Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-1610: Setting Up a SimLab Model Using JavaScript

HS-1610: Setting Up a SimLab Model Using JavaScript

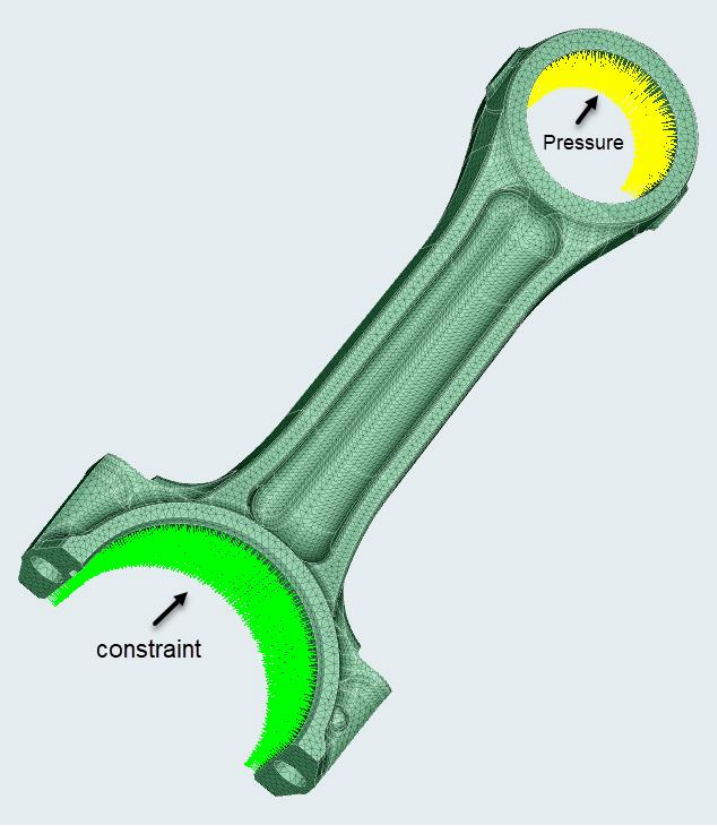
In this tutorial you will set up a mesh refinement study using a SimLab model to investigate the relationship between the SimLab mesh parameters and max Stress and max displacement.

Model Files

The files used in this tutorial can be found in `<hst.zip>/HS-1610/`. Copy these files to your working directory.

- ConnectingRod.xmt_txt
- Conrod_js_script.js
- HST_CONROD_Params.xml
- HST_CONROD_Responses.xml
- Mesh_spec_param.xml

The model used in this tutorial is a Parasolid CAD file (ConnectingRod.xmt_txt) that contains a connecting rod. The connecting rod is loaded at one end and constrained at the other.



Connecting rod mesh representation with loads and boundary conditions

Pre-Requisites

In this tutorial you are required to use Simlab v2017.2.1 and HyperStudy 2017.2.2. HyperStudy v2017.2.2 does not require you to set up the environment variable HW_HST_CMD_SIMLAB.

Before creating the parameters inside Simlab, pause the recording of javascript. This is a known limitation of SimLab. The parameters are passed separately from the `.xml` file to the `.js` file. If the parameter definition already exists in the `.js` file, then any changes in the values will be overwritten.

Exercise

Step 1: Register the SimLab Solver Script

1. Open HyperStudy.
2. From the menu bar, click **Edit** > **Register Solver Script**.
3. In the **Register Solver Script – HyperStudy** dialog, SimLab row, Path column, click .
4. In the **Open** dialog, navigate to `SimLab2017.2.1/bin/win64` and open the `SimLab.bat` file.
5. Click **OK** to close the **Register Solver Script – HyperStudy** dialog.

	Label	Varname	Type	Preference	Path
11	HyperStudy Batch	hstbatch	Generic	HyperWorks	D:/Altair/2017.2/hst/bin/win64/hstbatch.exe
12	HyperWorks	hw_exe	Generic	HyperWorks	D:/Altair/2017.2/hw/bin/win64/hw.exe
13	HV Trans	hvtrans	Generic	HyperWorks	D:/Altair/2017.2/io/result_readers/bin/win64/hvtrans.exe
14	None	HstSolver_None	None	Internal	D:/Altair/2017.2/hst/bin/win64/hstsolver_none.bat
15	SimLab	HstSolver_SimLab	SimLab	HyperStudy	D:/Altair/2017.2/SimLab2017.2.1/bin/win64/SimLab.bat

Step 2: Perform the Study Set Up

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 SimLab model.
    a. Click **Add Model**.
    b. In the **Add – HyperStudy** dialog, select **SimLab** and click **OK**.
    c. In the Resource column, click .

d. In the **HyperStudy – Load model resource** dialog, navigate to `HST_CONROD` and open the `Conrod_js_script.js` file.

Notice: The Solver Input Arguments field automatically displays `with -nographics -auto Conrod_js_script.js -param HST_CONROD_Params.xml -response HST_CONROD_Responses.xml`.

5. Click **Import Variables**.
6. Go to the **Define Input Variables** step.
7. In the work area, Active column, clear the checkboxes for the FilletMeshSize and Load input variables.

Note: In this tutorial you will only focus on the BodyMeshSize input variable.

8. For the BodyMeshSize input variable, change the Lower Bound to `2.0` and the Upper Bound to `8.0`.

	Active	Label	Varname	Lower Bound	Nominal	Upper Bound	Comment
1	☑	BodyMeshSize	var_1	2.0000000 ...	3.0000000 ...	8.0000000 ...	...
2	☐	FilletMeshSize	var_2	1.8000000 ...	2.0000000 ...	2.2000000 ...	...
3	☐	Load	var_3	-270000.00 ...	-300000.00 ...	-330000.00 ...	...

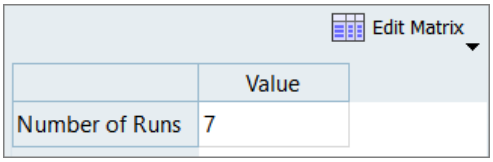
9. Go to the **Specifications** step.

Step 3: Perform the Sweep

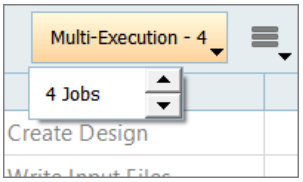
1. In the work area, set the **Mode** to **Sweep**.

	Mode	Label	Varname	Details
1	○	Nominal Run	Nom	Run system at nominal values
2	○	System Bounds Check	Chk	Run system at nominal values, then lower an...
3	◉	Sweep	Sweep	Sweep system values from lower to upper val...

2. From the Settings, set the Number of Runs to `7`.



3. Click **Apply**.
4. Go to the **Evaluate** step.
5. From Run Tasks, click **Multi-Execution** and enter `4 jobs`.

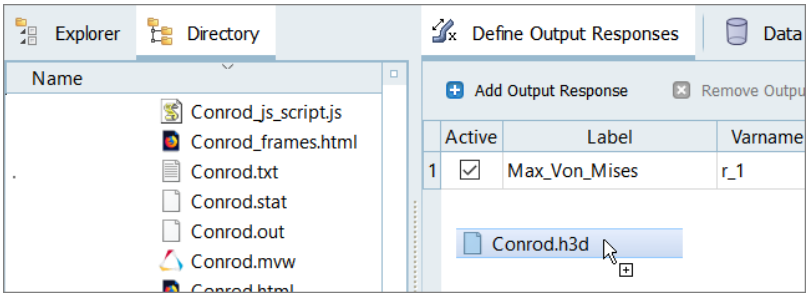


6. Click **Evaluate Tasks**.
7. Go to the **Define Output Responses** step.

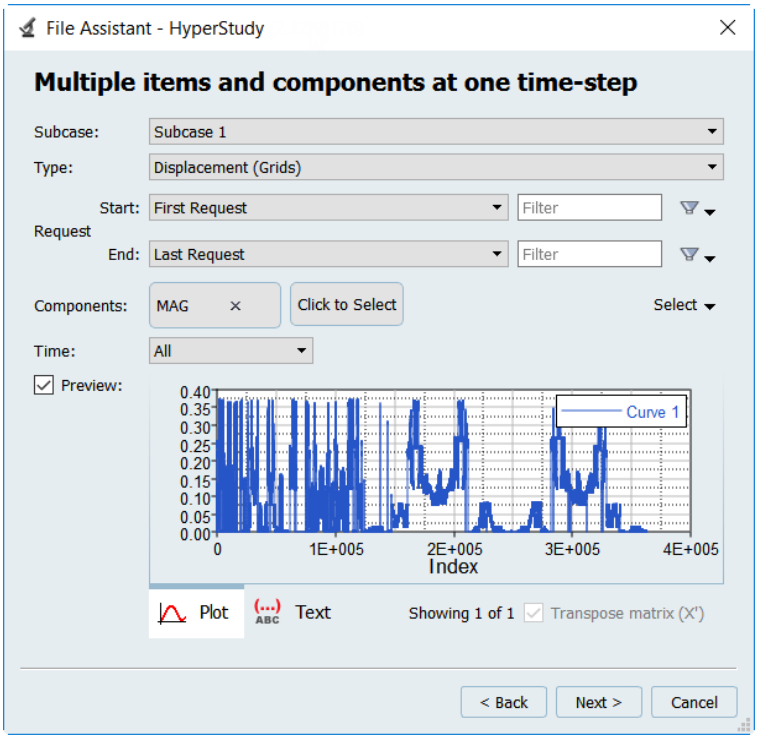
Step 4: Define Output Responses

The Max_Von_mises output responses was extracted from the `Conrod_js_script.js` file in the SimLab model. In this step you will create an additional output response, Max_Disp.

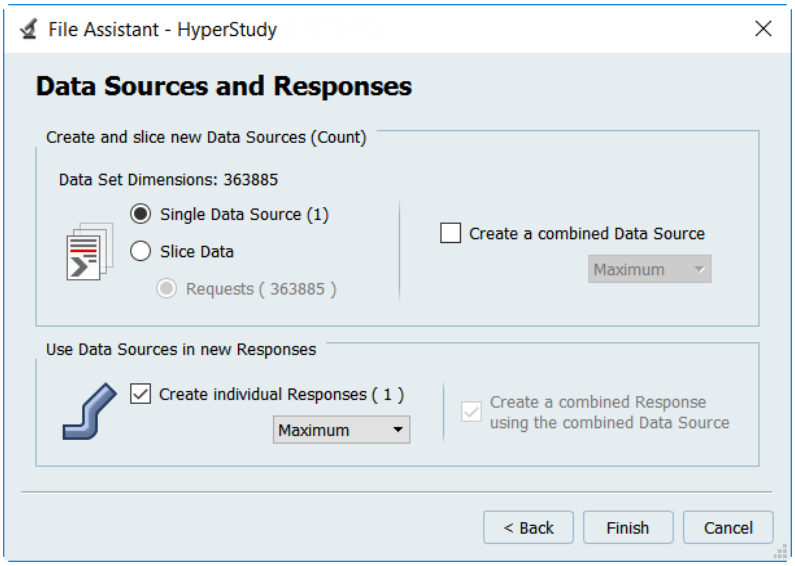
1. From the Directory, drag-and-drop the `Conrod.h3d` file, located in `approaches/nom_1/run_00001/m_1`, into the work area.



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 settings and click **Next**.
    a. Set Subcase to **Subcase 1**.
    b. Set Type to **Displacement (Grids)**.
    c. For Request, set Start to **First Request** and End to **Last Request**.
    d. For Components, select **MAG**.
    e. Set Time to **All**.



5. Define the following settings and click **Finish**.
   a. Set Data Set Dimensions to **Single Data Source (1)**.
   b. Clear the **Create a combined Data Source** checkbox.
   c. Enable the **Create individual Responses(1)** checkbox and set it to **Maximum**.

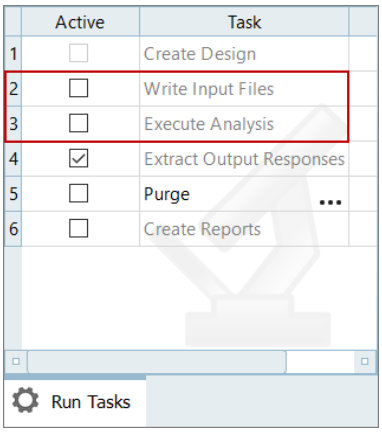


6. In the work area, Label the second output response `Max_Disp`.
7. Click **Evaluate** to extract the output response values.

	Active	Label	Varname	Expression	Value	Comment
1	☑	Max_Von_Mises	r_1	ds_1[0] ...	664.02800	...
2	☑	Max_Disp	r_2	max(ds_2) ...	0.3712362	...

Step 5: Extract Evaluation Data for all 7 Runs in the Sweep Study

1. Go to the **Evaluate** step.
2. From the Run Tasks tab, clear the **Write Input Files** and **Execute Analysis** checkboxoes.



3. Click **Evaluate** Tasks.

Step 6: Post-Processing

1. Go to the **Post-Processing** step.
2. Click the **Scatter** tab.
4. From the Channel selector, set the X Axis to **BodyMeshSize** and the Y Axis to **Max_Von_Mises** and **Max_Disp**.

   The results of the scatter plot indicate that as the size of the mesh gets smaller (moving along the x-axis to the left), displacement starts to converge. However, stress does not converge. This behavior is typical in finite element models when displacement converges before derived quantities such as stress. In this tutorial, the Max_Von_Mises output response may not converge at all due to the location of the maximum stress in the model (adjacent to the load application area), which can be seen by opening the resulting file in HyperView.