



Altair

---

**HyperWorks**

Altair HyperMesh 2019 Tutorials

HM-4070: OptiView

[altairhyperworks.com](http://altairhyperworks.com)

## HM-4070: OptiView

In this tutorial, you will:

- Import an optimization model
- Create a new set of optimization entities
- Organize these into optimization problems
- Run both problems
- View results

### Model Files

---


This exercise uses the `cclip.fem` file, which can be found in the `hm.zip` file. Copy the file(s) from this directory to your working directory.

### Exercise

---

#### Step 1: Launch HyperMesh and set the user profile to OptiStruct.

#### Step 2: Import the `cclip.fem` file.

1. Select **Import Solver Deck**  from the **Standard** toolbar.
2. Select **OptiStruct** for the **File type**.
3. Browse to `<installation_directory>\tutorial\hm\` and select `cclip.fem`.
4. Click **Import** to open the file.

#### Step 3: Create Problem 1 and organize optimization entities.

1. In the **Model** browser, click  (**Optimization View**).

- Review the **Optimization Repository**. The **Optimization Repository** gives an overview of all optimization related entities in the database. Info types and children entities help give a clear snapshot without having to review individual entities.

Entities	ID	Type	Info
Optimization Problems			
Optimization Repository			
Design Object Reference (0)			
Optimization Constraint (2)			
CONST2	2	DISPLACEMENT	LB -0.07
lowerdi	3	DISPLACEMENT	
Load Step (1)			
CONST1	1	DISPLACEMENT	UB 0.07
upperdi	2	DISPLACEMENT	
Load Step (1)			
Design Variable Link (0)			
Objective (1)			

Figure 1

- Right-click the **Optimization Problems** folder and click **Create > Optimization Problem**. HyperMesh creates an optimization problem and opens it in the **Entity Editor**.
- In the **Entity Editor**, name this problem `Topology`.
- Drag and drop all the entities from the repository into the newly created problem.
 

**Note:** You can drag and drop entities from the repository into problems or problems into problems. Any combination of selected entities can be dragged and dropped.

#### Step 4: Define a new set of optimization entities.

- In the **Model** browser, right-click and select **Create > Free Size Desvar** from the context menu. The **Free Size Optimization** panel opens, from which you can define a free size design variable.
 

Tip: Give the free size design variable a meaningful name so you can easily drag and drop.
- Use the **props** selector to select the **shells** property.
- You can create new response/constraint pairs, or anything else you want to change from problem to problem. For this tutorial, we will just compare **Topology** to **Free Size**.

#### Step 5: Create Problem 2 and organize optimization entities.

- In the **Model** browser, right-click and select **Create > Optimization Problem** from the context menu.
- In the **Entity Editor**, name this problem `Free Size`.
- Drag and drop all the entities defined in the repository to the newly created problem.
 

**Note:** Two design variables will be defined for the Free Size problem.

4. In the **Free Size** folder, right-click on the **shell** design variable and select **Remove from Problem** from the context menu.

**Note:** This will not delete the problem from the repository.

### Step 6: Set problems to export and run.

1. In the **Model** browser, right-click on the **Topology** problem and select **Set Export** from the context menu.

**Note:** Once problems are defined, only one can be exported at a time. The problem set to export is in bold, and furthermore, the **Entity State** browser shows these rules.

2. Open the **OptiStruct** panel.
3. Set the **export options** to **custom**.
4. Save the input file as `cclip_topology.fem`.
5. Click **OptiStruct** to run the analysis.

**Note:** The Optimization View allows one `.hm` for all optimization problems, and it is up to the user to wisely name each input file.