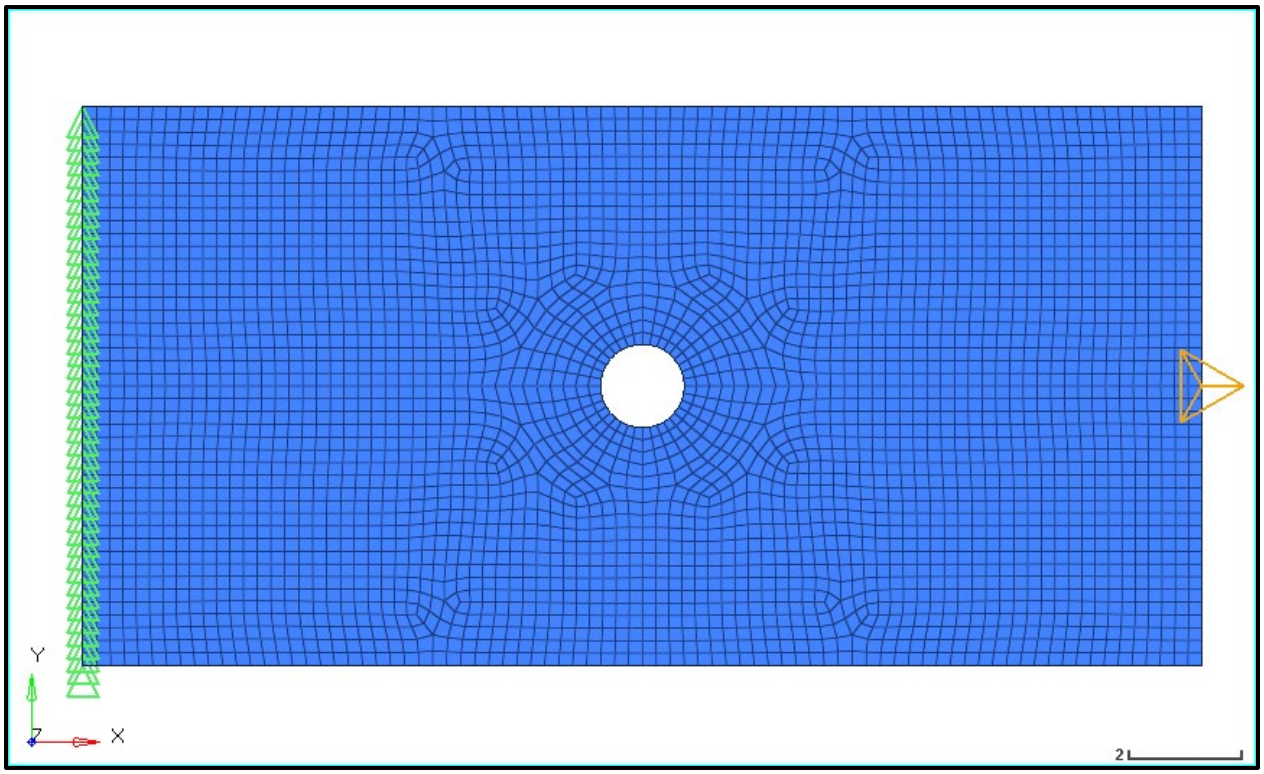
Exercise 6A: Composite Optimization of a Plate with Hole Coupon

This exercise introduces the user to the three-phase composite optimization-based design process through use of the previously prepared plate with hole composite analysis model. For the purposes of this optimization, a cantilevered loading analysis model is already set up and the user will focus on preparing the optimization entities needed to detail, design, and finish the model.



Problem Setup

You should copy the file: `plate_with_hole_opti_phase1.hm`

Phase 1: Free Size Optimization

Step 1: Open the model in HyperMesh Desktop with the OptiStruct user profile

Step 2: Create design variable for free sizing optimization

1. In the **Model Browser**, click the **Optimization View** button  .
2. Right-click in the **Model Browser** to access the context-sensitive menu and select the option ***Create > Free Size Desvar***.

Tip: A new design variable is created of type DSIZE and the new design variable properties are shown in the **Entity Editor**.

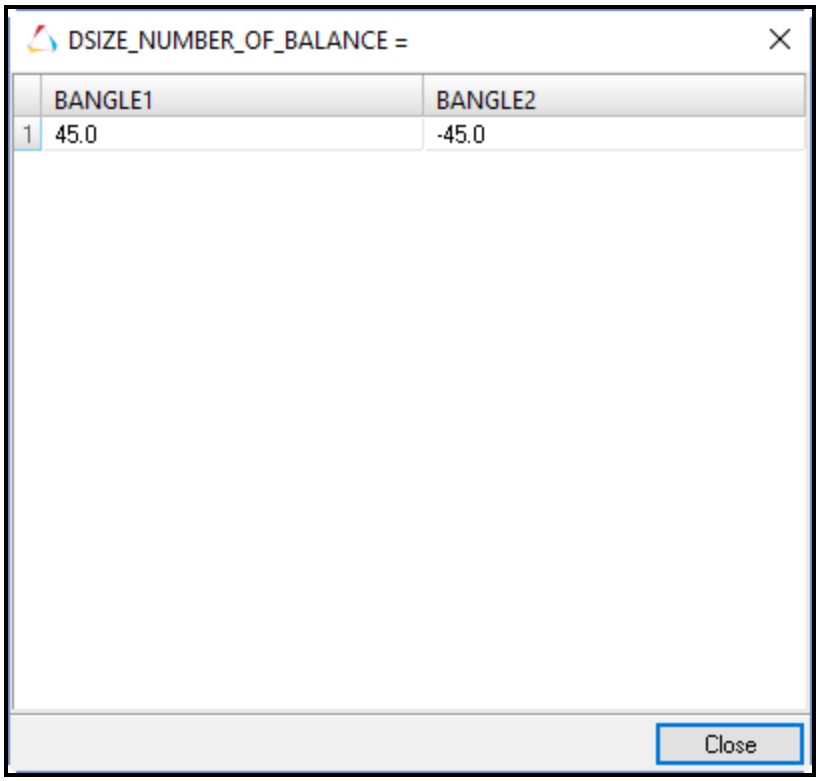
3. Name the new design variable `DSIZE`, set the ***Entity Type:*** to `STACK` and use the ***Property*** entity selector to choose the `Laminate` stack card as the design space.

Step 3: Add manufacturing constraints to the optimization

1. Click the `DSIZE` entry under **Design Variables** section of the **Model Browser Optimization View** to load that entry into the **Entity Editor**.
2. Enter a ***Minimum Laminate Thickness*** value of `0.05` and set the ***Maximum Laminate Thickness*** to `3`.
3. Under **Pattern Grouping**, select `1-pln sym` as the ***Pattern Type***.
4. Ensure that ***Anchor Point Selection*** and ***First Grid Selection*** are set to `node id`, and use the entity selectors to set the ***Anchor Point*** to node id `2111` and ***First Grid*** to `2220`.
5. Check the ***PLYMAN*** and ***BALANCE*** boxes.
6. Set ***PMMAN***, the manufacturable ply thickness, to `0.05`.

Tip: This defines that all plies will be a multiple of this value after this phase.

7. Click on the Table Data editor  under the BALANCE section and enter `-45.0` in the ***BANGLE1*** field and `45.0` for ***BANGLE2***.



Tip: The balance constraint requires plies of 45 or -45 to be in a pair, though not necessarily successive, through the laminate.

Step 4: Add responses to the free size optimization setup

1. Right-click in the **Model Browser Optimization View** to select ***Create > Responses***.
2. Name the new response `disp2111` and change the ***response type*** to `static displacement`.
3. Use the ***nodes*** entity selector to select node `2111` and change the displacement ***type*** from `dof1` to `total disp`.
4. Create another response of type `volume` named `volume` with the type option set to `total`.

Step 5: Create a constraint on the displacement response

1. Right-click in the **Model Browser** to select ***Create > Constraints***.

Tip: This allows users to create new constraints on responses for optimization.

2. Name the new response `Cdisp2111` and select ***response =*** `disp2111`.
3. Use the ***loadsteps*** entity selector to select the loadcase `Lateral`.

Tip: This will constrain the optimization algorithm to change the design if the displacement of node 2111 for the Lateral loadcase exceeds 0.5 units.

4. Enable the option for ***Upper Options*** and set and ***Upper Bound*** of `5e-1`.

Step 7: Create a minimize volume objective

1. Right-click in the **Model Browser** to create a new objective for the optimization.
2. Set the objective type to `Minimize` and select ***response =*** `volume`.

Step 8: Set output control cards to automatically output size optimization

1. Switch to the **Model View**  of the **Model Browser**.
2. In the menu, select ***Tools > Create Cards > OUTPUT***.

Tip: A new output card appears under the **Cards** section of the model tree.

3. Click on the new **OUTPUT** card and set the OUTPUT1 keyword to `FSTOSZ,` ***FREQ*** to `LAST,` & ***BUNDLES*** to `4`.

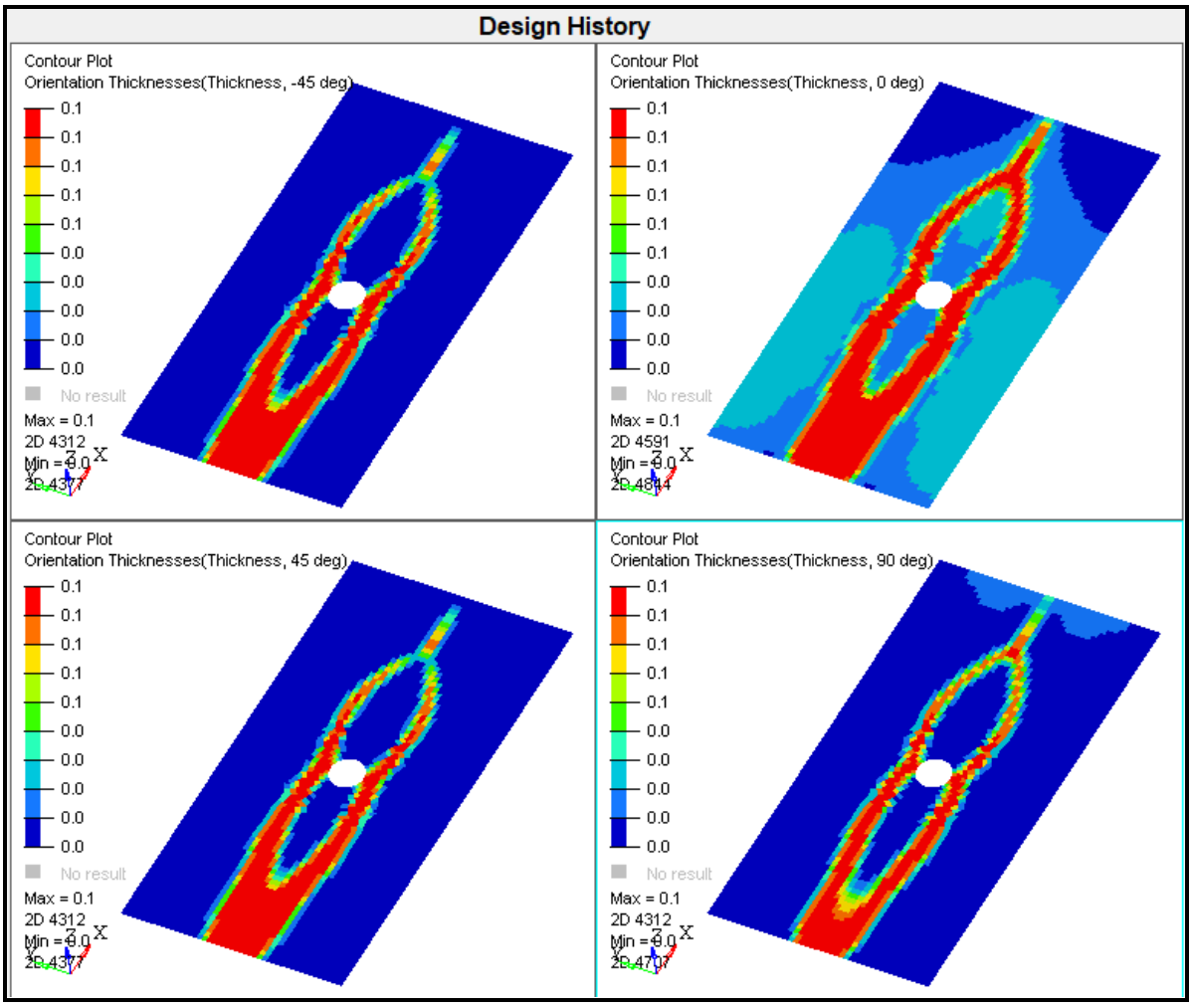
Step 9: Run the optimization

1. From the pull-down menu select ***Optimization > OptiStruct***.
2. Set ***export options***, ***run options*** & ***memory options*** to `all`, `optimization` & `memory default`.
3. Click ***OptiStruct***.

Step 10: Review the Free Size results

1. When the job is complete, load the optimization results into a new page with the **HyperView** client.
2. Use the **Contour** panel to plot the `Orientation Thickness` results per-ply for the final iteration of the design to determine how the optimization has distributed the material within each ply.

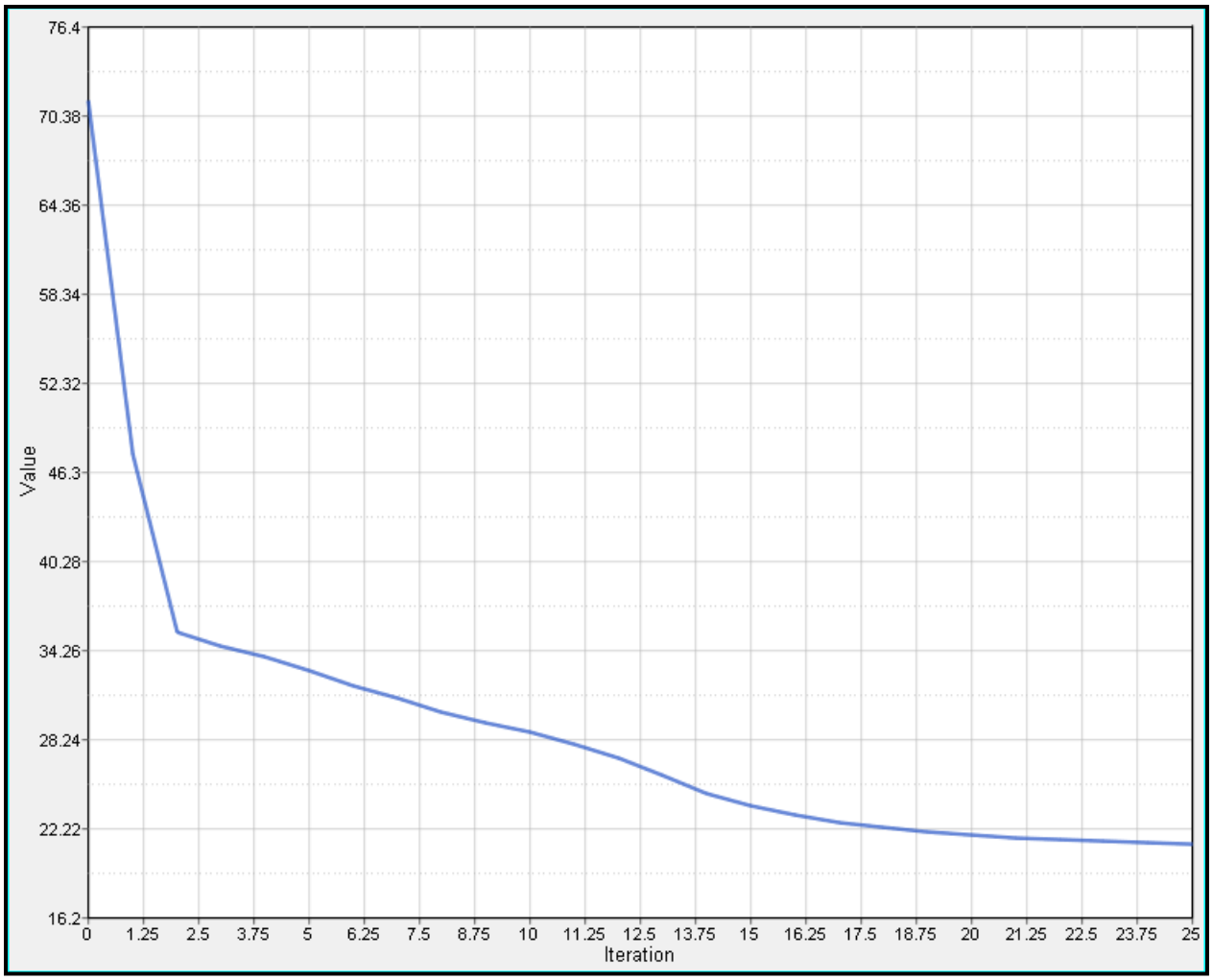
One important result that can be found graphically is understanding how the total element thickness is distributed over the topography of the laminate. Thickness changes can be displayed per-ply as shown below:



The breakdown of element thickness distribution

3. Users can also look at the optimization history to understand the design evolution by opening the *`_hist.mvw` file using **HyperGraph** and plotting the Constraint value and Objective Value against the Iteration:

Tip: These values are automatically prepared for any optimization and can be reviewed by opening the `plate_with_hole_opti_phase1_hist.mvw` file to display the optimization history.



A plot of the total volume of the plate at each successive iteration

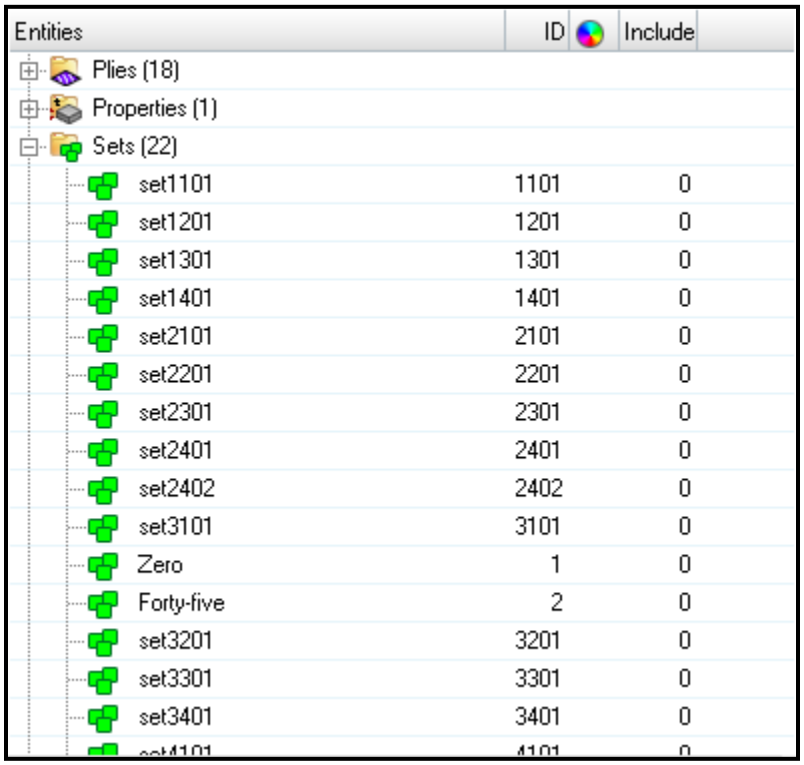
Phase II: Size Optimization

Step 11: Import the size optimization into a new session and review the ply bundles

1. Use the **Previous Page** button to navigate back to the **HyperMesh** session and begin a new session .

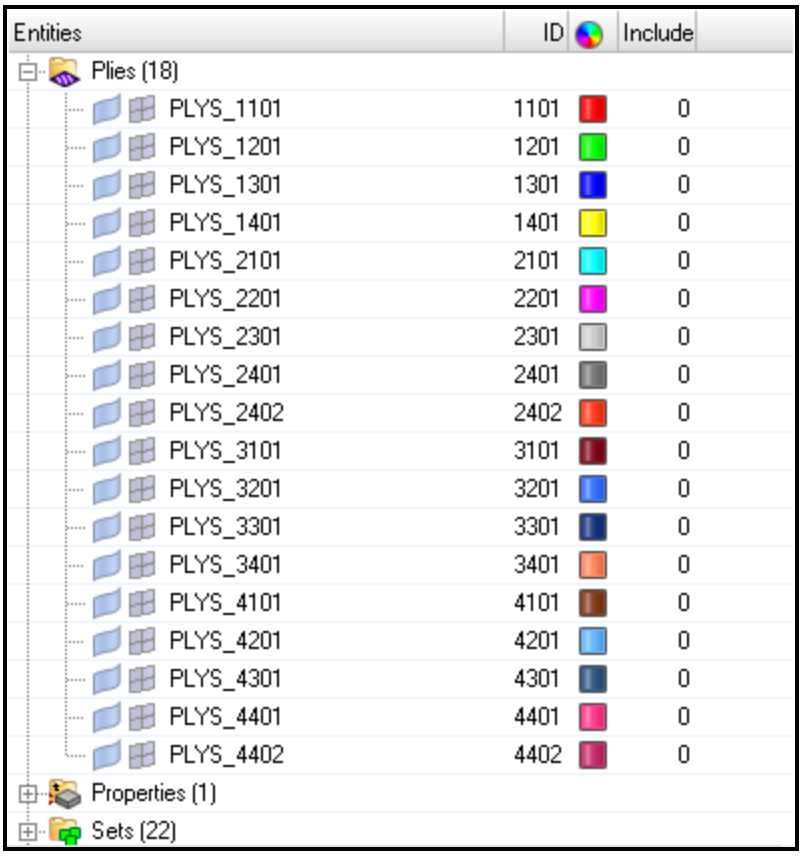
Tip: Beginning a new session from a HyperMesh client in **HyperMesh Desktop** will remove all but the first page and clear the current model from the **HyperMesh** database.

2. Locate and import the solver deck `plate_with_hole_opti_phase1_sizing.25.fem`.
3. Expand the **Sets** section of the **Model Browser**.

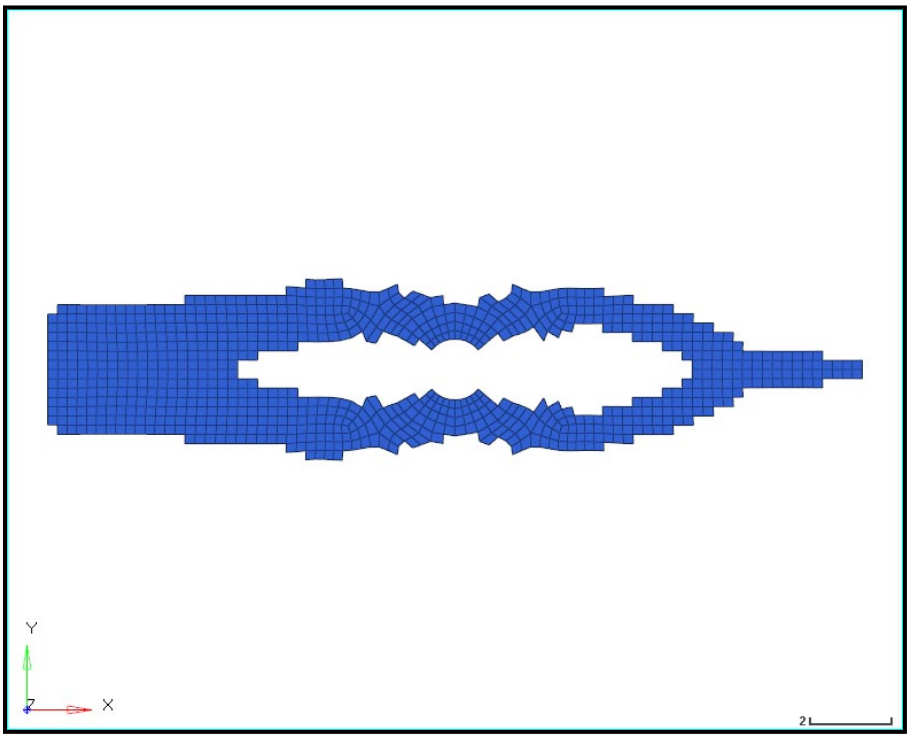


Tip: Twenty new sets are created: these are the sets automatically created from the ply bundles at the end of the Phase 1 optimization.

4. Right-click on the **Sets** section of the **Model Browser** and select ***Hide***.
5. One at a time, right-click on various sets in the list and select ***Isolate Only***. This will reveal the ply shapes associated with that set.
6. Expand the **Plies** section of the **Model Browser**. The original plies in the model have been replaced with plies automatically generated by the optimization output from the free-size optimization.

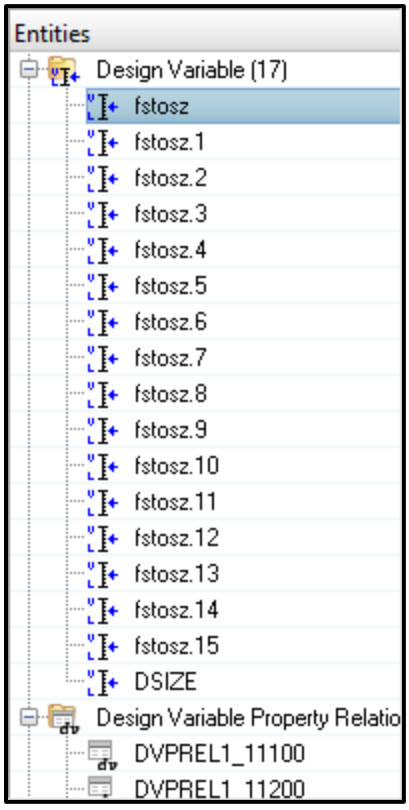


7. Review each **Ply** using the ***Isolate only*** feature to become familiar with the ply shapes associated with each orientation angle.

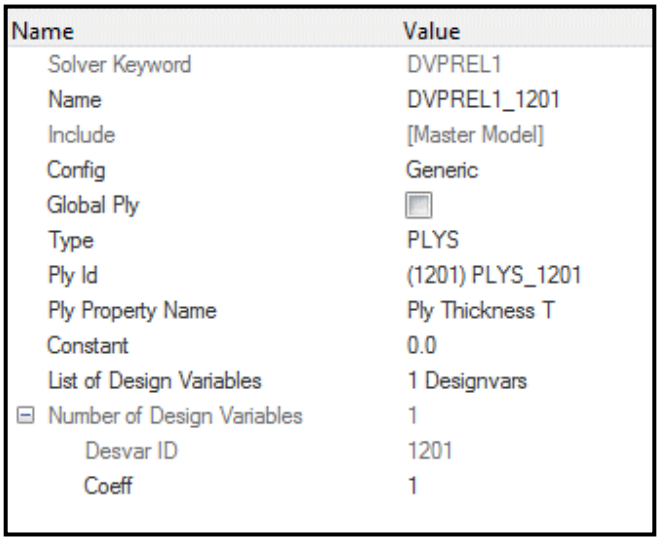


Step 12: Review and update the new optimization entities created by the automated FSTOSZ output

1. Expand the **Design Variables** section of the **Model Browser** to show the new design variables which were output by the FSTOSZ option. All of the design variables for size optimization are created and set with initial values and bounds for the ply thickness.
2. Update each of the ***Upper Bound*** values for the Design Variables to `0.1`.

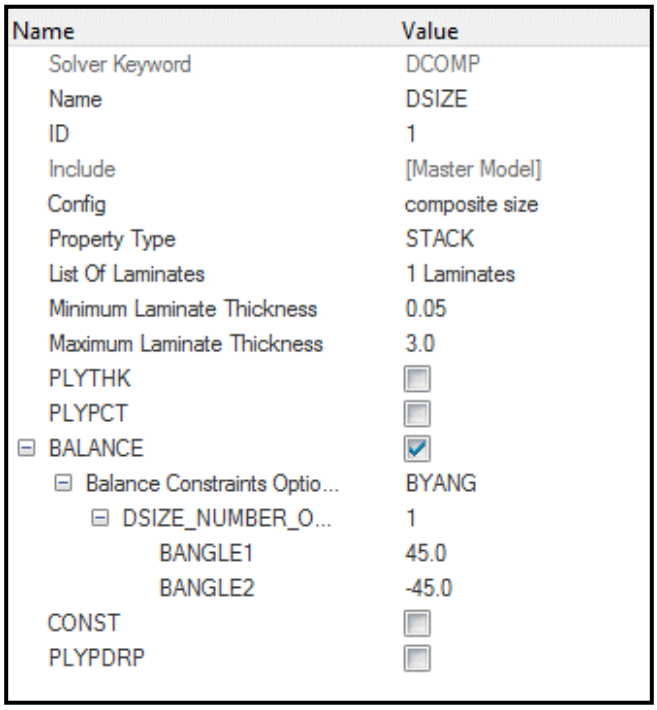


3. Expand the **Design Variable Property Relationship** section of the **Model Browser** and review some of the DVPREL cards in the **Entity Editor**.

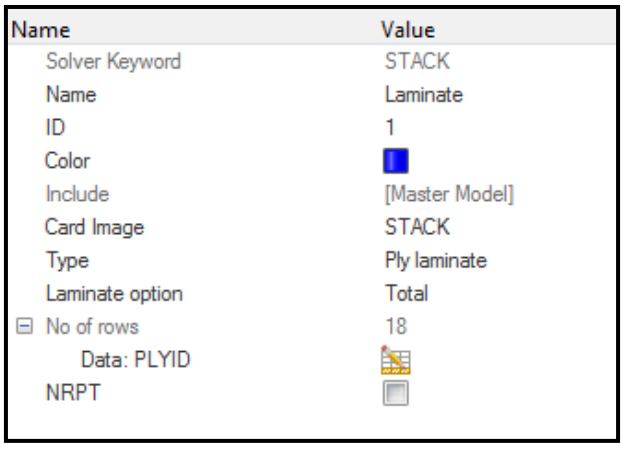


Step 13: Review the manufacturing constraints on the size optimization model

1. Switch to the **Model Browser Optimization View**.
2. Click on the DSIZE entry and review it, noting particularly that the laminate thickness and balance constraints have remained from the previous optimization while the Config (optimization type) has changed from Free Size optimization to Composite Size.



Step 14: In the Model View, change the Laminate option on the `Laminate` from `Smear` to `Total`



Step 15: Create a new composite stress response named `stress` which monitors all plies for `normal 1 stress` values

Name	Value
Solver Keyword	DRESP1
Name	stress
ID	3
Include	[Master Model]
Response Type	composite stress
Property	PCOMPP
List Of Properties	1 Properties
Exclude Elements	0 Elements
Region Identifier	
	normal 1
	all plies
DREPORT	☐

Step 16: Create a constraint on the `stress` response with a lower bound of `-25` and an upper bound of `25` for the `Lateral` loadstep

Name	Value
Solver Keyword	DCONSTR
Name	Cstress
ID	2
Include	[Master Model]
Response	(3) stress
List of Loadsteps	1 Loadsteps
Lower Options	
Lower Options	Lower bound
Lower Bound	-25.0
Upper Options	
Upper Options	Upper bound
Upper Bound	25.0
PROB	

Step 17: Update the control cards from FSTOSZ to SZTOSH

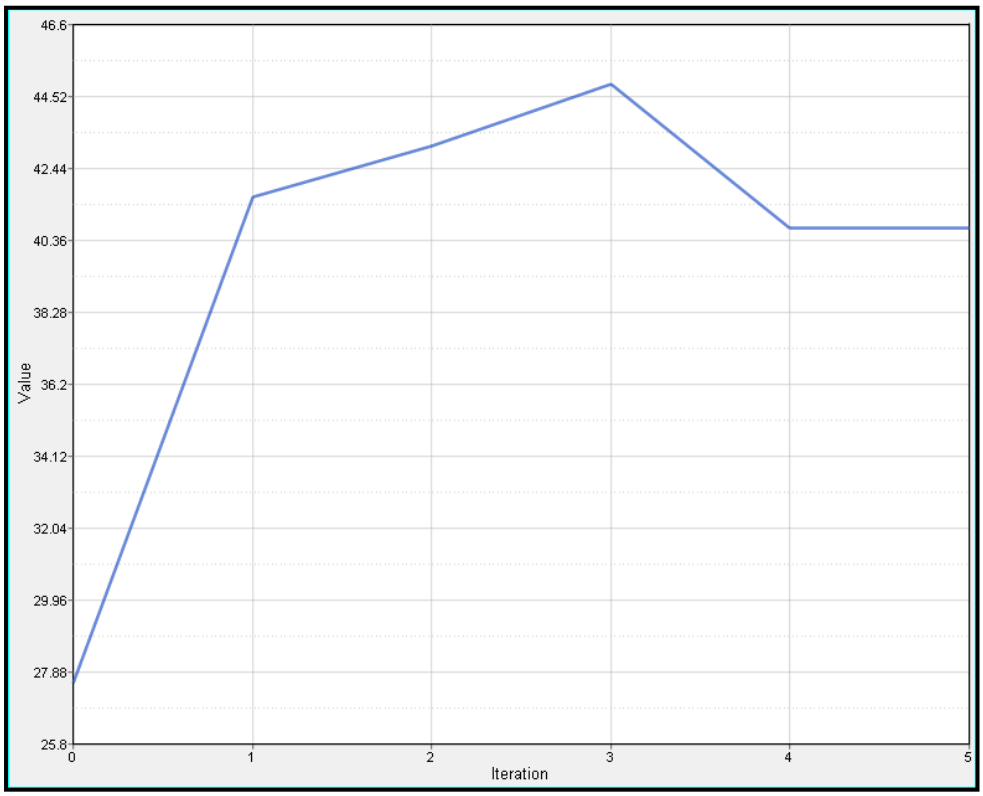
Name	Value
Include	[Master Model]
Status	☑
number_of_outputs =	1
OUTPUT 1	
KEYWORD	SZTOSH
FREQ	YES

Tip: This enables automated output of a composite shuffling optimization deck following the size optimization.

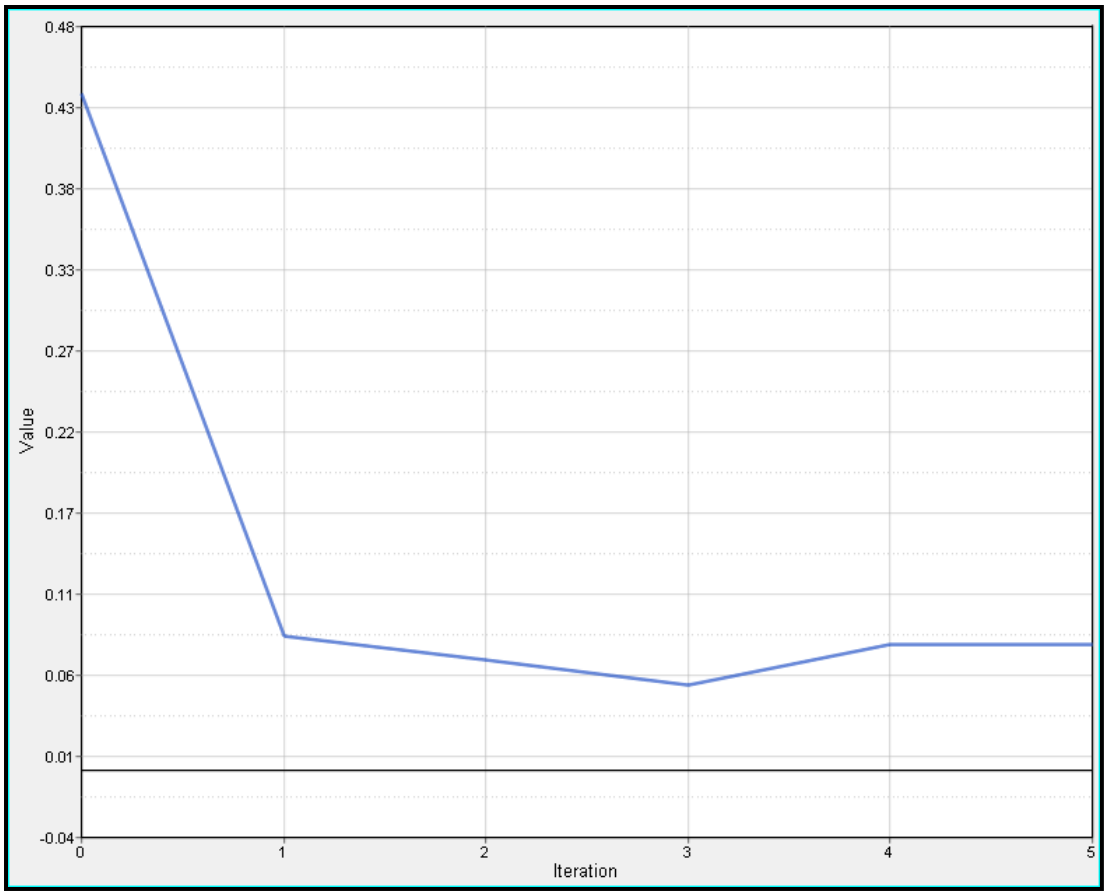
Step 18: Save the model as `plate_with_hole_opti_phase2.hm` and run the size optimization

Step 19: Review the size optimization results using the optimization history file

Tip: This can be done by opening the file `plate_with_hole_opti_phase2_hist.mvw` to view the relevant optimization drivers.



Measurement of the volume of the design space as a function of iteration



Maximum displacement of Node 2111 as a function of iteration

Tip: The thickness of each ply can be reviewed visually through **HyperView**, but can also be found in the *.out file (for each iteration) or within the *.prop file (for the final iteration).

```
818                    DESIGNED PROPERTY ITEMS TABLE
819   ------------------------------------------------------------------
820    DVPREL1/2  USER-ID  PROP-TYPE  PROP-ID  ITEM-CODE  PROP-VALUE
821   ------------------------------------------------------------------
822    DVPREL1       1101   PLY          1101   T          5.000E-02
823    DVPREL1       1201   PLY          1201   T          0.000E+00
824    DVPREL1       1301   PLY          1301   T          1.000E-01
825    DVPREL1       1401   PLY          1401   T          5.000E-02
826    DVPREL1       2101   PLY          2101   T          0.000E+00
827    DVPREL1       2201   PLY          2201   T          5.000E-02
828    DVPREL1       2301   PLY          2301   T          5.000E-02
829    DVPREL1       2401   PLY          2401   T          0.000E+00
830    DVPREL1       2402   PLY          2402   T          1.000E-01
831    DVPREL1       3101   PLY          3101   T          0.000E+00
832    DVPREL1       3201   PLY          3201   T          0.000E+00
833    DVPREL1       3301   PLY          3301   T          5.000E-02
834    DVPREL1       3401   PLY          3401   T          1.000E-01
```

Phase III: Shuffling optimization

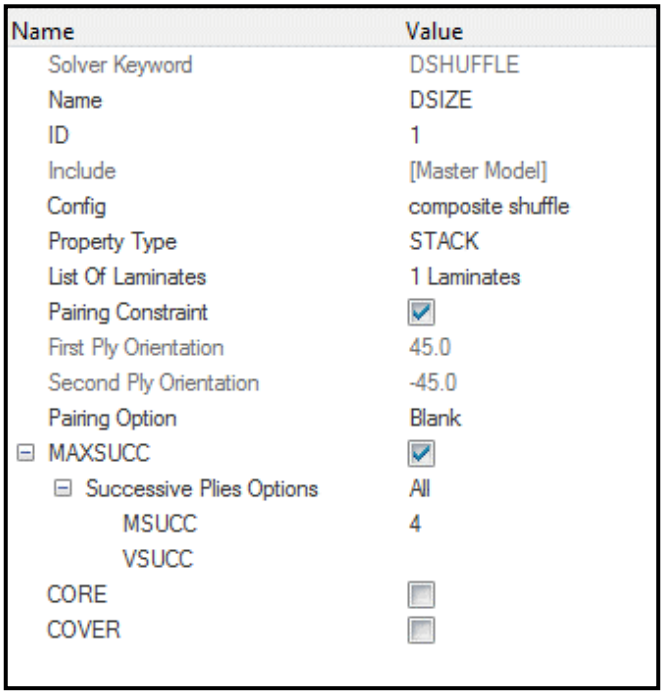
Step 20: Import the shuffling optimization file in a new HyperMesh session

Tip: The shuffling file automatically created by the size optimization solution will have a `*_shuffling.#.fem` suffix.

Step 21: Change the output control cards from SZTOSH to HTML

Step 22: Add ply book constraints to the shuffling optimization

1. In the **Optimization View**, click on the `DSIZE` design variable entry.
2. Update the design variable to add pairing constraints and maximum successive ply constraints as follows:



Step 23: Save the model as `plate_with_hole_opti_phase3.hm`

Step 24: Run the shuffling optimization

Step 25: Review the results using the `*.prop` file generated