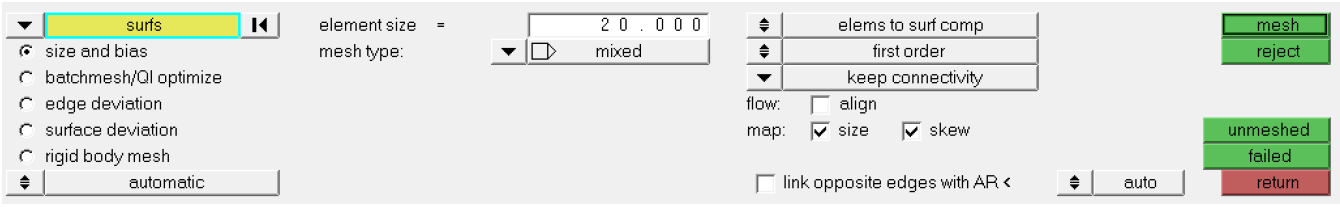
Includes and Fasteners

Local mesh manipulation, model organization by includes, creation of fastener connections.

Step 1: Remesh the full model with 20 mm shell mesh

1. Open the model using ***File > Open > Model:*** and select HM-08_ae-01_00.hm.
2. Open the **Automesh** panel using ***Mesh > Create > 2D Automesh*** or ***F12***.
3. Select the ***displayed*** surfaces.
4. Enter an **elements size** of 20.0.
5. Click the switch to ***automatic*** and verify the option is set to **elems to surf comp**.

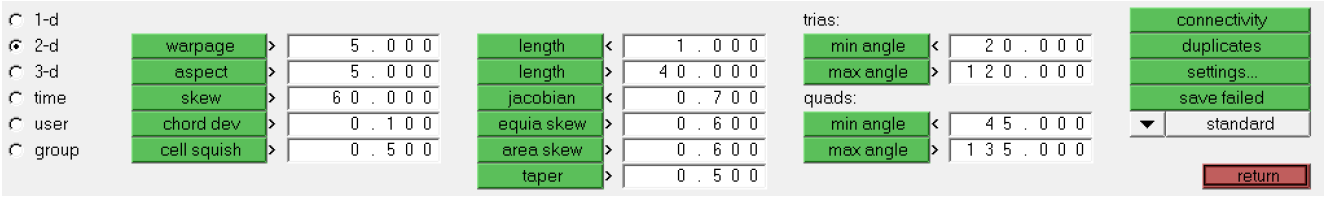


6. Click ***mesh*** to re-mesh the model.

   Note that because of “automatic” there was no second (interactive) stage of this panel, and because the original mesh was associated to the surfaces, it was replaced by the new mesh.

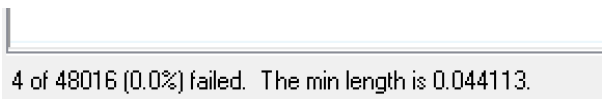
Step 2: Check the element quality and isolate failed ones with Patch Checker

1. Open the Check Elements panel using ***Mesh > Check > Elements > Check Elements*** or ***F10***.
2. Verify the ***2-d*** subpanel is active.
3. Enter 1.0 for the **length <** field.



4. Click the ***length <*** button.

   View the HyperMesh message bar of at the bottom left giving details about the failed elements.

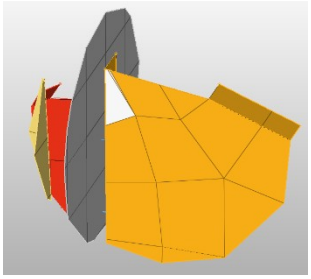


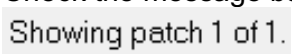
5. Click ***save failed*** to store the failed elements in HyperMesh’s temporary memory.
6. Turn on the Patch Checker toolbar using ***View > Toolbars > HyperMesh > Patch Checker***.

7. Verify that the selector is set to **Elements**  .

8. Activate Patch Checker by clicking on the icon  .

   Note that the model display will jump to a detail around the first patch of elements in HyperMesh's temporary memory (the ones saved just before), showing them in spherical clipping.

9. Check the message bar to see that patch one is shown and only one patch exists  .

   If there are additional patches, you can navigate to the next patch and increase/decrease the clipping radius.

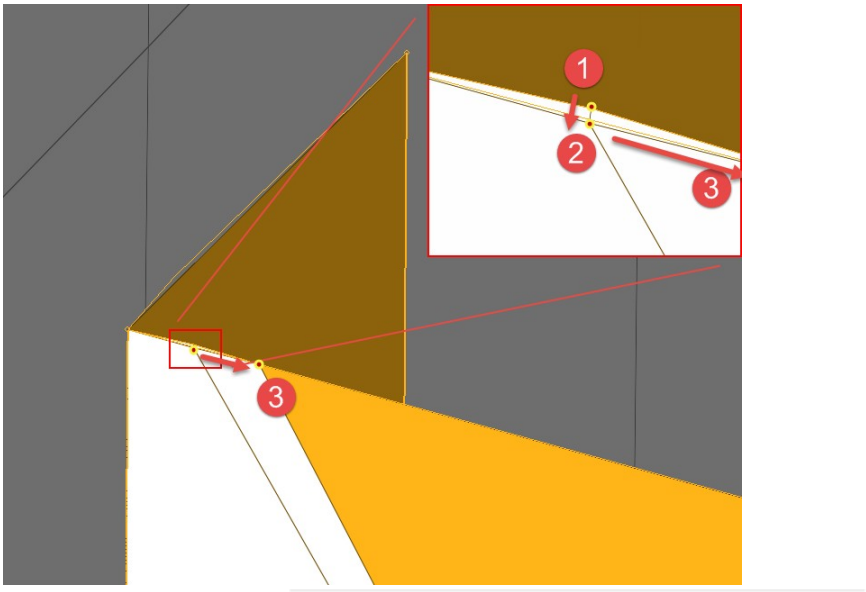
   

Step 3: Fix element quality locally by replacing nodes and adjusting distance

1. Zoom into the highlighted elements shown in the Patch Checker view and see that there are two very small and flat elements, due to a very small surface patch.

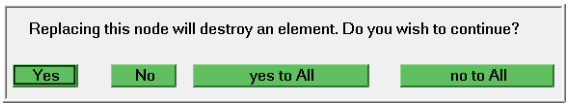
   While you could solve it by fixing the geometry and then remeshing it, the fastest way – described here – is to manipulate elements directly, without respecting geometry, using replace.

2. Activate the replace nodes panel using ***Mesh > Edit > Replace Nodes***.
3. Zoom into the detail so far that you can distinguish the nodes of the small, flat elements.
4. Click the outer node of the small elements (1) then on the inner one (2).

Note that you didn't need to change the selector button in the panel, it is switching automatically after selecting the first node.

5. A warning will pop up, informing that elements will be destroyed (and automatically deleted) by replacing theses nodes. This is what we want, and we want to do it one more time, so to not get this message again, confirm with ***yes to All***.

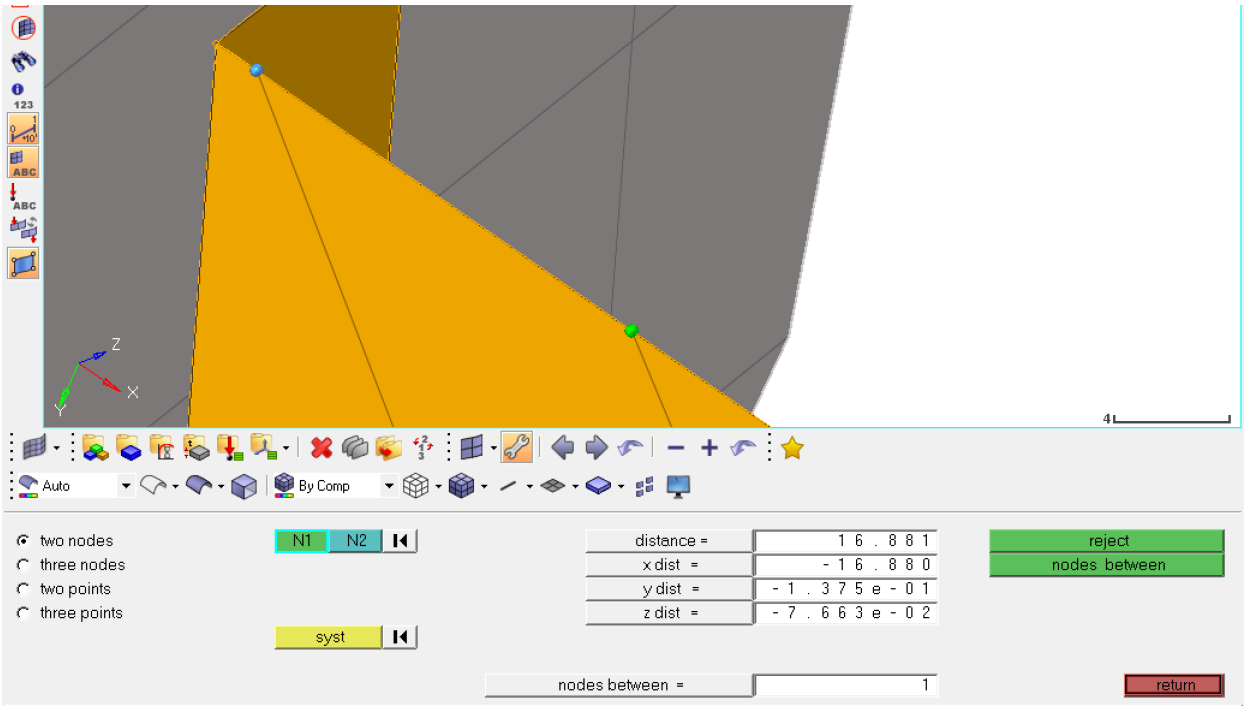


6. Now that the two small elements are fixed (deleted) we still need to fix the remaining larger but flat tria-element, click the remaining node from the last operation, then click the node to the right at the short edge of the tria-element.

   The node will be replaced and the collapsing element deleted with no further warning message.

   Finally, there is a quad-element remaining which has a very short edge. This can be fixed easiest by adjusting the node position using distance:

7. Open the **Distance** panel using ***Mesh > Check > Nodes > Distance***.
8. Select **N1** and **N2** in the order shown in the image.



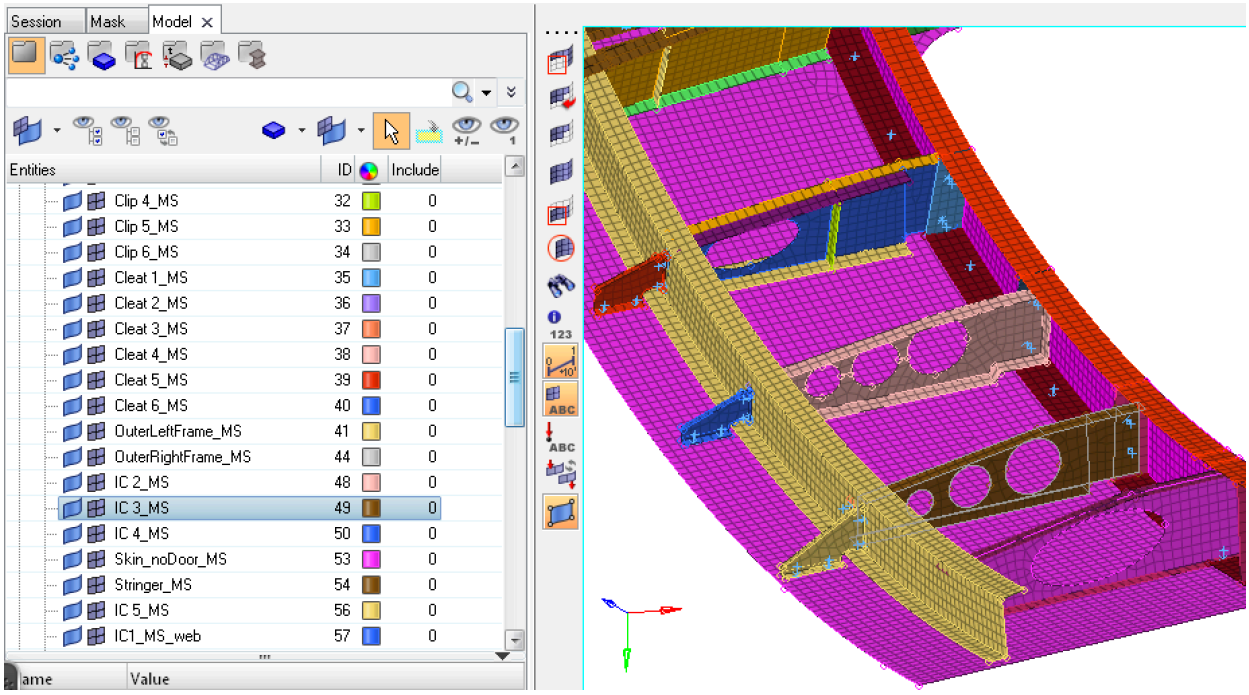
9. Enter 10.0 in the field of ***distance =***.

   N2 is adjusted to a distance of 10.0 to N1.

Step 4: Delete elements and geometry that will be replaced by new Part

1. Turn off the Patch Checker by clicking on the wrench symbol .
2. Activate the **IC3-delete** view in the Model Browser.
3. Click the ***Selector***  icon in the Model Browser to activate it.
4. Select the ***IC3_MS*** component in the display area.
5. Right-click on **IC3_MS** and click ***Delete*** from the dialogue.

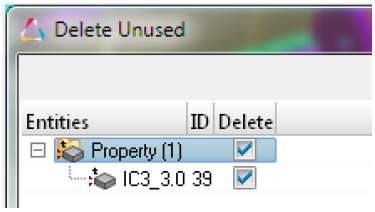
6. Confirm the message. See that the elements and surfaces disappear.



Find and delete unused Properties and empty Assemblies by deleting the elements of **IC3_MS** along with the property that is not used by any element anymore. Additionally, an Assembly that referenced this Component is empty now.

To clean up the model, use Unused and Empty in the RMB-menu of the Model Browser

7. Right click on the **Properties** folder and click ***Unused***.
8. In the dialog box, check the box of the unused property that was found and click ***Delete*** to remove the property.



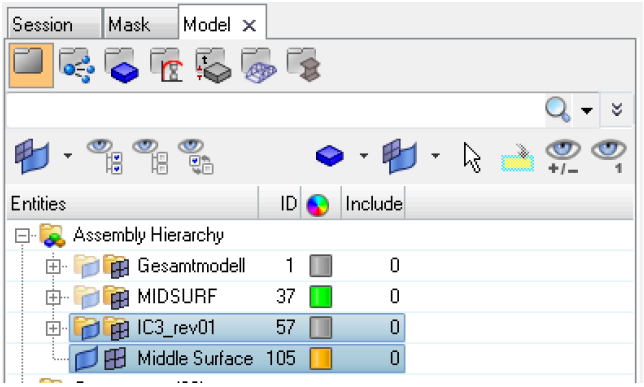
9. Right click on the **Assembly Hierarchy** folder and click ***Empty***.
10. Select the empty component and click ***Delete.***
11. A new HyperMesh model can now be imported to replace the part. Click ***File > Import > Model*** to open the Import tab.
12. Select the model HM-04_ae-01_08.hm from the list.

    A new meshed part appears where IC3_MS was deleted.

Step 5: Check Assembly Structure

1. Check the Assembly Hierarchy in the Model Browser.

   See that by importing the HyperMesh file you find the assembly IC3_rev01 from that file in the Hierarchy, as well as the component Middle Surface, which is not referenced by any assembly.



2. To review the model and find what was imported right click and select ***Isolate*** to display only the imported part.

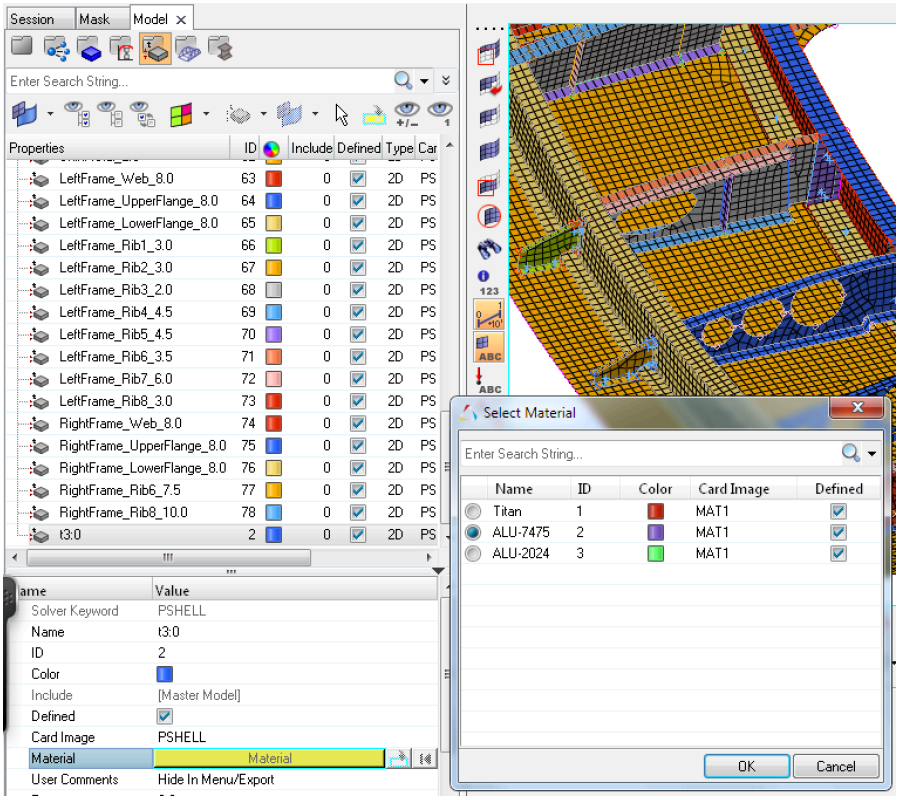
   You could use drag + drop to organize the assemblies and components – not done here.

Step 6: Check the property and assign the material to the new part

To be able to use the Fastener tool in correctly (but also to have all elements defined fully) all elements that will be connected need to have a material assigned. The new part imported in the last step has a PSHELL property, but no material assigned.

1. Open the **Properties View**  in the Model Browser.
2. Activate the ***Selector*** and click on the newly imported part in the display.

   Its property is highlighted in the browser list and property details are shown in the entity editor.
3. Click twice on the ***Material*** field to open the selection dialog.
4. Select ***ALU-7475*** from the list of materials.

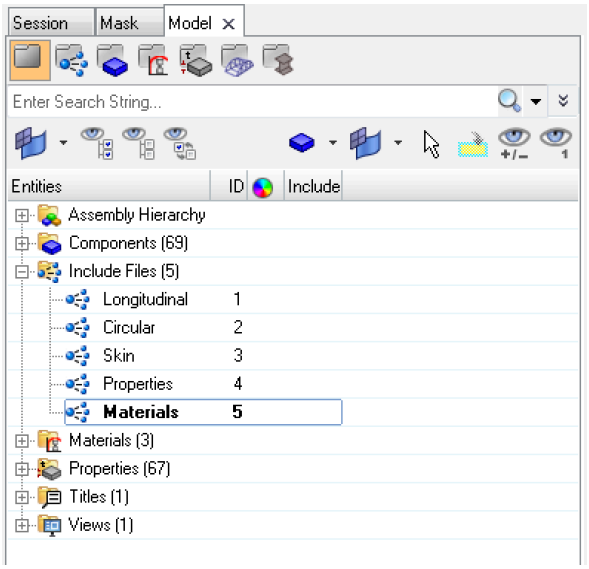


The property, and through it the elements, now has this material assigned.

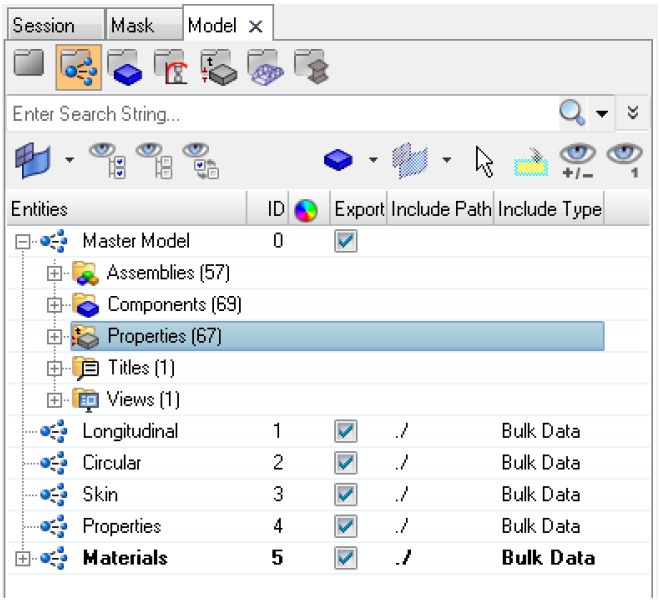
Step 7: Create and Organize Includes

While assemblies are a way to create a structure of Components (elements and geometry) inside HyperMesh, you may want to use the INCLUDE keywords to generate a data structure that is recognized by Nastran and OptiStruct. You can do so starting from the Model Browser. Assuming you want to organize your structure according to longitudinal, circular and skin structures into separate includes, as well as properties and materials:

1. Open the ***Model View***  in the Model Browser.
2. Right click in the Model Browser to open the menu and select ***Create > Include File***.
3. Create five includes named **Longitudinal, Circular, Skin, Properties** and **Materials**.



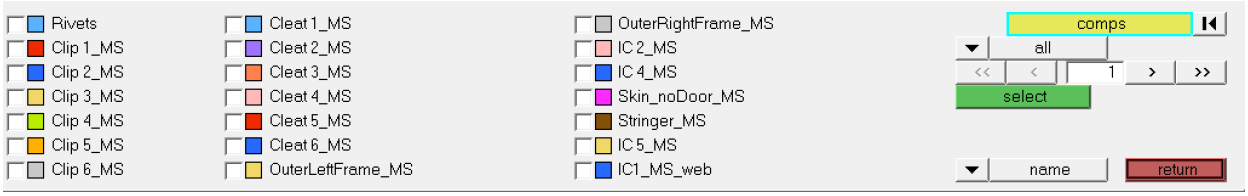
4. Switch to the ***Include View***  .
5. Expand the  Master Model in the list.
6. Move materials and properties into the respective includes by dragging and dropping the full category onto the include name.



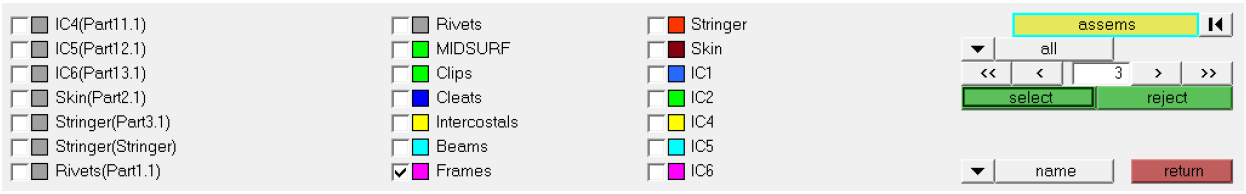
Dragging and dropping can also be done for components (respectively elements and nodes), but in case of many of them, would be time consuming. In this case, it is easier to use the organize functionality.

7. Open the organize panel using ***Collectors > Organize > Components***.

8. Click the ***comps*** selector to open the selection panel.



9. Click on the ***comps*** selector to open the extended entity selection and click ***by assems***.
10. Click the Next button > to browse to the **Frames** assembly and check the box.



11. Confirm the selection and leave the panel by clicking ***select***.
12. Click ***select*** again to return to the initial panel.
13. For **dest =** select the ***Circular*** include.
14. Verify the boxes to move nodes and elements are checked,
15. Click ***move*** to complete the organization of the entities.

While this was the fastest way to select and organize all circular components, (because they were referenced by one assembly) the easiest way to organize the remaining structures of skin and longitudinal structures into their respective includes is as follows:

16. In the **Include View**, click on the ***Entities*** column header to sort by name,

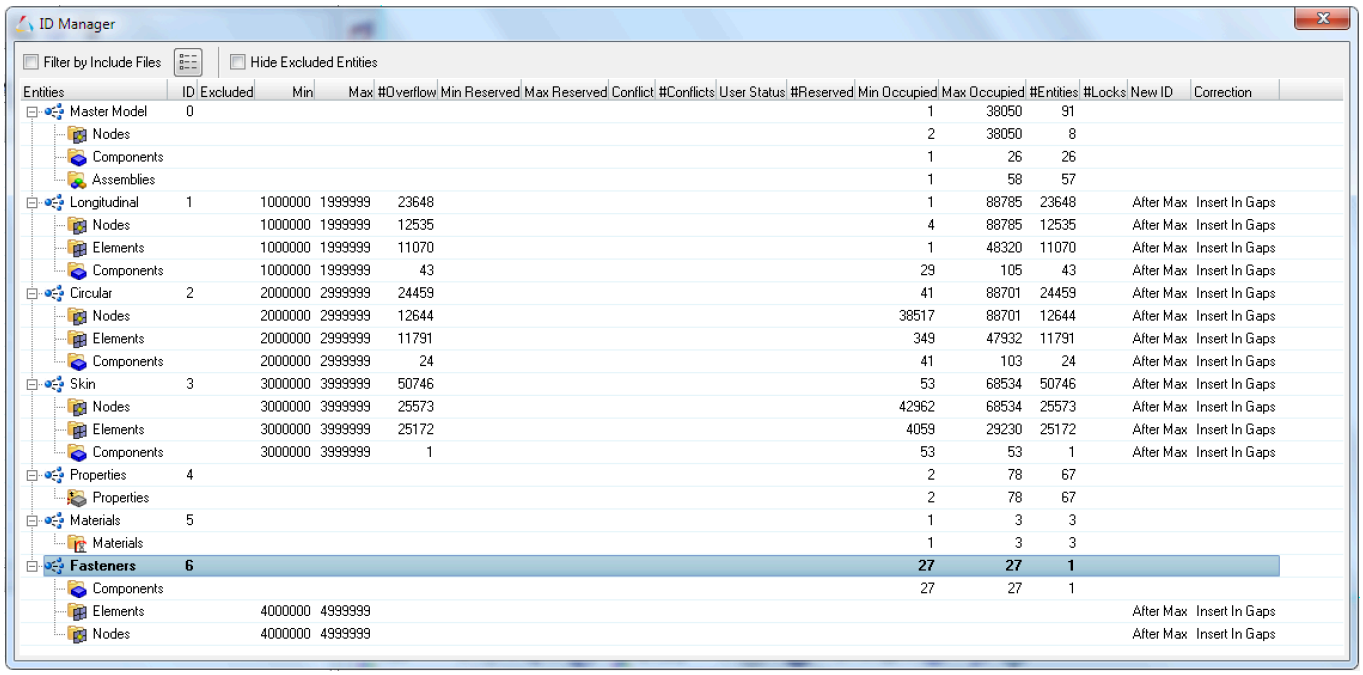
    This moves the skin component Skin_noDoor_MS close to the includes.
17. Drag and drop the component **Skin_noDoor_MS** into the include ***Skin***.
18. Confirm the message to move the elements and nodes.
19. Right click on the Master Model and click ***Show*** to make sure all remaining components in this include are displayed.
20. Right click on the Circular and Skin includes and click ***Hide*** to make sure all components that are already moved into these includes are not displayed.
21. Verify visually that only the longitudinal structures show up in the display area now.
22. Perform an autofit (CTRL + click MMB) to make sure nothing is outside the display area.
23. Open the Organize panel using ***Collectors > Organize > Components***.
24. Using SHIFT and dragging the LMB, draw a window to select all displayed components.

25. For **dest =** select the ***Longitudinal*** include.
26. Click ***move*** to complete the operation.

    All components (elements, nodes) should now be sorted to the respective includes.
27. Verify the components by right clicking on each include and click ***Isolate***.
28. To prepare for organization of the Fastener elements to be created, create another include, named **Fasteners**.
29. Organize the component **Rivets** (currently in the Longitudinal include) into Fasteners using the drag and drop method or ***Collectors > Organize***.

Step 8: Use ID Manager to define ID rules for includes

1. Open the ID Manager using ***Tools > ID Manager***.
2. Set ranges of allowed IDs inside each include of interest, as shown in the image.
3. For the **Fasteners** include, right click and select ***Create > ID-Range*** for Nodes and Elements.



Entities	ID	Excluded	Min	Max	#Overflow	Min Reserved	Max Reserved	Conflict	#Conflicts	User Status	#Reserved	Min Occupied	Max Occupied	#Entities	#Locks	New ID	Correction
Master Model	0											1	38050	91			
Nodes												2	38050	8			
Components												1	26	26			
Assemblies												1	58	57			
Longitudinal	1		1000000	1999999	23648							1	88785	23648		After Max	Insert In Gaps
Nodes			1000000	1999999	12535							4	88785	12535		After Max	Insert In Gaps
Elements			1000000	1999999	11070							1	48320	11070		After Max	Insert In Gaps
Components			1000000	1999999	43							29	105	43		After Max	Insert In Gaps
Circular	2		2000000	2999999	24459							41	88701	24459		After Max	Insert In Gaps
Nodes			2000000	2999999	12644							38517	88701	12644		After Max	Insert In Gaps
Elements			2000000	2999999	11791							349	47932	11791		After Max	Insert In Gaps
Components			2000000	2999999	24							41	103	24		After Max	Insert In Gaps
Skin	3		3000000	3999999	50746							53	68534	50746		After Max	Insert In Gaps
Nodes			3000000	3999999	25573							42962	68534	25573		After Max	Insert In Gaps
Elements			3000000	3999999	25172							4059	29230	25172		After Max	Insert In Gaps
Components			3000000	3999999	1							53	53	1		After Max	Insert In Gaps
Properties	4											2	78	67			
Properties												2	78	67			
Materials	5											1	3	3			
Materials												1	3	3			
Fasteners	**6**											**27**	**27**	**1**			
Components												27	27	1			
Elements			4000000	4999999												After Max	Insert In Gaps
Nodes			4000000	4999999												After Max	Insert In Gaps

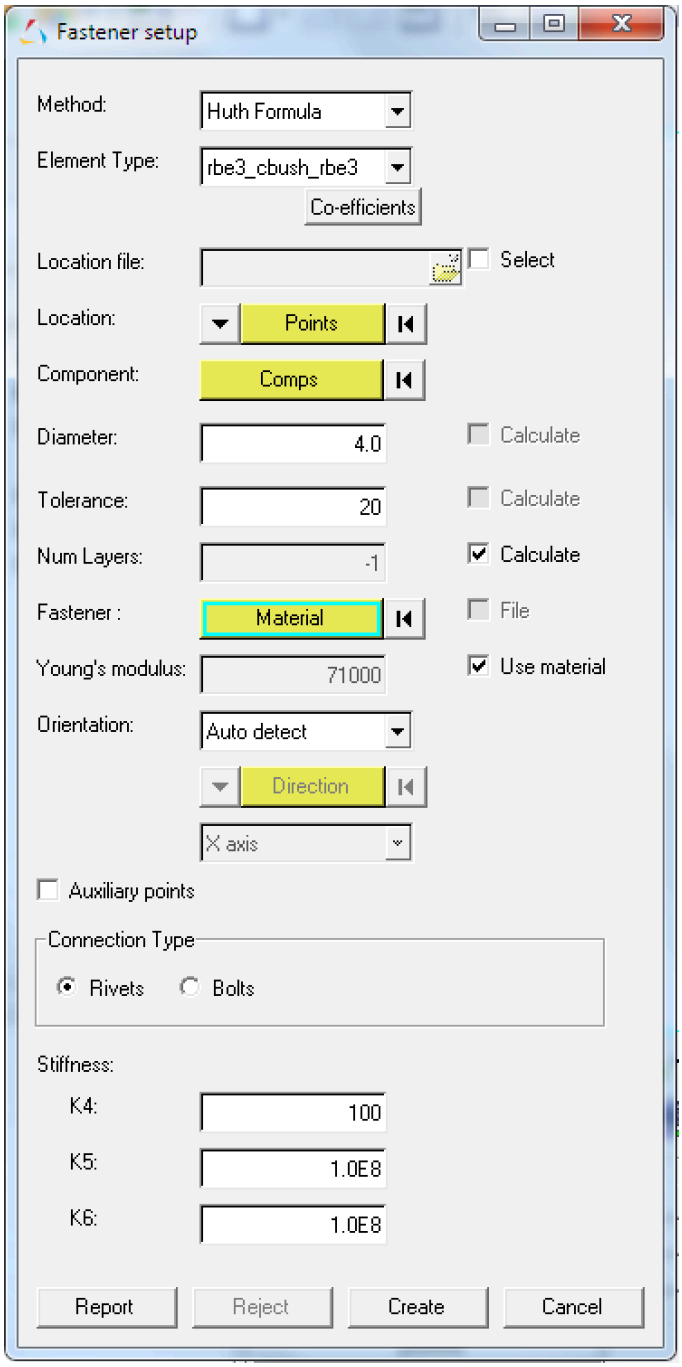
   The **#Overflow** column shows the number of entities with an ID out of the defined range.
4. Right click on individual or multiple includes and select ***Correct > Overflow*** to renumber entities with IDs out of the range.
5. Close the **ID Manager** dialog.

Step 9: Create Fasteners according to the Huth Formula

1. Open the fastener setup using ***Aerospace > Connections > Fastener setup***.

   Use this tool (which makes use of HyperMesh's connector technology in the background) to automatically create fastener connections (RBE3- CBUSH-RBE3) with stiffness according to Huth Formula.
2. Switch **Location** to ***Points***.
3. Click the ***points*** selector to open the extended entity selection.

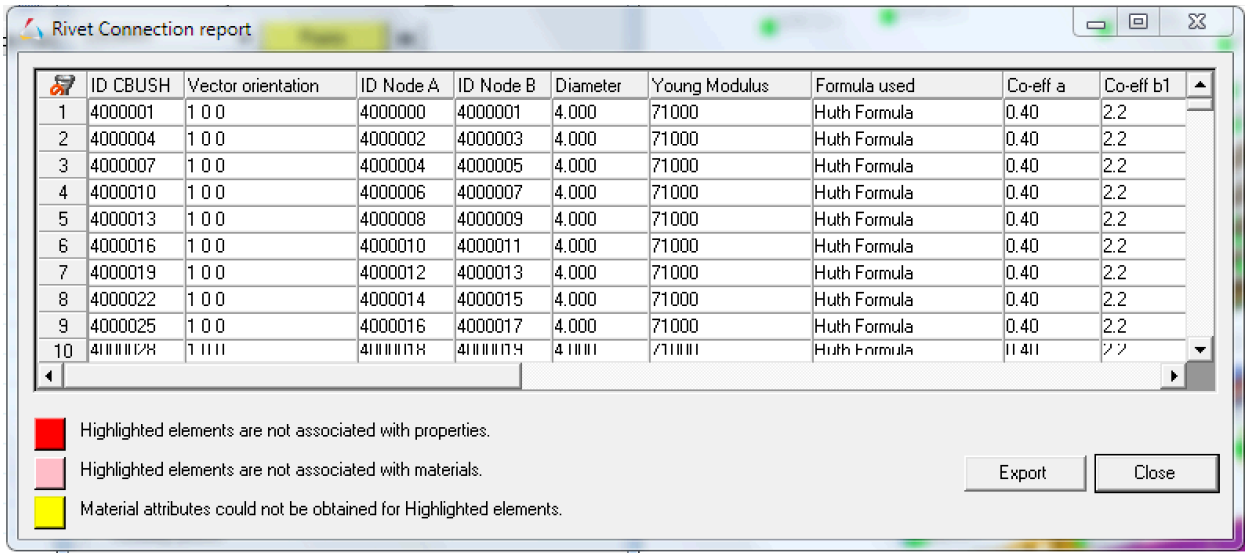
4. Click ***by collector*** and select the **Rivets** collector.
5. Click ***proceed*** to complete the selection.
6. Select ***all*** components.
7. Select the ***ALU-7475*** material.
8. Define the rest of the settings as shown in the image. – don't forget to switch **Connection Type** to Rivets.



9. Click ***Create*** to create the rivets.
10. Optionally, open a report table to check the realized connection types including stiffnesses.

    Note that the generated elements and connectors are organized in a new component rivetConn_comp1, that is organized in the **Fasteners** include. The fasteners wer placed there because Fasteners was the current include, because it was the latest one created.

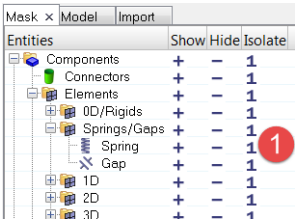
Note also that the new component, as well as the new nodes and elements were automatically renumbered to the ID range. You can check that with ***Tools > ID-Manager***.



	ID CBUSH	Vector orientation	ID Node A	ID Node B	Diameter	Young Modulus	Formula used	Co-eff a	Co-eff b1
1	4000001	1 0 0	4000000	4000001	4.000	71000	Huth Formula	0.40	2.2
2	4000004	1 0 0	4000002	4000003	4.000	71000	Huth Formula	0.40	2.2
3	4000007	1 0 0	4000004	4000005	4.000	71000	Huth Formula	0.40	2.2
4	4000010	1 0 0	4000006	4000007	4.000	71000	Huth Formula	0.40	2.2
5	4000013	1 0 0	4000008	4000009	4.000	71000	Huth Formula	0.40	2.2
6	4000016	1 0 0	4000010	4000011	4.000	71000	Huth Formula	0.40	2.2
7	4000019	1 0 0	4000012	4000013	4.000	71000	Huth Formula	0.40	2.2
8	4000022	1 0 0	4000014	4000015	4.000	71000	Huth Formula	0.40	2.2
9	4000025	1 0 0	4000016	4000017	4.000	71000	Huth Formula	0.40	2.2
10	4000028	1 0 0	4000018	4000019	4.000	71000	Huth Formula	0.40	2.2

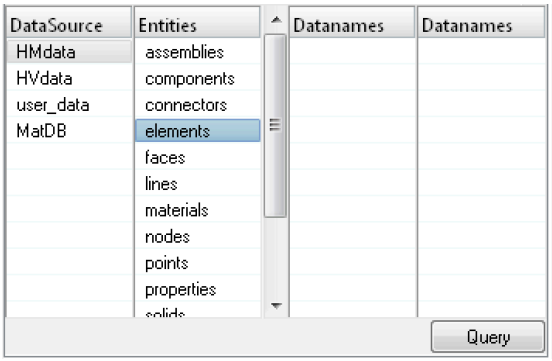
You can now use the **Matrix Browser** to Query the Fastener Stiffness.

11. Isolate the ***CBUSH*** elements created earlier.
12. In the Mask Browser expand the **Springs/Gaps** section.

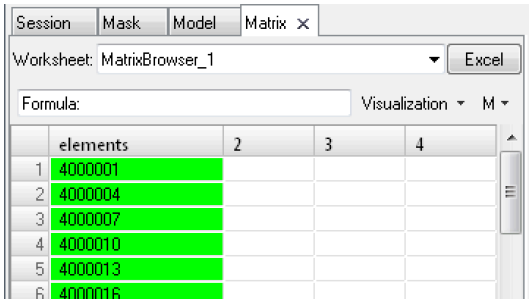


13. Left click on ***1*** to isolate spring elements.
14. Open the **Matrix Browser** using ***Tools > Matrix Browser.***

    A new tab opens with an upper and a lower section. In the lower one:
15. In the **DataSource** column select ***HMdata.***



16. In the **Entities** column, select ***elements.***
17. Click ***Query*** to query the entities.
18. With the selection panel open, select the displayed CBUSH elements (e.g. by window with SHIFT + LMB).
19. Click ***proceed*** and the first column populates with element IDs and is named elements.
20. Click on the ***elements*** columns' header to activate that column.

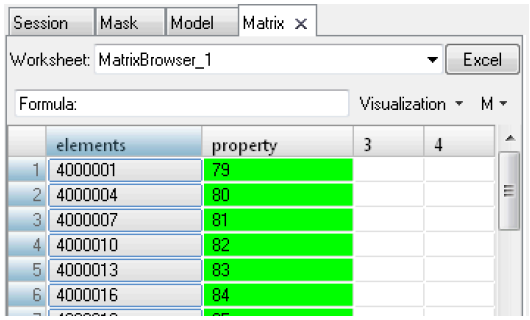


21. In the **Datanames** column, select ***property.***
22. Click ***Query***.

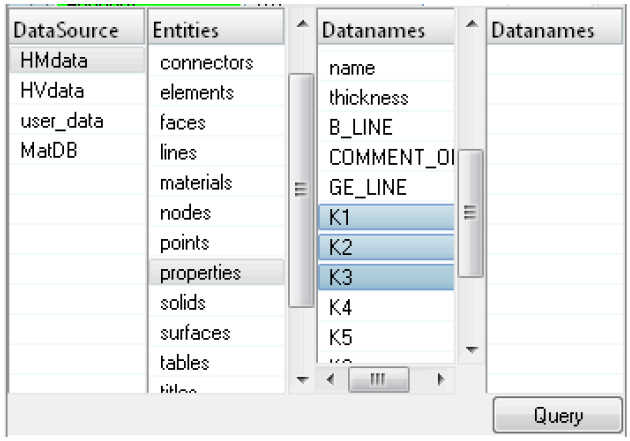
    A new column named property appears in the Matrix Browser.

    Note: The property id may be different than the IDs shown in the model browser, due to internal ID handling. To show the solver id that will export to input deck, select ***Datanames: Derived_Dataname > solverID > Query.***

23. Click the ***property*** column header.



24. In the ***Datanames*** column, select ***K1, K2,*** and ***K3***.

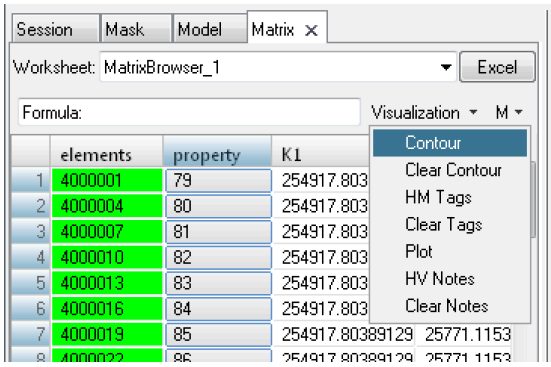


25. Click ***Query***.

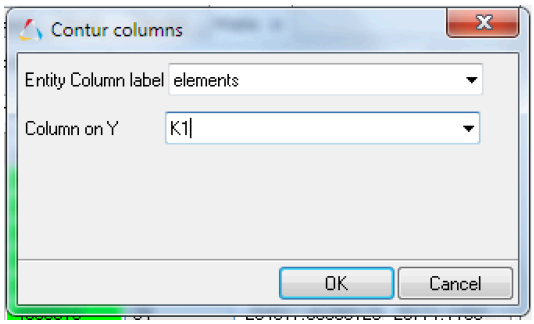
    Three new columns K1, K2, K3 appear in the matrix browser.

Step 10: Visualize the stiffness in the graphics area

1. Click ***Visualization > Contour*** from the menu bar of the **Matrix Browser**.

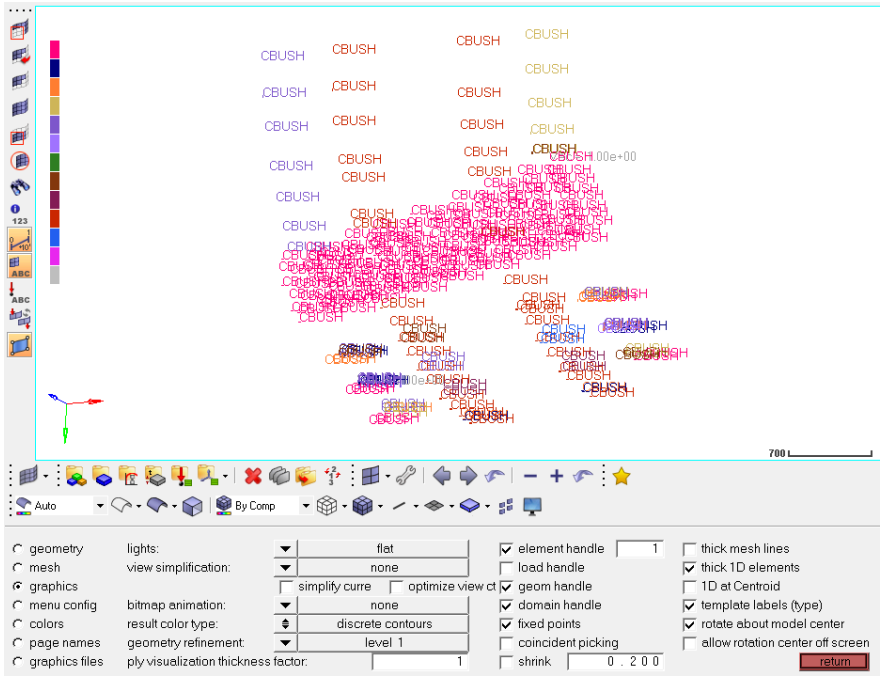


2. In the dialog, select ***elements*** in the **Entity Column label** field.



3. Set **Column on Y** to K1.
4. Click ***OK*** to complete the operation.

   The color of the CBUSH elements should turn to the corresponding value as in the legend.
5. Click ***Preferences > Graphics*** to open the panel.
6. Set **element handle** to 1 to see the element handles and with the color of the contouring for K1 better.



7. In the **Matrix Browser** tab select ***Visualization > Clear Contour*** to return to normal display.

Step 11: Save the file