Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-1540: Shape Optimization Study Using HyperMesh and ANSYS

HS-1540: Shape Optimization Study Using HyperMesh and ANSYS

In this tutorial, you will learn how to pshape parameterization using HyperStudy.

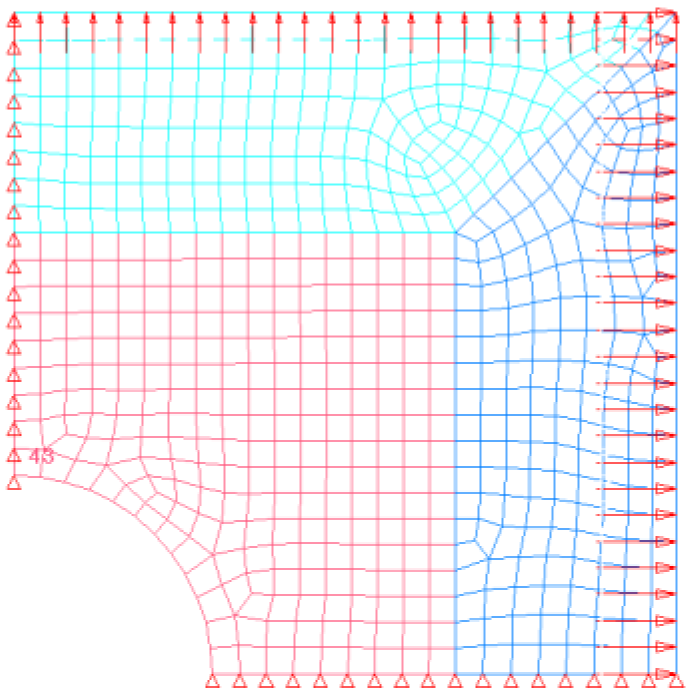
- erform a shape optimization started from inside HyperMesh using the direct link to HyperStudy. The finite element solver is Ansys. HyperMorph is used to do the shape parameterization. The objective is to minimize the maximum stress of a plate with a hole. The solution can be expected to be some kind of ellipse. Hence, the input variables are the half-axes of the hole.

By the end of this tutorial, you will know how to:

- Do a Run a study from inside HyperMesh.
- Set-up a study.
- Run an optimization study.

The files used in this tutorial can be found in `<hst.zip>/HS-1540/`. Copy the tutorial files from this directory to your working directory. The tutorial directory includes the following files:

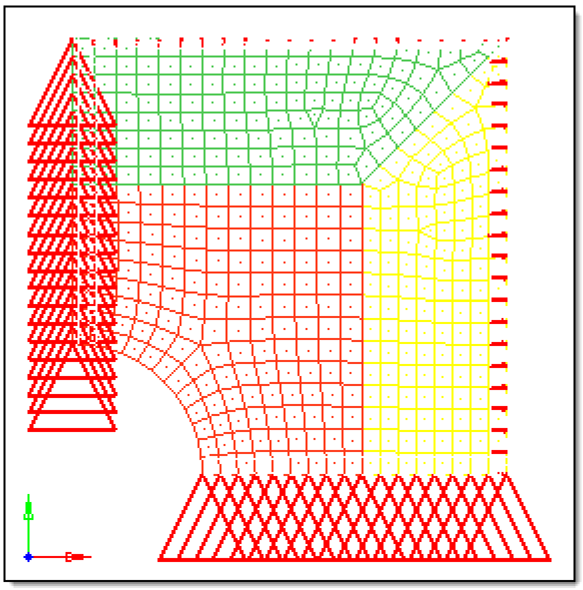
`plate.cdb` -	Original Ansys input file.
`ansys.bat` -	Sample execution script for the educational version of Ansys.



Double symmetric plate model.

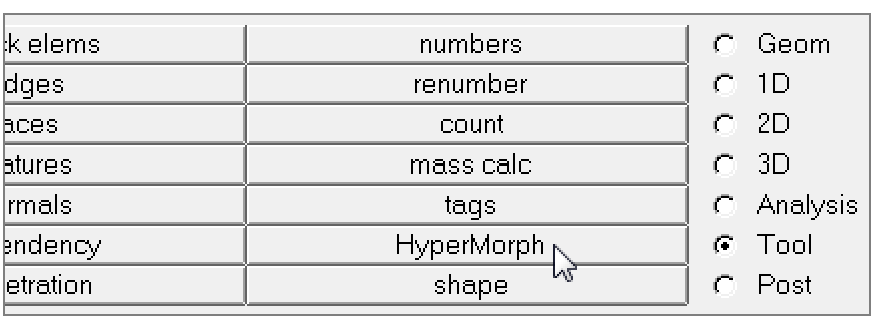
Step 1: Setup the Model in HyperMesh

1. Start HyperMesh Desktop.
2. In the **User Profiles** dialog, set the user profile to ***Ansys***.
3. From the menu bar, click ***File*** > ***Import*** > ***Solver Deck***.
4. Set **File type** to ***Ansys***.
5. In the **File** field, open the `plate.cdb` file.
6. Click ***Import***. A finite element model appears in the graphics area.



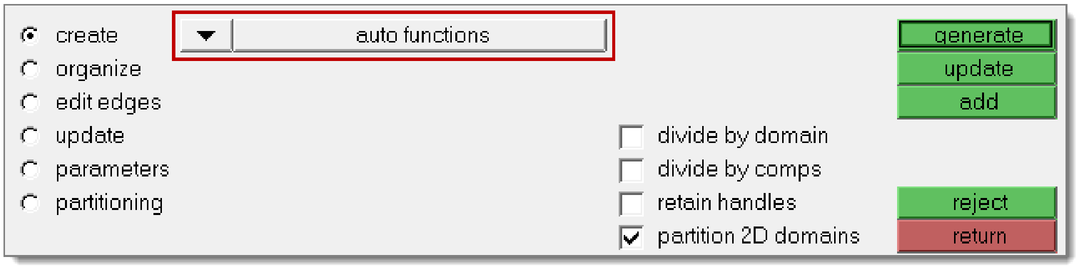
Step 2: Do the Shape Parameterization in HyperMorph

1. From the **Tool** page, click **HyperMorph**.

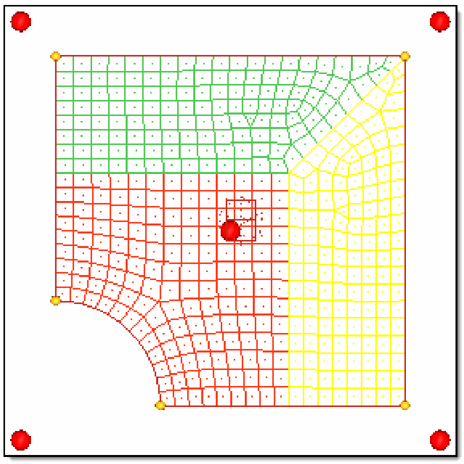


2. Click ***domains***.
3. Go to the ***create*** subpanel

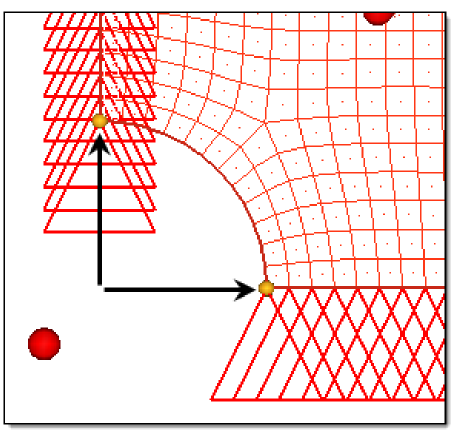
4. Click the first arrow and select ***auto functions***.



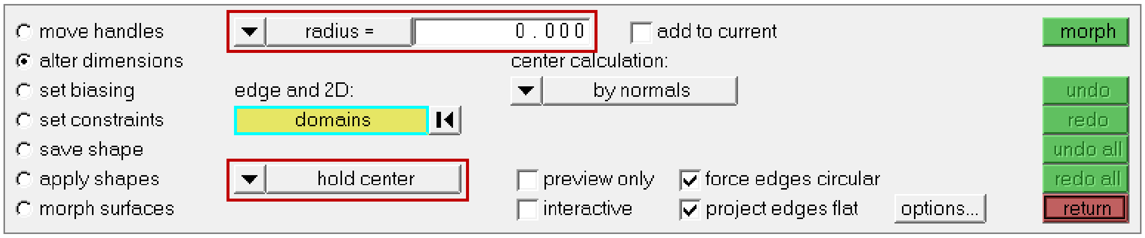
5. Click ***generate***. HyperMesh generates the domains and handles that you will use to manipulate the shape of the mesh and to generate shape perturbations for shape optimization.



6. Click ***return***.
7. In the panel area, click ***morph***.
8. Go to the ***set biasing*** subpanel.
9. In the graphics area, click the two yellow handles located in the corner of the quarter circle.

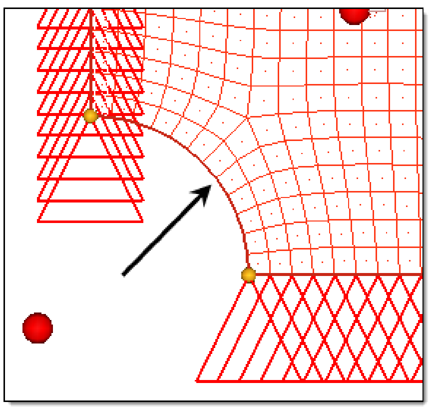


10. In the **bias=** field, enter `2.000`.
11. Click ***update***.
12. Go to the ***alter dimensions*** subpanel.
13. Define the radius as a shape by clicking the first arrow and selecting ***radius***.
14. Click the bottom arrow, and select ***hold center***.

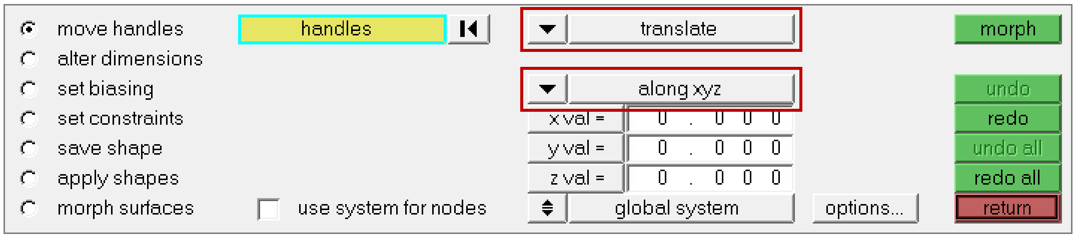


Settings for steps 2.13 and 2.14.

15. In the graphics area, click the red edge of the hole.

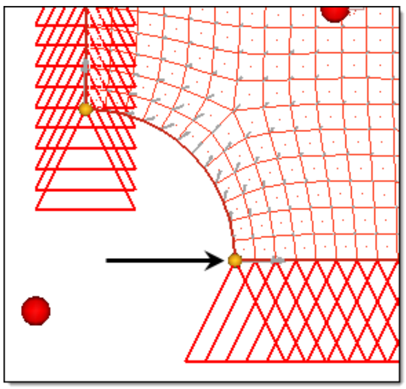


16. In the **radius=** field, enter `20.0000`.
17. Click ***morph***. The first shape is generated.
18. Go to the ***save shape*** subpanel.
19. In the **name=** field, enter `sh1`.
20. Click ***save***.
21. Click ***undo*** to prepare for the generation of the next shape.
22. Go to the ***move handles*** subpanel.
23. Click the second arrow and select ***translate***.
24. Click the third arrow and select ***along xyz***.

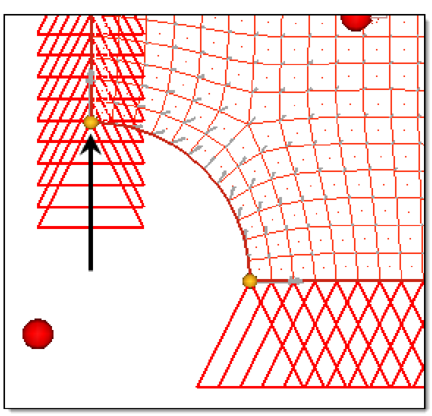


Settings for steps 2.23 and 2.24.

25. In the graphics area, click the lower yellow handle located in the corner of the quarter circle.



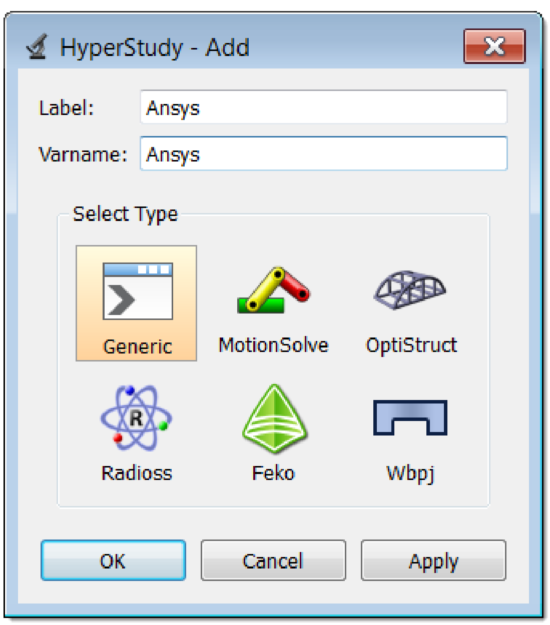
26. In the **x val=** field, enter `10.0000`.
27. Click ***morph***. The second shape is generated.
28. Go to the ***save shape*** subpanel.
29. In the **name=** field, enter `sh2`.
30. Click ***save***.
31. Click ***undo*** to prepare for the generation of the next shape.
32. Go to the ***move handles*** subpanel.
33. In the graphics area, click the upper yellow handle located in the corner of the quarter circle.



34. In the **x val=** field, enter `0.000`.
35. In the **y val=** field, enter `10.0000`.
36. Click ***morph***. The third shape is generated.
37. Go to the ***save shape*** subpanel.
38. In the **name=** field, enter `sh3`.
39. Click ***save***.
40. Click ***undo*** to restore the initial mesh.
41. Save the HyperMesh model by clicking ***File*** > ***Save As*** > ***Model*** from the menu bar.
42. In the **Save Model As** dialog, save the file as `plateDV.hm`.
43. Close HyperMesh Desktop.

Step 3: Register ANSYS as a Solver

1. Start HyperStudy.
2. From the menu bar, click ***Edit*** > ***Register Solver Script***.
3. In the **Register Solver Script** dialog, click ***Add Solver Script***.
4. In the **Add - HyperStudy** dialog, enter `Ansys` in the **Label** and **Varname** fields.
5. From the list of solver script types, select ***Generic***.
6. Click ***OK***.



7. In the **Path** column of the script **Ansys**, click .
8. In the **Open** dialog, open the `ansys.bat` file.

Note: The script `ansys.bat` is a sample of an execution script for Ansys on Windows. Copy `ansys.bat` to your working directory to use it.

9. Click ***Save***.
10. In the **Save Preferences** dialog, navigate to your working directory.

    Note: On UNIX, the preference file can also be saved in your home directory or in the working directory from which you launched HyperStudy.

11. In the **File name** field, enter a name for the new user preference file (example: `userprefs.mvw`).
12. Click ***Save***.

    Note: Do not overwrite the system preferences file, which is located in <install_directory>/hw by default.

13. Click ***Close***.

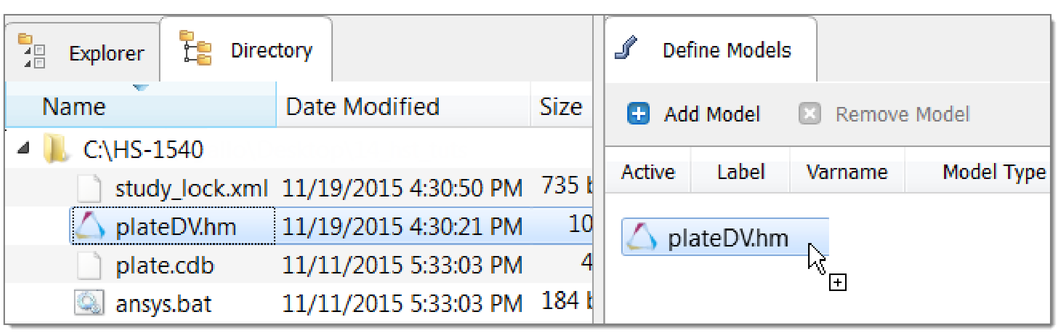
    Note: When you start a new HyperStudy session, you can load your preference file by clicking ***File*** > ***Set Preference File*** from the menu bar, and then selecting the preference file you just saved. HyperStudy reads the default preference file in the installation directory, followed by the preference file that you specify. This will ensure that all solvers, readers and import templates are available.

    You can also append the current user preference file using **Append**, or you can exit solver registration using **Close**. In the last case, the solver will only be registered for the current study.

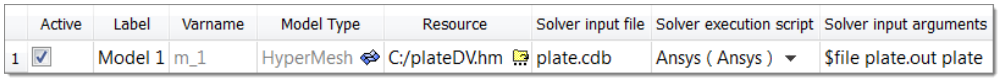
Step 4: Perform the Study Setup

1. To start a new study, click ***File*** > ***New*** from the menu bar, or click  on the toolbar.
2. In the **HyperStudy – Add** dialog, enter a study name, select a location for the study, and click ***OK***.
3. Go to the **Define models** step.
4. Add a HyperMesh model.

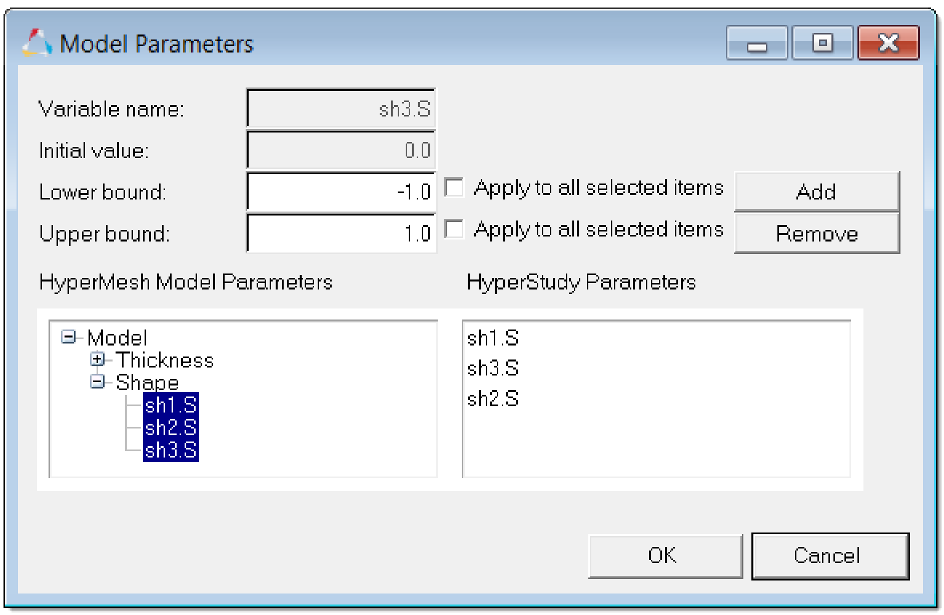
    a. From the **Directory**, drag-and-drop the HyperMesh (`.hm`) file `plateDV.hm` into the work area.



   b. In the **Solver input file** column, enter `plate.cdb`. This is the name of the solver input file HyperStudy writes during any evaluation.
   c. In the **Solver execution script** column, select ***Ansys***.
   d. In the **Solver input arguments** column, enter `plate.out plate` after `$file`.



5. Click ***Import Variables***.
6. In the **Model Parameters** dialog, select parameters to import into HyperStudy.
   a. Expand ***Shape***, and select ***sh1.S***, ***sh2.S***, and ***sh3.S***.
   b. Verify that the **Lower bound** is `-1.0` and the **Upper bound** is `1.0`.
   c. Click ***Add***.
   d. Click ***OK***.



7. Go to the **Define Input Variables** step.
8. Review the input variable's lower, initial, and upper bounds.
9. Go to the **Specifications** step.

Step 5: Perform the Nominal Run

1. In the work area, set the **Mode** to ***Nominal Run***.
2. Click ***Apply***.
3. Go to the **Evaluate** step.

4. Click ***Evaluate Tasks***.
5. Go to the **Define Output Responses** step.

Step 6: Create and Define Output Responses

1. From the **Directory**, drag-and-drop the `plate.rst` file, located in the `approaches/nom_1/run__00001/m_1` directory, into the work area.
2. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks®*** and click ***Next***.
3. Select ***Multiple items at multiple time steps (readsim)***, then click ***Next***.
4. Define the following options, and then click ***Next***.
    a. Set **Subcase** to ***Step 1***.
    b. Set **Type** to ***Stress (2D)***.
    c. Set **Request** to ***E132*** - ***E423***.
    d. Set **Component** to ***vonMises***.
5. Optional. Enter labels for the data source and output response.
6. Set **Expression** to ***Maximum***.
7. Click ***Finish***. Output response 1 is added to the work area.
8. Click ***Evaluate Expressions***.

Step 7: Run an Optimization Study

1. In the **Explorer**, right-click and select ***Add*** from the context menu.
2. In the **Add - HyperStudy** dialog, select ***Optimization*** and click ***OK***.
3. Go to the **Select Input Variables** step.
4. Review the input variable's lower and upper bound ranges.
5. Go to the **Select Output Responses** step.
6. Click ***Add Objective***.
7. In the **Add - HyperStudy** dialog, add one objective.
8. Define the objective.
    a. Set **Type** to ***Minimize***.
    b. Set **Apply On** to ***Response 1 (r_1)***.

	Active	Label	Varname	Objectives	Constraints	Evaluate From	Expression	Comment
1	☑	Response 1	m_1_r_1	Minimize ...		Solver	m_1_ds_1[0]	Data Source 1 ...

9. Click ***Apply***.
10. Go to the **Specifications** step.
11. In the work area, set the **Mode** to ***Adaptive Response Surface Method (ARSM)***.

    Note: Only the methods that are valid for the problem formulation are enabled.
12. Click ***Apply***.
13. Go to the **Evaluate** step.
14. Click ***Evaluate Tasks***.

Step 8: View the Iteration History of an Optimization Study

1. Click the ***Iteration History*** tab to review the Optimization results.

   Note: The optimal design is highlighted in green.
2. Click the ***Iteration Plot*** tab to plot the optimization results.