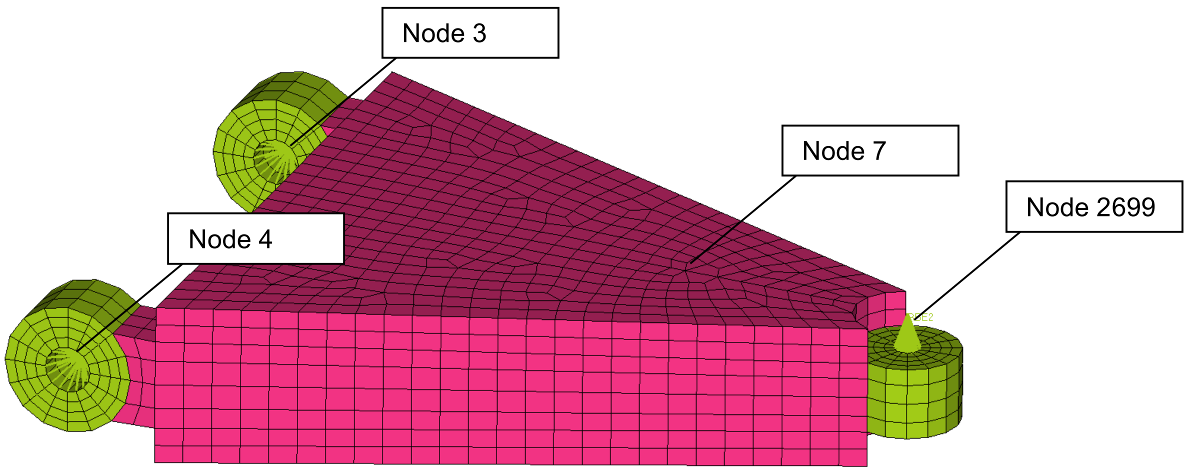
Exercise 1a: Topology Optimization of a Control Arm

The purpose of this exercise is to determine the basic minimum information required to run a topology optimization exercise.

The control arm can be considered totally fixed for all load cases as follows:

- NODE(3) ***X***,***Y*** and ***Z*** . (Bolted)
- NODE(4) ***Y*** and ***Z*** . (Cylindrical joint)
- NODE(7) ***Z***. (Damp link)

This control arm is set up with three different load cases.



Problem Setup

You should copy the file: `carm.fem`

Step 1: Import the model: `carm.fem` into HyperMesh Desktop

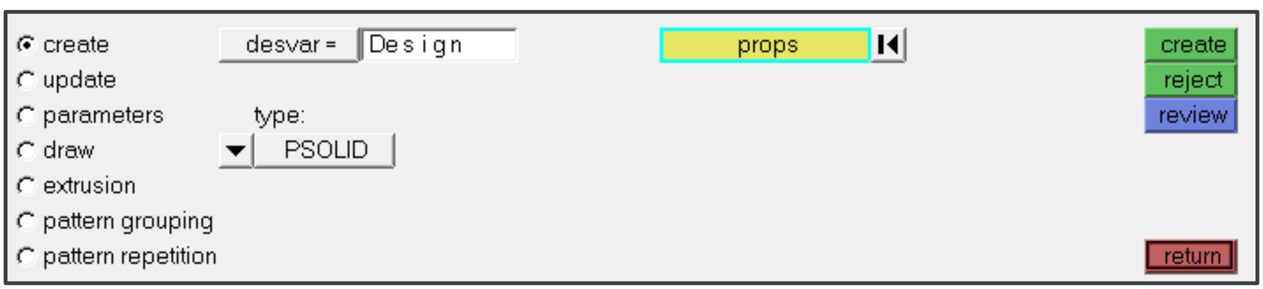
1. Select the ***Import*** button .

   An ***Import*** tab is added to your tab menu.

2. Make sure the **Import type:** is set to ***FE Model*** 
3. Make sure the **File type:** is set to ***Optistruct***.
4. Click the ***Select files***  button.
5. Browse for `carm.fem` and select it.
6. Click ***Open***.
7. Click ***Import***.

Step 2: Define a topology design variable for the design region

1. In the ***topology*** panel under the **optimization** section of the **Analysis** page, under ***desvar =***, enter the name `Design`.
2. Under the drop-down selector for ***type***, select `PSOLID`.
3. Click the ***props*** entity selector box and select the property `design`.
4. Click ***create*** to create the new design variable.
5. Click ***return*** to exit the panel.



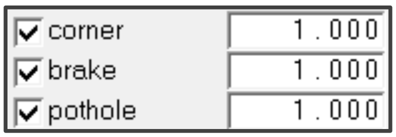
Step 3: Define a volume fraction response (total) named `volfrac`

1. In the ***responses*** panel under the **optimization** section of the **Analysis** page, under ***response =***, enter the name `volfrac`.
2. Under the drop-down selector for ***response type***, select `volume fraction`.
3. Set the ***type*** in the drop-down next to the response type as `total`.
4. Ensure that the ***regionid*** toggle is set to `no regionid`.
5. Click ***create*** to create the new response.

This creates a new response which will monitor the volume of the design space of the model at every iteration and output it as a normalized factor of the original volume of the design space. For more information on response types, please consult the HyperWorks Help entry on Response Types in OptiStruct.

Step 4: Define a weighted compliance response and name it `wcomp`.

1. In the ***responses*** panel, under ***response =***, enter the name `wcomp`.
2. Under the drop-down selector for response type, select `weighted compliance`.
3. Click the ***loadsteps*** selector to set the loadsteps and weights associated with this response.
4. Check all of the loadsteps and set the weights to `1.0` as shown below.



5. Click ***return*** to return to the response panel.
6. Click ***create*** to create the new response.
7. Click ***return*** to exit the panel.

   This creates a response which will sum the compliance of the model for each loadstep, each multiplied by its respective weight coefficient, for every iteration. For more information on response types, please consult the HyperWorks Help entry on Response Types in OptiStruct.

Step 5: In the constraints panel, use the volume fraction constraint to create an upper bound constraint.

1. In the ***dconstraints*** panel, under ***response =***, click on `volfrac`.
2. For ***constraint =***, enter the name `Cvolfrac`.
3. Click the checkbox next to ***upper bound*** and enter a value of `0.3`.
4. Click ***create*** to create the constraint.
5. Click ***return*** to exit the panel.

   This constrains the volume fraction to remain below 0.3 for the duration of the optimization. In the event that OptiStruct is unable to complete the objective while meeting this constraint, an error will be logged in the output file for the run.

Step 6: Create the objective function as type `min` and response `wcomp`

1. In the ***objective*** panel, set the drop-down selector to `min`.
2. For ***response =***, click on the response `wcomp`.
3. Click ***create*** to create the objective.

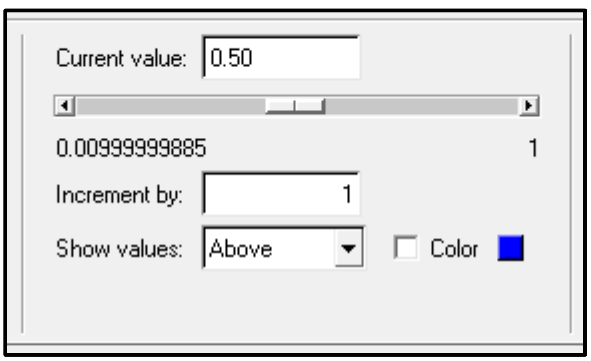
4. Click ***return*** to exit the panel.

Step 7: Submit the optimization run from the OptiStruct panel as `carm.fem`.

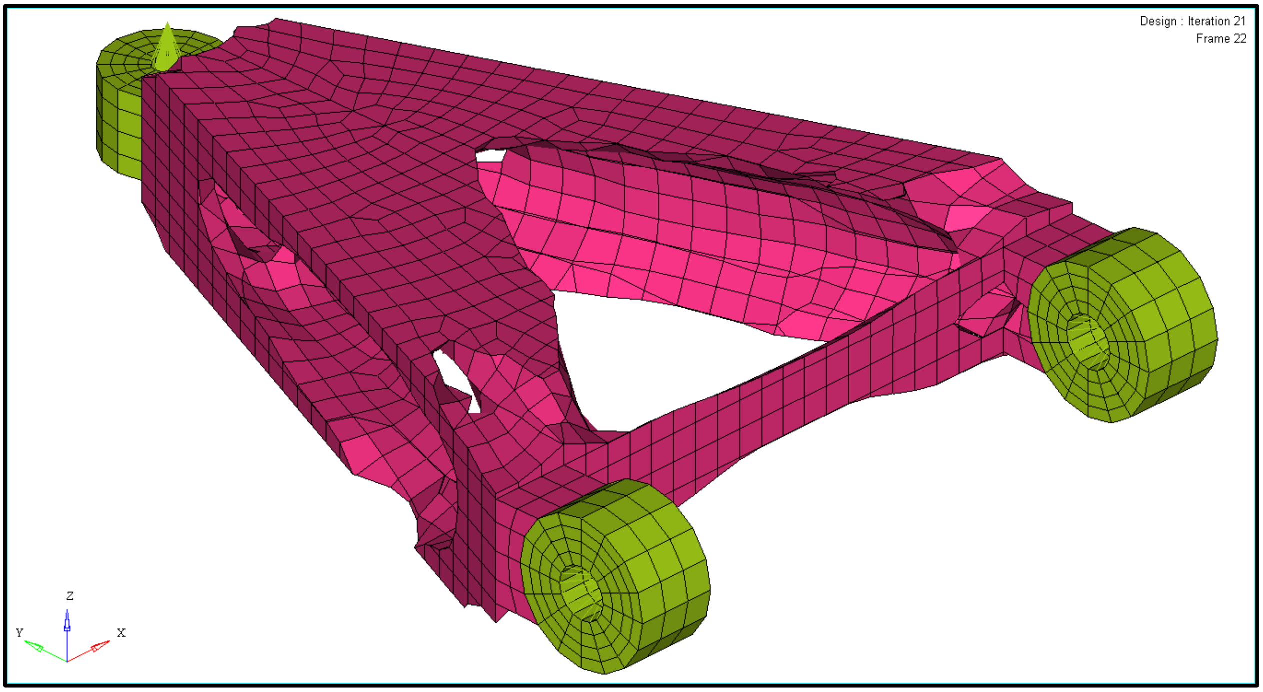
1. In the ***OptiStruct*** panel under the **Analysis** page, ensure that the ***export options:*** is set to `all`, the ***run options:*** is set to `optimization`.
2. Click ***OptiStruct*** to run the optimization, exporting the *.fem file with the default naming convention.

Step 8: Review the results in HyperView

1. When the run has completed, from the ***OptiStruct*** panel, click on the ***HyperView*** button to launch the post-processing report `carm.mvw` which was automatically created from the optimization.
2. Set the animation type to ***Transient***  and move the animation slider to the far right, viewing the final optimization iteration.
3. Proceed to the ***isosurface*** panel , and ensure the result type is set to `Element Densities (s).`
4. Click ***Apply*** to enable the isosurface view with a default setting.
5. Move the value slider, shown below, until the value is set to `0.5`, or type `0.5` into the ***Current value:*** field.



6. In the button bar immediately above the panel area, set the view controls to ***Shaded elements and mesh lines*** .



Design proposed by OptiStruct

Can you answer these questions:

1. Did the solution converge to a feasible solution?

How many iterations did the solution take to converge and what is the final volume of the part?

Plot the Iso-contour for the density on the last iteration, does it look like a manufacturable part?