Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-1535: Coupling HyperStudy, Morphing and AcuSolve
(HyperStudy Job Launcher)

HS-1535: Coupling HyperStudy, Morphing and AcuSolve (HyperStudy Job Launcher)

In this tutorial, you will learn how to perform a DOE study using HyperStudy and the HyperStudy Job Launcher within HyperMesh. HyperMorph is used to parameterize the shape of the design. The subject of the study is to analyze sensitivity of flow to changes in the shape (bending) of a pipe.

After performing the baseline simulation, a DOE study will be executed to analyze the effect of changes in pipe shape on the pressure drop between inlet and outlet. This is one of the many types of studies that can be done using AcuSolve with HyperMesh and HyperStudy.

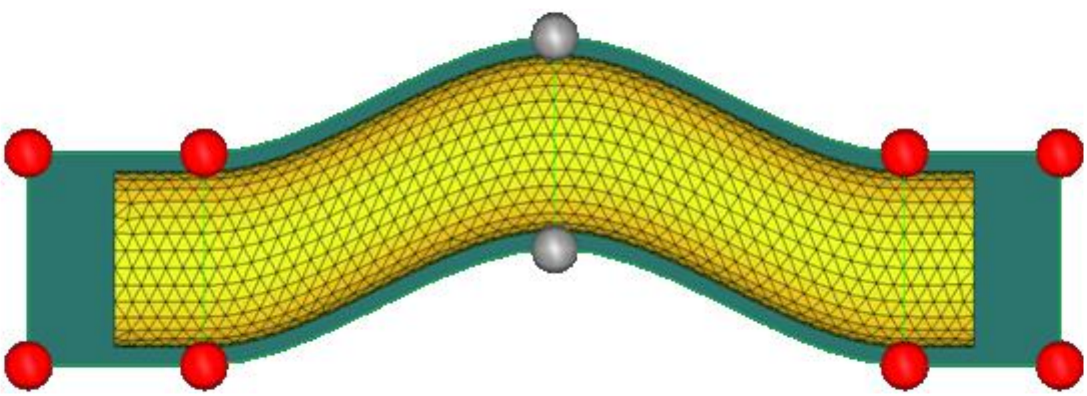
A DOE or optimization study starts from a baseline model. This would be a model that has already been simulated with AcuSolve. For completeness, this tutorial also describes typical steps followed during the initial or baseline AcuSolve simulation. To this end, we use the file `pipe.hm`, a mesh created in HyperMesh.

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

- Parameterize the model using HyperMorph and HyperStudy
- Use the HyperStudy Job Launcher to couple AcuSolve and HyperStudy
- Set up and run a DOE study

The files used in this tutorial can be found in `<hst.zip>/HS-1535/`. Copy the tutorial files from this directory to your working directory. The tutorial directory includes the following files:

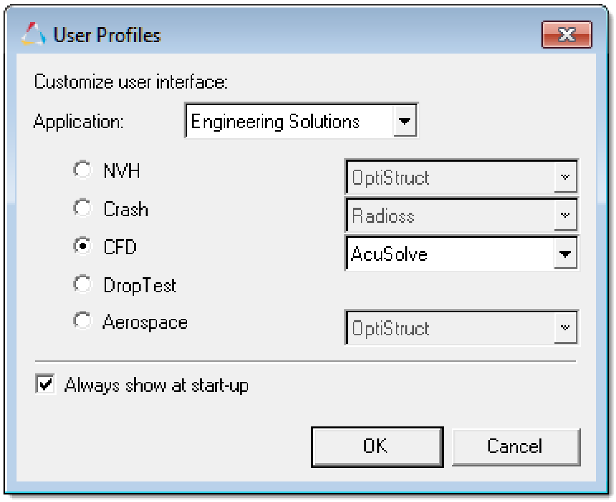
- `pipe.hm` - HyperMesh model of the pipe.
- `run_acusolve.bat` - Customizable execution script for AcuSolve (Windows). The batch file needs to be adapted to the current directory structure.



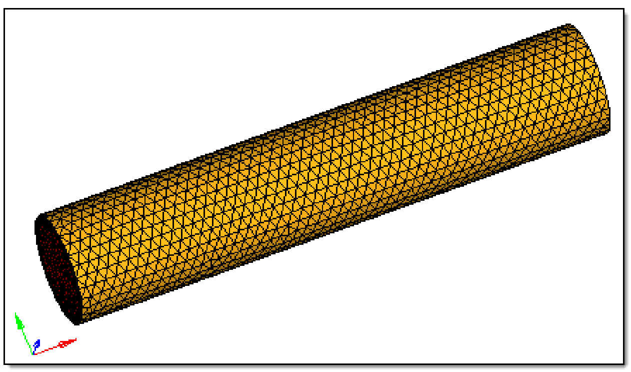
Pipe model with handles for the shapes

Step 1: Load the Model in HyperMesh Desktop

1. Start HyperMesh Desktop.
2. In the **User Profiles** dialog, set the user profile to ***Engineering Solutions***, ***CFD***, ***AcuSolve***.

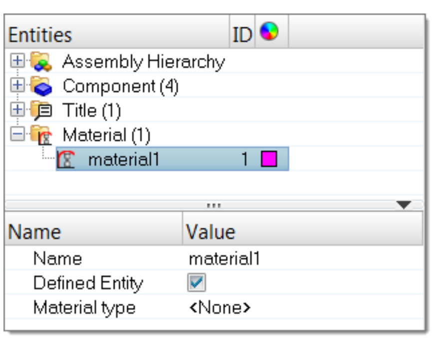


3. From the menu bar, click ***File*** > ***Open*** > ***Model***.
4. In the **Open Model** dialog, open the `pipe.hm` file. A finite element model appears in the graphics area.

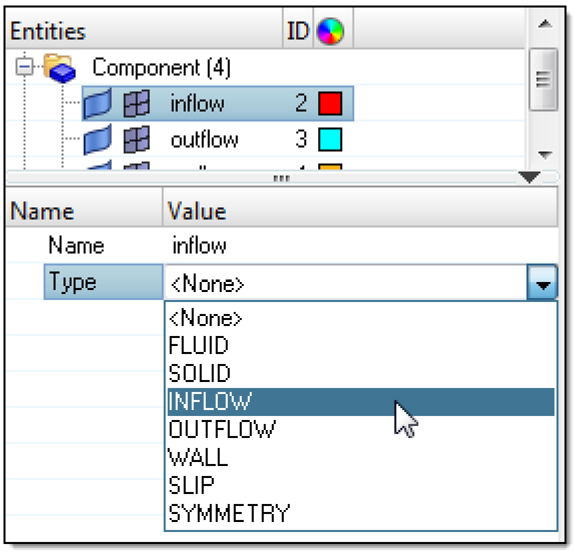


Step 2: AcuSolve Case Setup in HyperMesh

1. In the **Model** browser, right-click and select ***Create*** > ***Material*** from the context menu. A new material opens in the **Entity Editor**.



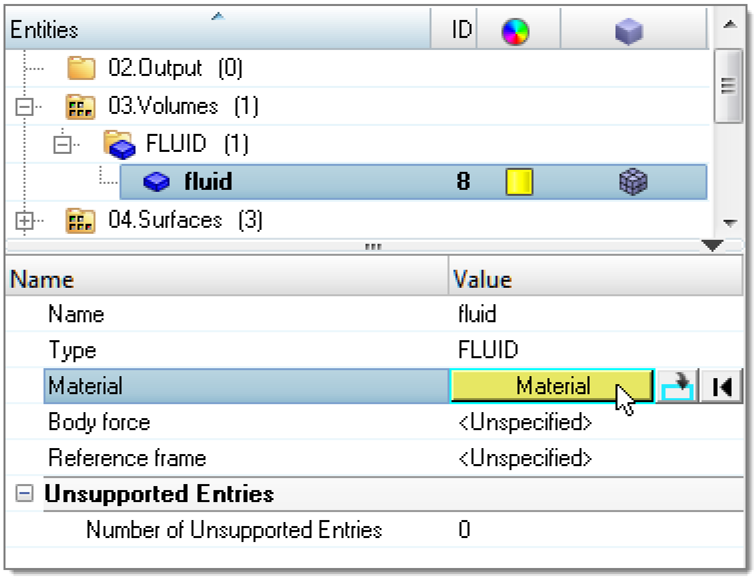
2. In the **Entity Editor**:
   a. For **Name**, enter `Air_HM`.
   b. Set **Card Image** to ***FLUID***.
3. Create a second material named **Water_HM**, and set the card image to **FLUID**.
4. In the **Model** browser, **Component** folder, click ***inflow***. The **Entity Editor** opens and displays the component's corresponding data.
5. For **Type**, select ***INFLOW***.



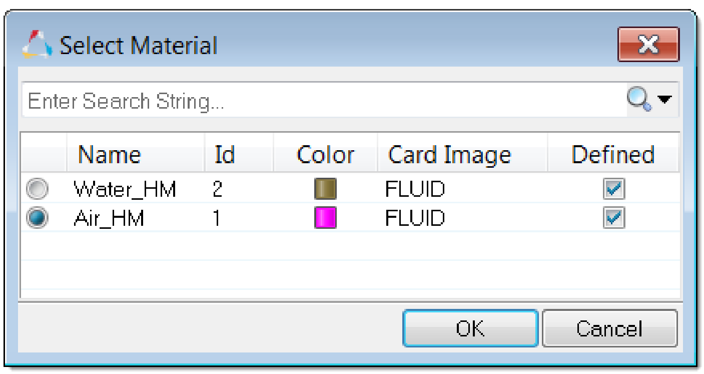
6. Change the **Type** for the following components:

Component	Type
outflow	OUTFLOW
wall	WALL
fluid	FLUID

7. In the **Model** browser, **Components** folder, click ***fluid***. The **Entity Editor** opens and display's the component's corresponding data.
8. For **Material**, click ***Unspecified*** >> ***Material***.



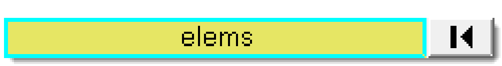
9. In the **Select Material** dialog, select ***Air_HM*** and then click ***OK***.



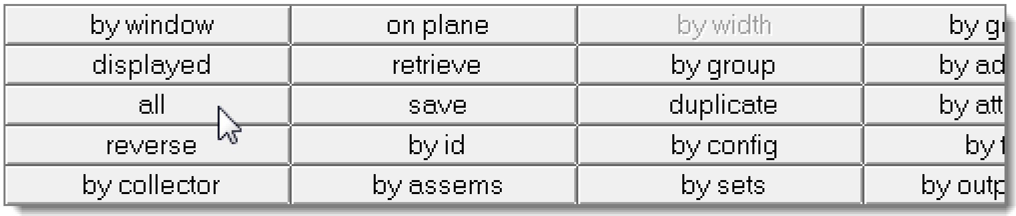
10. In the **Model** browser, **Components** folder, click ***inflow***. The **Entity Editor** opens and display's the component's corresponding data.
11. For **Inflow type**, select ***Mass flux***.
12. For **Mass flux ## (kg/sec)**, enter `0.0003`.

Step 3: Morphing

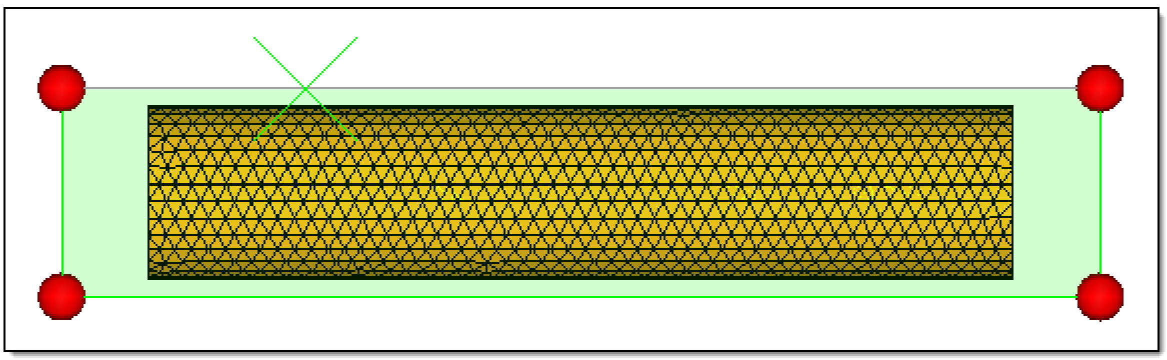
1. In the panel area, click ***HyperMorph*** > ***morph volumes***.
2. Go to the ***create*** subpanel.
3. Activate the entity selector.



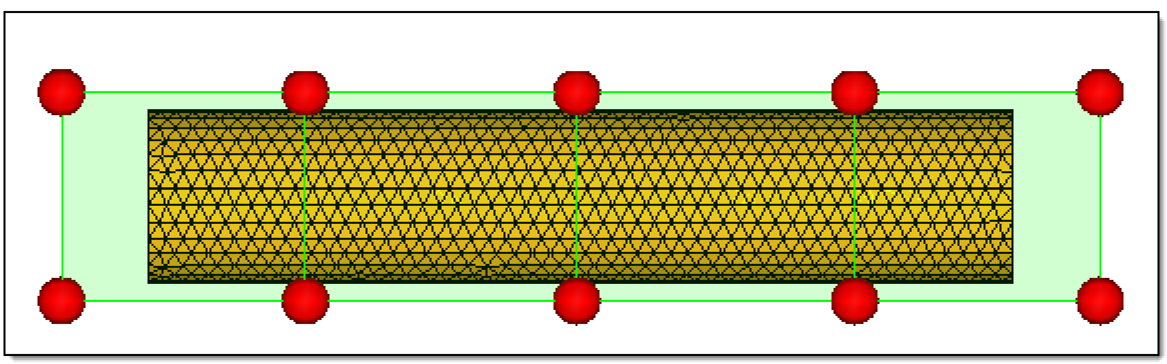
4. Click ***elems*** > ***all***. All of the elements in the model are selected.



5. Click ***create***. A new morph volume is created.
6. Go to the ***split/combine*** subpanel.
7. In the graphics area, double-click on the edge of the model that is marked by a green cross in the image below:



8. Click ***split***. The morph volume splits.
9. Continue splitting the morph volume so that it resembles the image below.

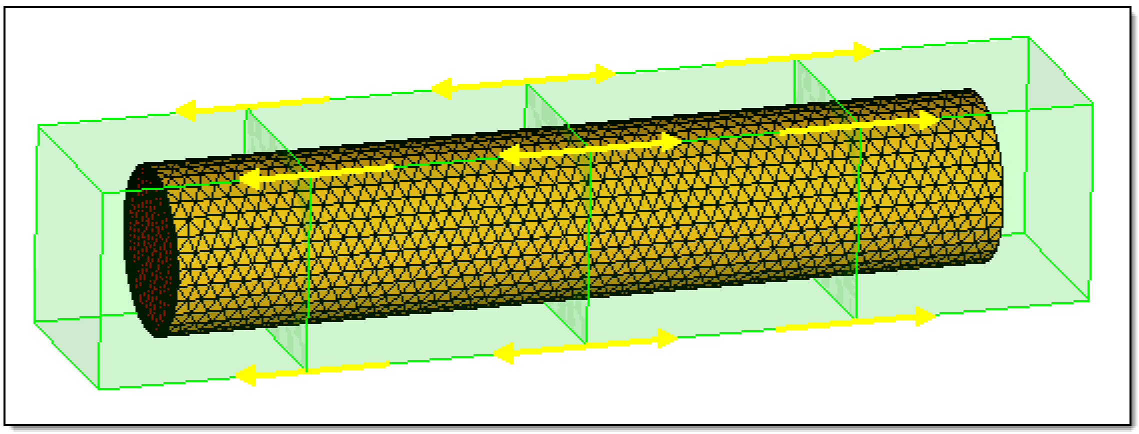


10. Go to the ***update edges*** subpanel.
11. Click the first arrow and select ***update ends***.
12. Click the third arrow and select ***master-slave***.

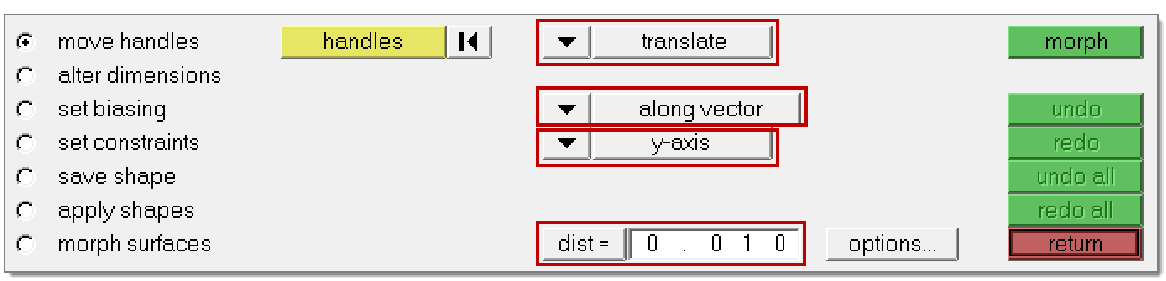
    Note: This options allows you to link any two edges together with a "master-slave" relationship between two morph volumes.

Settings for steps 3.11 and 3.12.

13. Define the master edge by clicking the first edge of the morph volume.
14. Define the slave edge by clicking the edge that touches the master edge.
15. Repeat steps 3.13 and 3.14 for all of the edges until your model resembles the image below (look for yellow arrows).

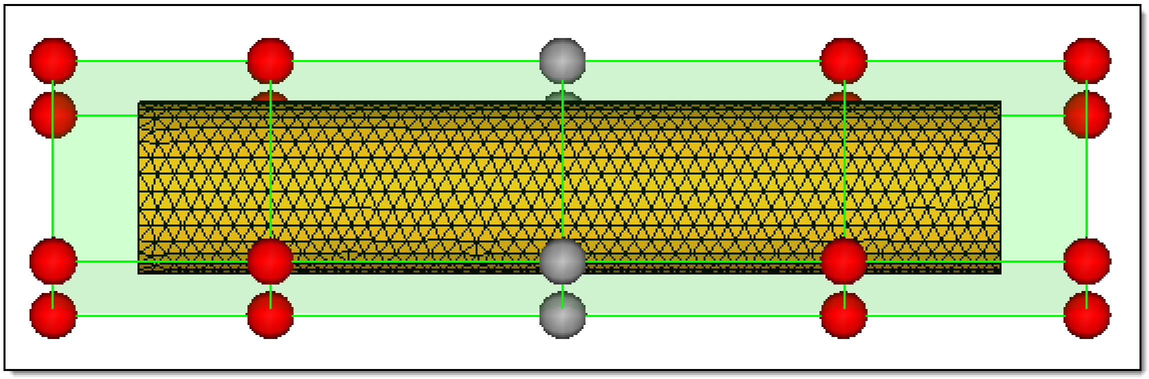


16. Click ***return*** to go back to the HyperMorph panel.
17. In the panel area, click ***morph***.
18. Go to the ***move handles*** subpanel.
19. Click the second arrow and select ***translate***.
20. Under **translate**, click the arrow and select ***along vector***.
21. Under **along vector**, set the orientation selector to ***y-axis***.
22. In the **dist =** field, enter `0.010`.

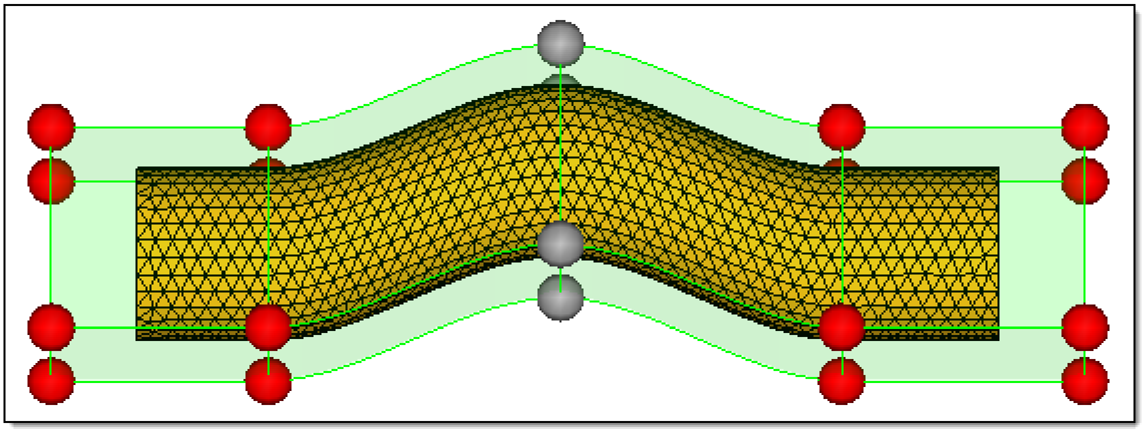


Settings for steps 3.19 through 3.22.

23. Activate the ***handles*** selector.
24. Select the four middle handles, highlighted in grey, in the image below:



25. Click ***morph***. The grid is morphed.



26. Go to the ***save shape*** subpanel.
27. In the **name=** field, enter `s1`.
28. Switch the first toggle from **as handle perturbations** to ***as node perturbations***.
29. Click ***save***.
30. In the window that appears, click ***Yes***.

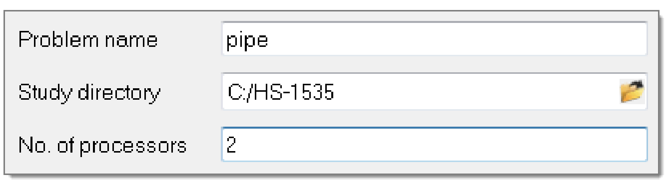
31. Click ***undo all***.
32. In the **Model** browser, right-click on the ***Shape*** folder and select ***Hide*** from the context sensitive menu. The shape, **s1** is hidden.
33. Right-click on the ***Shape*** folder again and select ***Show*** from the context sensitive menu. The shape, **sh1** reappears.

34. From the menu bar, click ***View*** > ***Browsers*** > ***HyperMesh*** > ***Utility***.
35. In the **Utility** browser, click ***Disp***.
36. Remove any temporary nodes in the model by clicking ***Clear Temp Nodes***.
37. Create design variables by clicking ***Design Study*** > ***Define DV*** from the menu bar.
38. Go to the ***desvar*** subpanel.
39. In the **desvar =** field, enter `bend`.
40. Click ***shape =***.
41. Link a shape with the shape design variable by clicking ***s1***.
42. Click ***create***. The design variable **bend** is created.
43. Click ***return***.

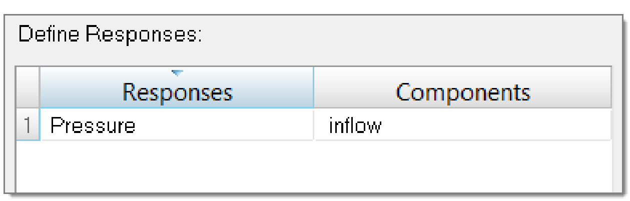
Step 4: Perform the Study in the HyperStudy Job Launcher

1. On the **CFD** toolbar, click . The **HyperStudy Job Launcher** opens.
2. In the **Study directory** field, navigate to the location of your working directory.

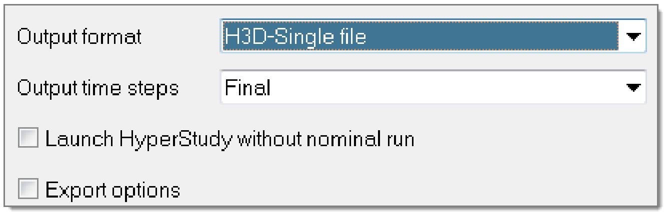
   Note: By default path will be the same directory in which the .hm file is saved. It is recommended that you create a separate folder for your study directory so that all of the files will be placed in that folder.
3. In the **No. of processors** field, enter `2`.



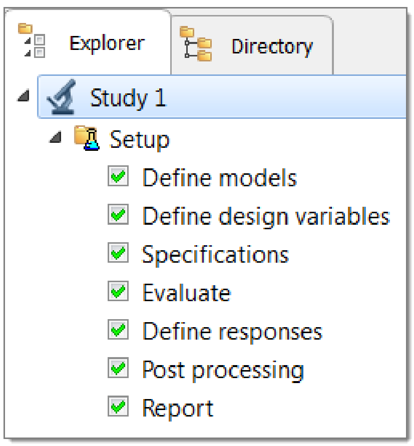
4. In the **Define Output Responses** table, select the following to identify what the pressure change will be at inflow due to shape changes.
   a. Set **Responses** to ***Pressure***.
   b. Set **Components** to ***inflow***.



5. To see how the pressure contours for each optimum design will look, set **Output format** to ***H3D-Single file***.



6. Click ***Launch***.
7. In the dialog that opens asking if you would like to continue, click ***Yes***.
8. In the dialog that opens informing you of the files that were created, click ***OK***.
9. In the dialog that opens asking if you would like to continue, click ***Yes***.
10. A nominal run is submitted, and acuProbe and acuTrail are launched to provide you with updated information about the run. Once the run is finished, HyperStudy opens and with the study **Setup** completed.



11. Go through each step to ensure that everything was properly defined.
12. In the **Define models** step, you will see that the resource file, solver input file, and solver execution script have been defined.

	Active	Label	Varname	Model Type	Resource	Solver input file	Solver execution script	Solver input arguments
1	☑	Model 1	m_1	Parameterized File {}	C:/...	pipe.crd	acusolve (acusolve)	$file

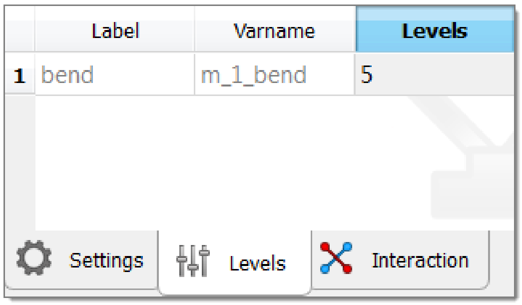
13. In the **Define Input Variables** step, you will see that the input variable you defined in HyperMesh Desktop has been imported.

	Active	Label	Varname	Lower Bound	Initial	Upper Bound
1	☑	bend	m_1_bend	-1.0000000 ...	0.0000000 ...	1.0000000 ...

14. In the **Define Output Responses** step, ensure that the output response is defined correctly.

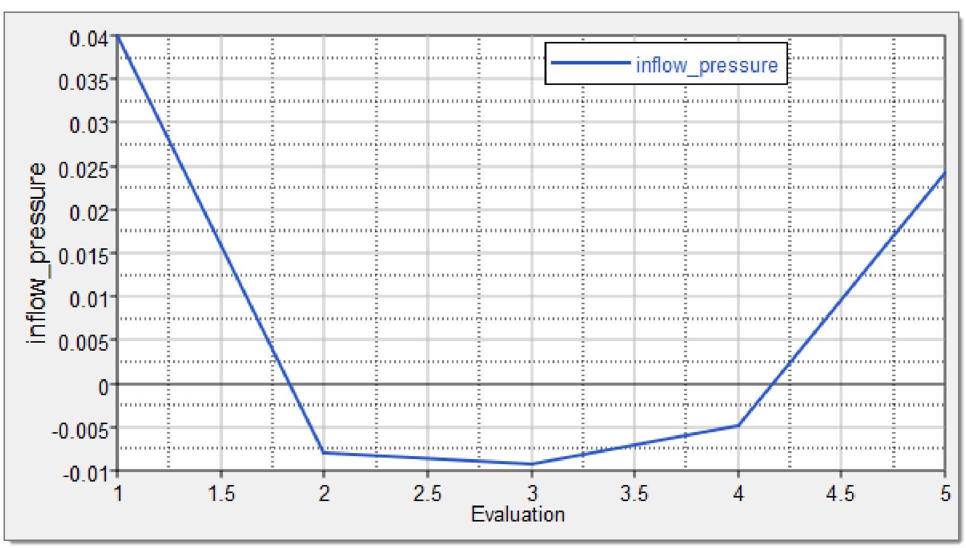
Step 5: Run a DOE Study

1. In the **Explorer**, right-click and select ***Add*** from the context menu.
2. In the **Add - HyperStudy** dialog, select ***Doe*** and click ***OK***.
3. Go to the **Specifications** step.
4. In the work area, set the **Mode** to ***Full Factorial***.
5. Click the ***Levels*** tab.
6. Change the number of levels to `5`.

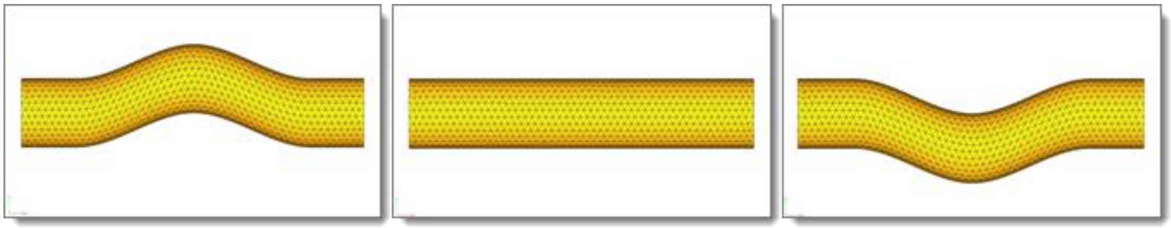


7. Click ***Apply***.
8. Go to the **Evaluate** step. The table used to run the study appears, showing all of the runs (1 - 5) to be executed.
9. Click ***Evaluate Tasks***.
10. To view the optimization results of the five runs in a table, click the ***Evaluation Data*** tab. The table displays the results of the five runs for the output responses.
11. To plot the optimization results of the five runs, click the ***Evaluation Plot*** tab.

    Plot the results of the **inflow_pressure** output response by selecting **inflow_pressure** with the **Channel** selector..



12. The extreme left (bend = -1), middle (bend = 0) and extreme right (bend = 1) results correspond to the following pipe shapes.



13. Click ***File*** > ***Save As*** from the menu bar to save the study.
14. In the **HyperStudy - Select location** dialog, navigate to your working directory and save the study.
15. The results of the DOE can be visualized in HyperView. Load the corresponding `*.h3d` file from the run folder into HyperView.