. The **Element Based Surface** dialog opens.
8. In the **Define** tab, set **Define surface for** to ***3D shell, membrane, rigid***.
9. Click ***Elements*** to select the elements on which the surface will be defined.
10. In the panel area, click ***elems >> by collector***.
11. Select the component, ***FixTube***.
12. Click ***select***.
13. Click ***proceed***. The normals for the selected elements display. The normals should be pointing out of the fixed tube, which indicates the desired direction.
14. Optional. SPOS will be written to the input file for the elements in this contact surface. Specify SNEG in the input file by selecting the **Reverse** checkbox in the **Contact Manager** before going to the next step. This does not change the element normals.
15. Click ***Add*** to add the elements to the surface.
16. Click ***Close*** to return to the **Contact Manager**. Notice the surface **Sl_Fixed** is now listed in the **Surface** tab.

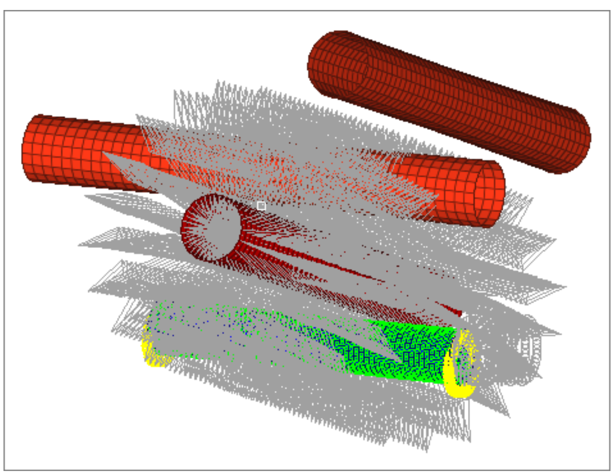


Step 4: Create master *SURFACE on MoveTube by selecting a set of elements in the Contact Manager

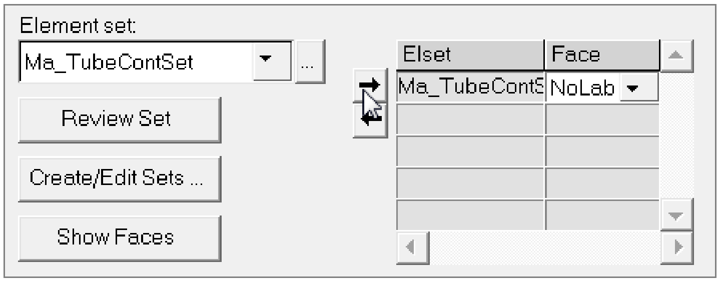
1. In the **Surface** tab, click ***New*** to begin defining a second surface.
2. In the **Create New Surface** dialog, **Name** field, enter `Ma_Moving`.
3. Leave the surface **Type** set to ***Element based***.
4. Optional. Select a **Color** for visualizing the surface.
5. Click ***Create***. The **Element Based Surface** dialog opens.
6. Set **Define surface for** to ***Element set.***
7. Set **Element set** to ***Ma_TubeContSet***.
8. Click ***Review Set***. The elements in the selected set highlight.



9. Return the elements to their original color by right-clicking on ***Review Set***.
10. Click ***Show Faces*** to view the direction of the element normals. The normals should be pointing into the moving tube, which indicates the faces on the inside of the moving tube elements are SPOS.

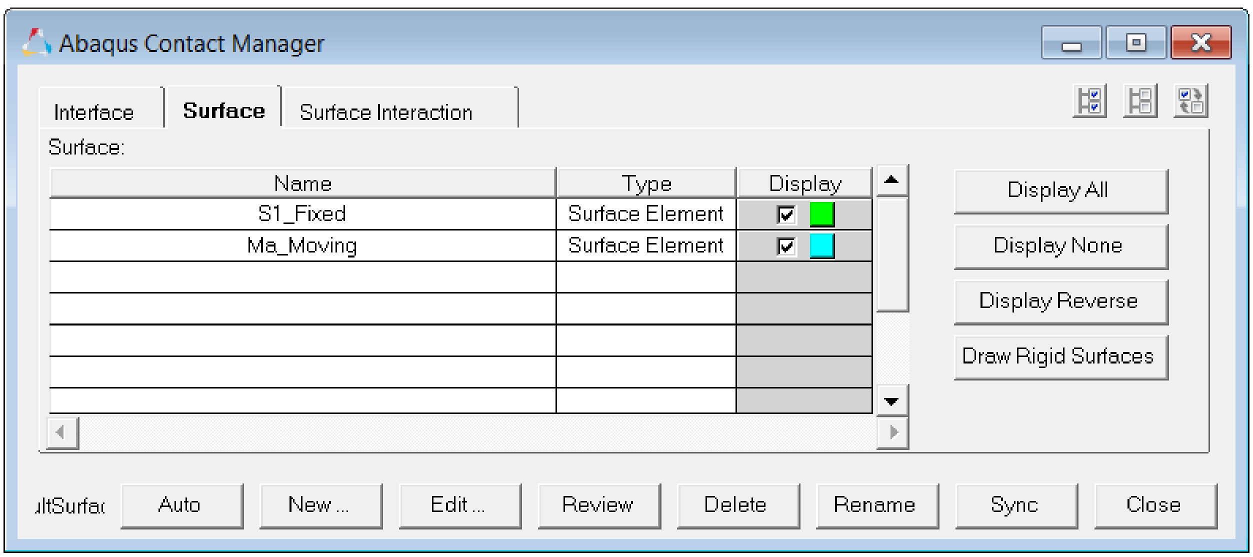


11. In the **Element Based Surface** dialog, click on the right arrow key to move the **Ma-TubeContSet** element set into the table.



12. In the **Face** column, click the pull down-menu and select ***SNEG***.

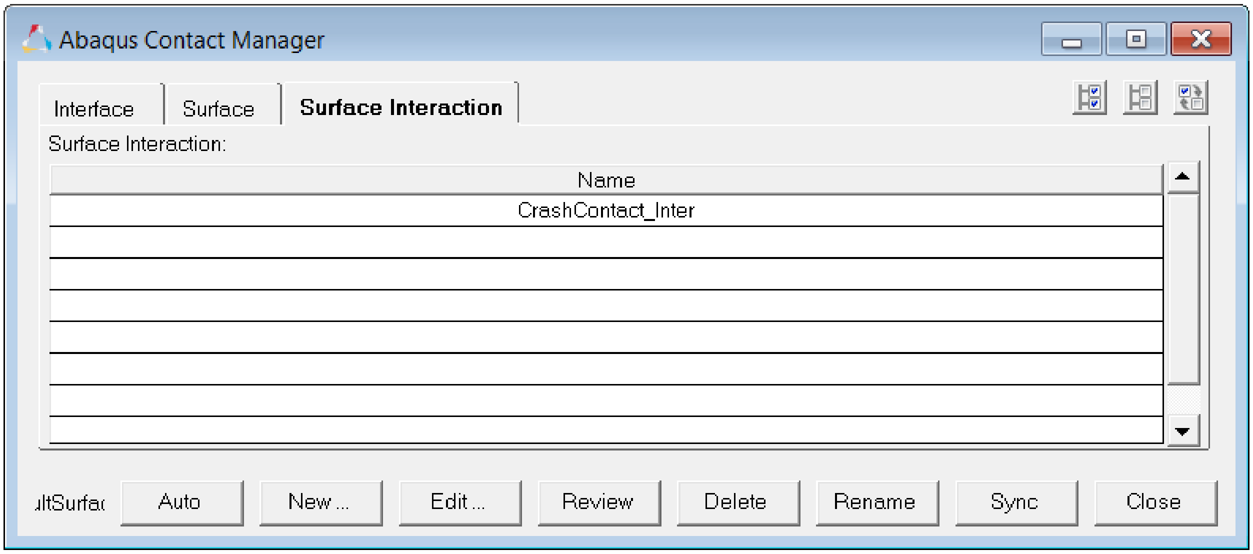
    This specifies the faces on the outside of the moving tube elements. SNEG is written to the input file for the set of elements forming this master contact surface.
13. Select the ***Display*** checkbox, then click ***Update***.
14. In the **Confirm** dialog, click ***Yes***.
15. Click ***Close*** to return to the **Contact Manager**. Notice the surface **Ma_Moving** is now listed in the **Surface** tab.



Step 5: Create *SURFACE INTERACTION in the Contact Manager

1. In the **Contact Manager**, click the **Surface Interaction** tab.
2. Click ***New*** to create a new surface interaction.
3. In the **Create New Surface Interaction** dialog, **Name** field, enter `CrashContact_Inter`.
4. Click ***Create***. The **Surface Interaction** dialog opens.
5. In the **Define** tab, select the **Friction** checkbox.
6. Click the **Friction** tab.
7. In the table at the bottom of the dialog, enter `0.2` in the **Friction Coeff** column.

8. Click ***OK*** to return to the **Contact Manager**. Notice the surface interaction **CrashContact_Inter** is now listed in the **Surface Interaction** tab.



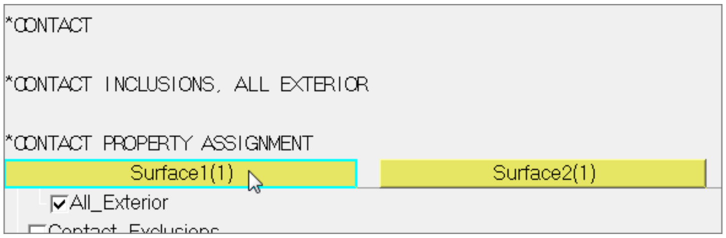
Step 6: Define a general contact *CONTACT in the Contact Manager

1. In the **Contact Manager**, click the **Interface** tab.
2. Click ***New*** to define a new interface.
3. In the **Create New Surface** dialog, **Name** field, enter `CrashContact`.
4. Set **Type** to **General Contact**.
5. Click ***Create***. The **Card Image** opens with *CONTACT shown.
6. Select the **Contact_Inclusions** checkbox to create *CONTACT INCLUSIONS.
7. Select the **All_Exterior** checkbox.

8. Select the **Contact_Property_Assignment** checkbox to create *CONTACT PROPERTY ASSIGNMENT.

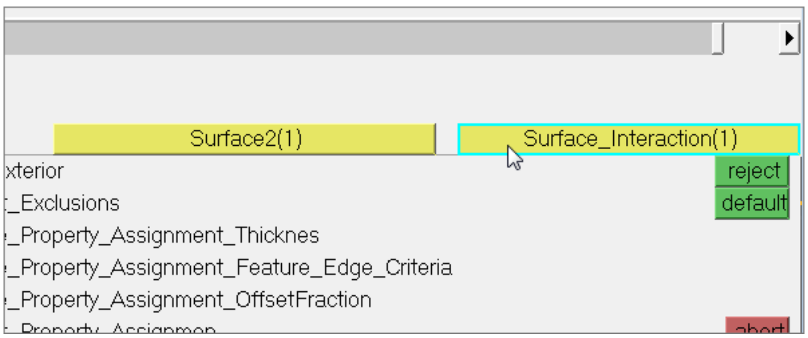
   Tip: Use the vertical scroll bar to display the **Contact_Property_Assignment** parameter if it is not visible.

9. Double-click the ***Surface1(1)*** selector.



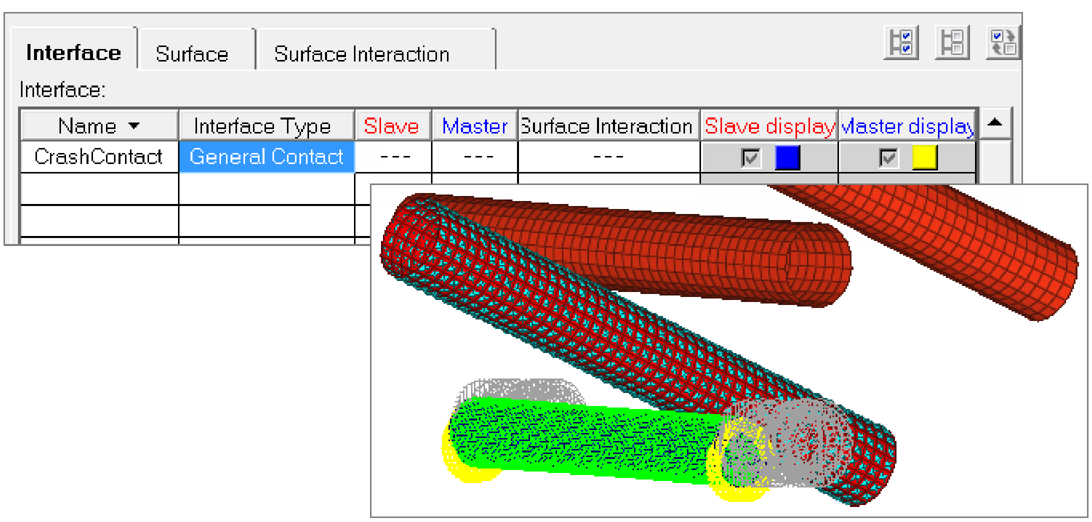
10. Select the surface, ***S1_Fixed***.
11. Double-click the ***Surface 2(1)*** selector.
12. Select the surface, ***Ma_Moving***.
13. Double-click ***Surface_Interaction(1)***.

    Tip: Use the horizontal scroll bar to display the ***Surface_Interaction(1)*** selector if it is not visible.



14. Select the interface, **CrashContact_Inter**.
15. Click ***return*** to return to the **Contact Manager**.
16. Click ***Close*** to close the **Contact Manager**.

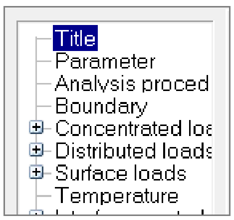
Defining the general contact between the tubes is complete.



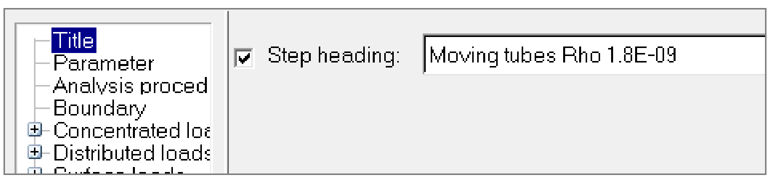
Step 7: Create *STEP

For this analysis, only one *STEP is needed. You will create a *DYNAMIC, EXPLICIT step and specify field output requests. Lastly, you will add the general *CONTACT and *SURFACE INTERACTION (groups) to the step. Adding the latter is required for Abaqus/Explicit, but not Abaqus/Standard. It is history data for Explicit.

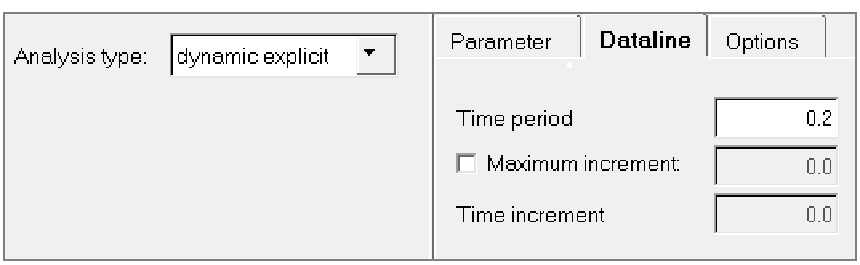
1. From the menu bar, click ***Tools*** > ***Load Step Browser***. The **Step Manager** opens.
2. In the **Step** tab, click ***New*** to begin defining a step.
3. In the **Create New Step** dialog, **Name** field, enter `Crash`.
4. Click ***Create***.
5. In the first pane, click ***Title***.



6. Select the **Step heading** checkbox, then enter `Moving tubes Rho 1.8E-09`.



7. Click ***Update***.
8. In the first pane, click ***Analysis procedure***.
9. Set **Analysis type** to ***dynamic explicit***. Additional tabs appear.
10. Click the ***Dataline*** tab.
11. In the **Time period** field, enter `0.2`.



12. Click ***Update***.
13. Close the ***Step Manager***.

Understanding boundary conditions for this model

In Abaqus/Standard, common boundary conditions are *BOUNDARY constraints to prevent rigid body motion and *CLOAD and *DLOAD (forces and pressures). In Abaqus/Explicit, common boundary conditions are constraints and time varying boundary conditions, like time varying displacement, velocity or acceleration, causing dynamic structural response.

For this analysis, the nodes at the ends of the fixed tube are fully constrained with *BOUNDARY constraints. These *BOUNDARY constraints are model data. In HyperMesh, they are organized into a load collector named "Constraint" with the card image INITIAL_CONDITION.

*DLOAD, TYPE = GRAVITY, AMPLITUDE = curve will be created for all nodes of the moving tubes. This is a constant acceleration applied in the global x direction. *AMPLITUDE is not required for gravity since it has a constant magnitude. However, *AMPLITUDE is assigned to the *DLOAD in this section for the purpose of showing you how to do this in HyperMesh. *AMPLITUDE allows arbitrary time variations of the applied condition throughout a step.

Step 8: Create *AMPLITUDE in HyperMesh

*AMPLITUDE is an xy curve in HyperMesh. There are two methods for creating *AMPLITUDE in HyperMesh.

Method 1:

Create *AMPLITUDE using the **Curve Editor**, which can be accessed by clicking ***XY Plots*** > ***Curve Editor*** from the menu bar. This is a quick and easy way to create new AMPLITUDE cards.

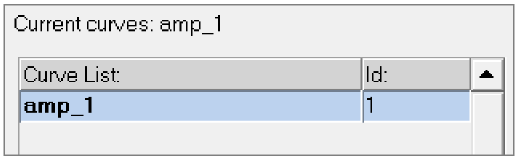
Method 2:

Create plots and curves by clicking ***XY-Plots*** > ***Create*** > ***Plots or Curves*** from the menu bar. This method provides additional functionalities, such as reading data from a file or generating curves by simple math. Please refer to XY Plotting in the online documentation for more information.

HyperMesh supports *AMPLITUDE with DEFINITION = TABULAR, EQUALLY SPACED and SMOOTH STEP. Use the **Step Manager** to associate a *AMPLITUDE to a load in HyperMesh.

Complete the steps below to create *AMPLITUDE in the **Curve Editor**.

1. From the menu bar, click ***XYPlots*** > ***Curve Editor***. The **Curve Editor** opens.
2. Click ***New*** to create a new curve.
3. In the panel area, enter `amp_1` in the **Name** field.
4. Click ***proceed***.
5. From the **Curve List**, select ***amp_1*** to activate the new curve.

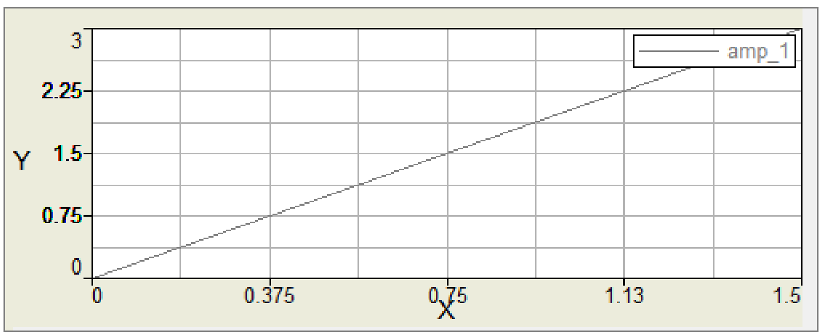


6. In the X, Y table, enter the following values.

   Tip: You can also copy and paste values column by column from a spreadsheet.

X	**Y**
0.0	0.0
0.5	1.0
1.0	2.0
1.5	3.0

7. Click ***Update***. The new amplitude curve displays.

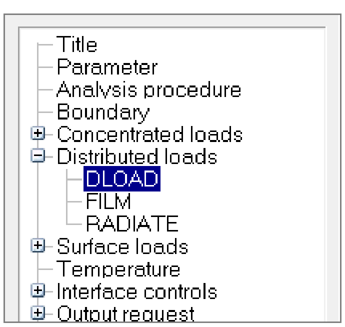


8. Click ***Close***.

Step 9: Define *DLOAD, TYPE = GRAVITY

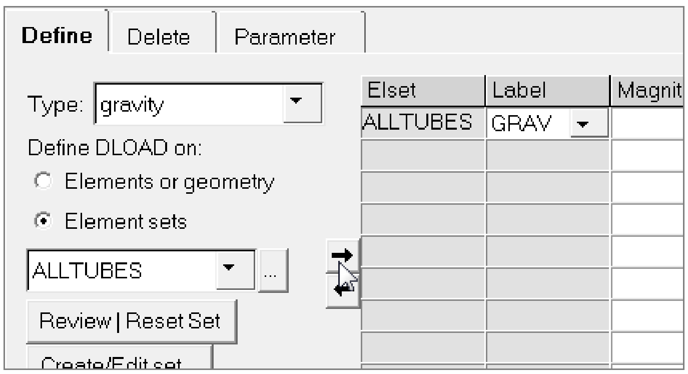
In this step you will define *DLOAD, TYPE=GRAVITY on all of the nodes of the moving tubes using the **Step Manager** for the step named Crash.

1. From the menu bar, click ***Tools*** > ***Load Step Browser***. The **Step Manager** opens.
2. In the **Step** tab, click the step **Crash**.
3. Click ***Edit***. The **Load Step** dialog opens.
4. In the first pane, expand **Distributed loads**, and click ***DLOAD***.



5. Click ***New***.
6. In the **Create Load Collector** dialog, **Name** field, enter `GRAVITY`.
7. Optional. Select a display **Color** for the **GRAVITY** load collector.
8. Click ***Create***.
9. In the **Load collector** table, click **GRAVITY** to make the collector active.
10. In the **Define** tab, set **Type** to ***gravity***.
11. Set **Define DLOAD on** to ***Element sets***.
12. Set **Element sets** to **ALLTUBES**.

    Tip: Click ... to view the enhanced browser, which provides filtering and sorting options for easier selections.
13. Click the right-arrow button to add the **ALLTUBES** set to the table.



14. In the **Magnitude** column, enter `9810`.
15. Enter `1`, `0`, `0` in the table for **Comp1**, **Comp2**, **Comp3** respectively. These values define a unit vector in the global x direction.

Elset	Label	Magnitude	Comp1	Comp2	Comp3
ALLTUBES	GRAV	9810	1	0	0

16. Click ***Update***.
17. Click the **Parameter** tab.
18. Select the ***Amplitude curve*** checkbox, then select ***amp_1***.
19. Click ***Review/Edit | Reset***.
20. Click ***Close*** to close the **Curve Editor.**
21. Click ***Update*** to update the step and write the changes to the database.

Step 10: Define output requests for the ODB file

In this step, you will use the **Step Manager** to define an output request for the step **Crash**.

1. In the first pane of the **Load Step** dialog, expand **Output request**, and click ***ODB file***.
2. Click ***New***.
3. In the **Create Output block** dialog, **Name** field, enter `field_output`.
4. Click ***Create.***
5. In the **Output block** table, click **field_output** to make it active.
6. In the **Output** tab, select the ***Output*** checkbox and set it to ***field***.
7. Select the **Node output** and **Element output** checkboxes.
8. Select the ***Time marks*** checkbox, and set it to ***yes***.
9. Select the ***Number interval*** checkbox, and specify `20` intervals.
10. Click ***Update***.
11. Click the **Node Output** tab.
12. From the list of output options, expand **Displacement**, and select ***U*** to request nodal displacement output.
13. Click ***Update***.
14. Click the **Element Output** tab.
15. From the list of output options, expand **Section_points** > **O***,* and select ***0***, ***1***, ***2***, ***3***, ***4***, and ***5*** to request results on element layers 1 through 5.
16. Expand **Stress** and select ***S*** to request element stress output.
17. Click ***Update***.
18. Close the **Load Step** dialog and **Step Manager**.

Step 11: Export the model

Use the steps below to export the model file as an INP file using the Explicit template.

1. From the menu bar, click ***File*** > ***Export*** > ***Solver Deck***.
2. In the **File** field, enter the file name as `crash_tubes_Complete.inp`.
3. Set **Template** to ***Explicit***.
4. Click ***Export***.

In this tutorial we introduced some of the concepts that govern the HyperMesh interface in Abaqus. We used the **Contact Manager** to setup a general contact between all of the tubes. We also used the **Step Manager** to do basic modeling in terms of Abaqus such as defining boundary conditions, output requests and steps.

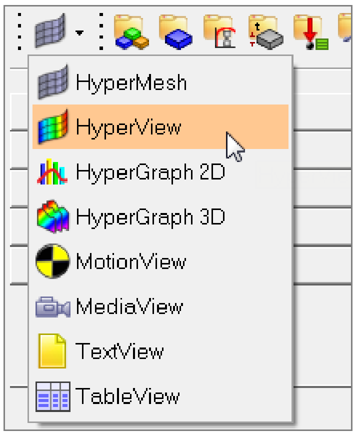
Notes:

- After you quit HyperMesh, you can run the Abaqus solver using the `job1.inp` file that was written from HyperMesh.
- At your site, you can use the ABAQUS license to run this model.

- If the batch mode option is being used, then enter the name of the `.inp` file exported in the previous step as the input file.
- After you have successfully completed the analysis, the result file will be available in your working directory with the name `<jobname.odb>`.
- Use HvTrans to translate the Abaqus solver result file to an H3D file.

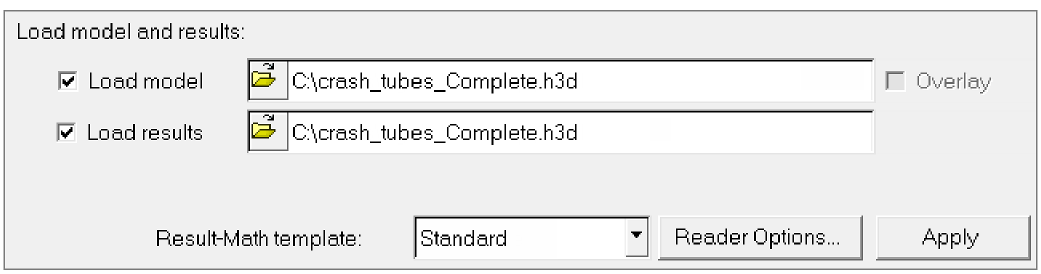
Step 12: Open HyperView from the Application Menu

1. On the **Client Selector** toolbar, select ***HyperView***.The HyperView environment displays.

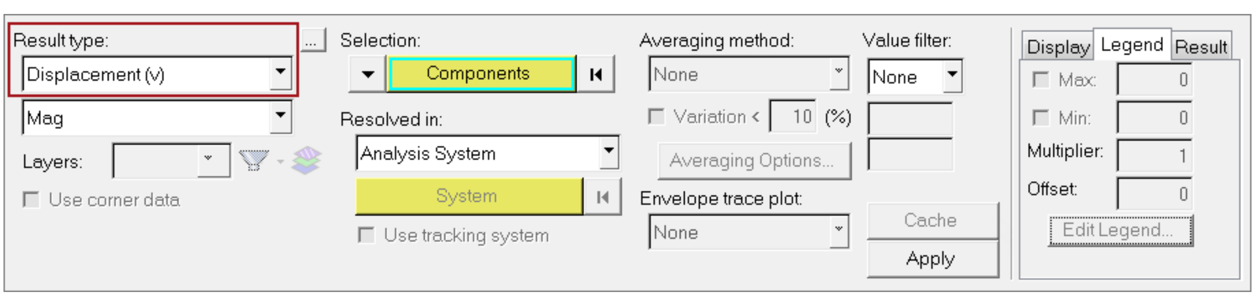


2. In the panel area, load the model and results files.

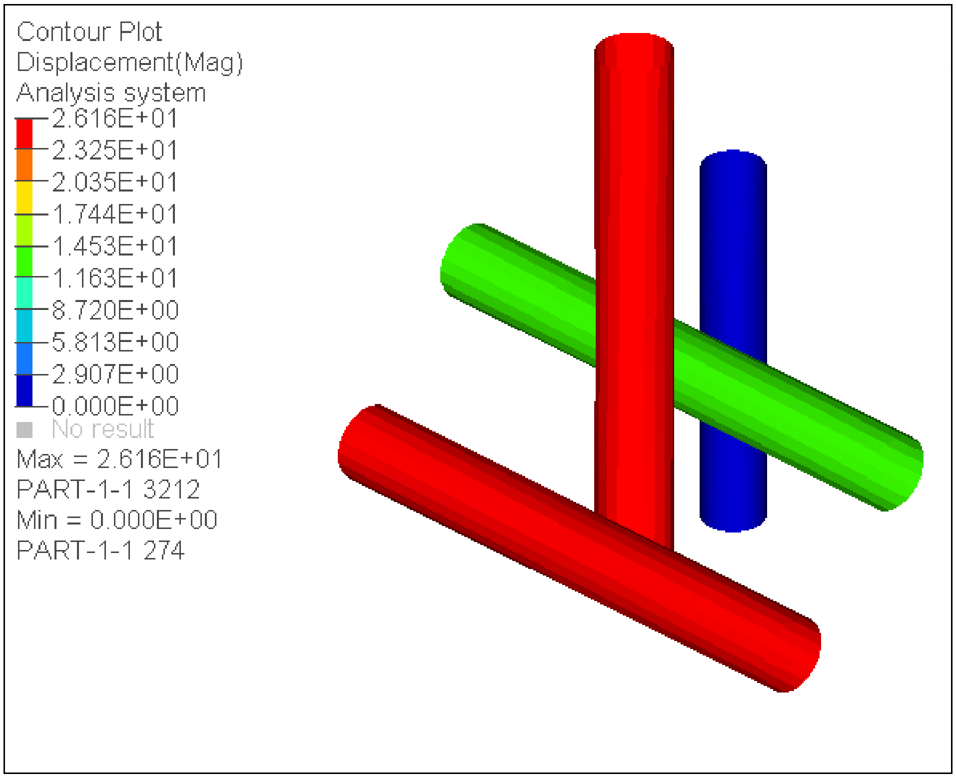
   Note: Load *.h3d files for both the model and result files.



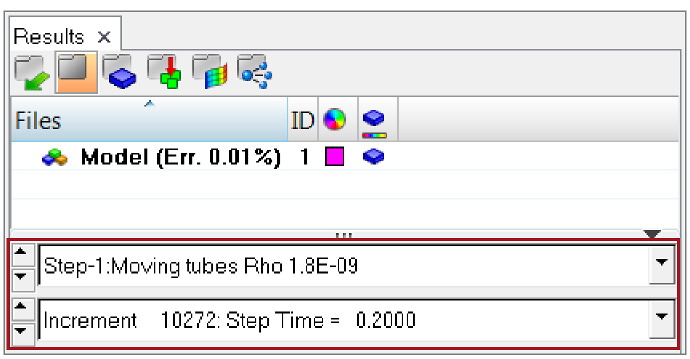
3. Click ***Apply***.
4. On the **Results** toolbar, click to open the **Contour** panel.
5. Review displacement (v) results by setting the **Result type** to ***Displacement (v)***.



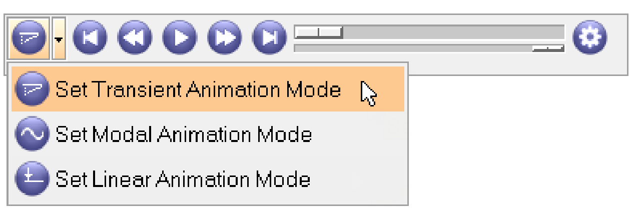
6. Click ***Apply***.



7. In the **Results** browser, review steps and increments.

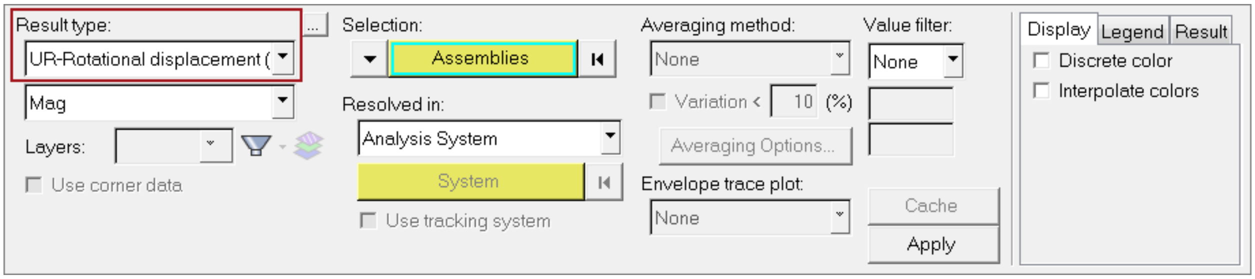


8. On the **Animation** toolbar, set the animation mode to **transient**.



9. Review the animation by clicking .

10. Review UR-Rotational displacement (v) results by setting the **Result type** to ***UR-Rotational displacement (v)*** in the **Contour** panel.



11. Click ***Apply***.