

Altair

HyperWorks

HM-3140: Batch Meshing

In this tutorial, you will learn how to:

- Define a configuration for the batch mesh
- Edit the criteria and parameter files
- Setup a simple user procedure for a post-batch mesh run

Batch Mesher is a tool that can perform geometry cleanup and automeshing (in batch mode) for given CAD files. Batch Mesher performs a variety of geometry cleanup operations to improve the quality of the mesh created for the selected element size and type. Cleanup operations include: equivalencing of "red" free edges, fixing small surfaces (relative to the element size), and detecting features.

Batch Mesher also performs specified surface editing/defeaturing operations such as: removal of pinholes (less than specified size), removal of edge fillets, and addition of a layer of washer elements around holes.

All user-defined criteria determines the quality index (QI) of a model. The QI value is used to assess the potential of each geometry cleanup and meshing tool, and apply them accordingly. QI optimized meshing and node placement optimization are performed to obtain the best quality meshing. Final results are stored in a HyperMesh database file.

Tools

To start Batch Mesher on Windows:

- Click on the **Start** menu > **All Programs** > **Altair HyperWorks** > **BatchMesher**.

Or

- Type `hw_batchmesh` with the full path (`~altairhome\hm\batchmesh\hw_batchmesh`).

To start Batch Mesher on UNIX:

- Type the `hw_batchmesh` command to invoke the user interface or `hw_batchmesh -nogui...` to perform the batch mesh without a user interface.

Model Files

This exercise uses the following files found in the `hm.zip` file. Copy the file(s) from this directory to your working directory.

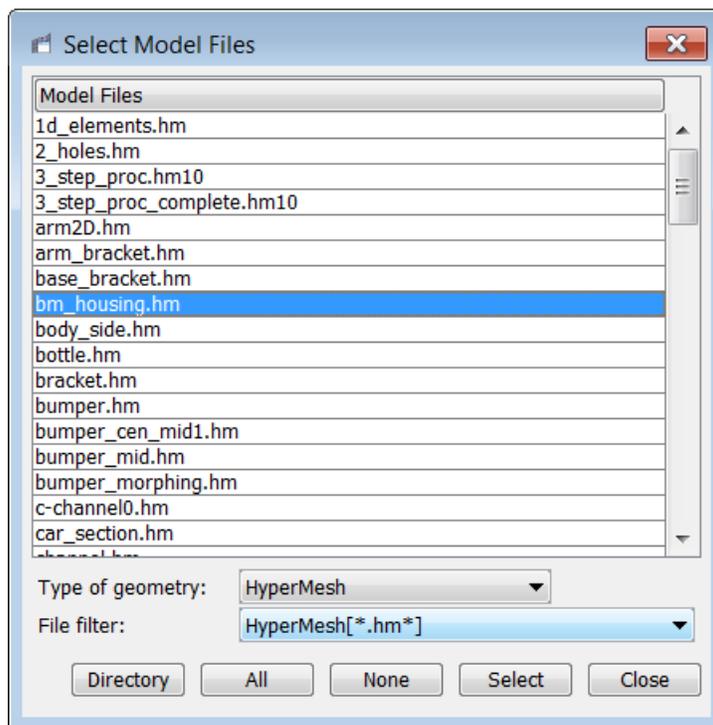
- `part1.hm`
- `part2.hm`
- `bm_housing.hm`
- `bm_housing.criteria`

- bm_housing.param
- bm_housing.tcl

Exercise

Step 1: Open BatchMesher.

1. Start **BatchMesher** as described above.
2. In the **Input model directory** field, click  and navigate to your working directory.
Note: In this exercise, you will use files located in the `hm.zip` file.
3. In the **Output directory**, click  and navigate to the appropriate directory if different from the Input model directory.
4. Optional: Move all of the `.hm` output files to the corresponding directory of the input model by selecting the **Relocate .hm files to input model directory** checkbox.
5. On the right-hand side of BatchMesher, click . The **Select Model Files** dialog appears.



6. From the **Type of Geometry** list, select the appropriate CAD data type. For this tutorial, use the geometry already loaded in HyperMesh, therefore select **HyperMesh**.
Note: A filter will help select applicable HyperMesh models for batch meshing.

7. Select the following model files: `part1.hm`, `part2.hm` and `bm_housing.hm`.

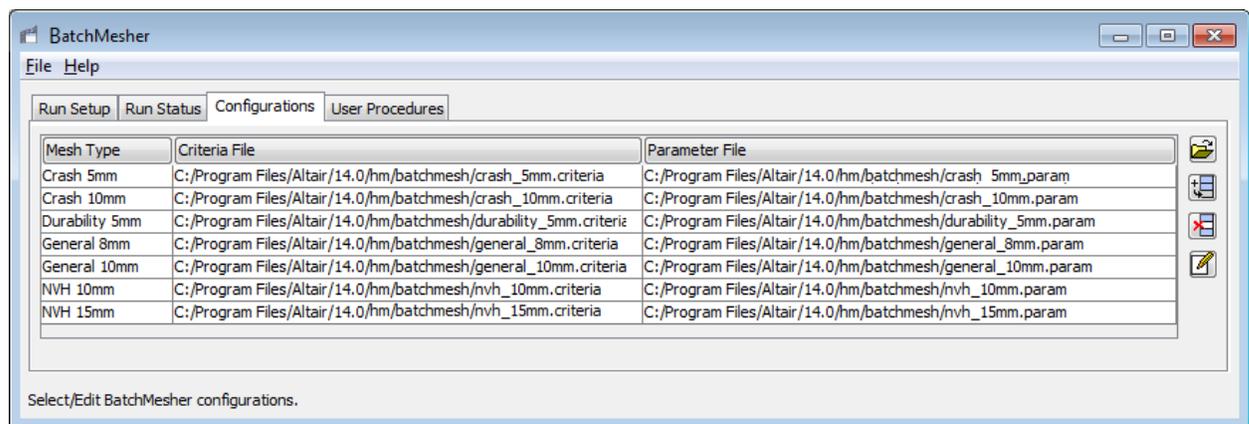
Tip: Select multiple files by pressing CTRL while selecting files.



8. Click **Select**.

Step 2: Define a configuration for the batch mesh run.

1. Click the **Configurations** tab. BatchMesh displays several common configurations already available.

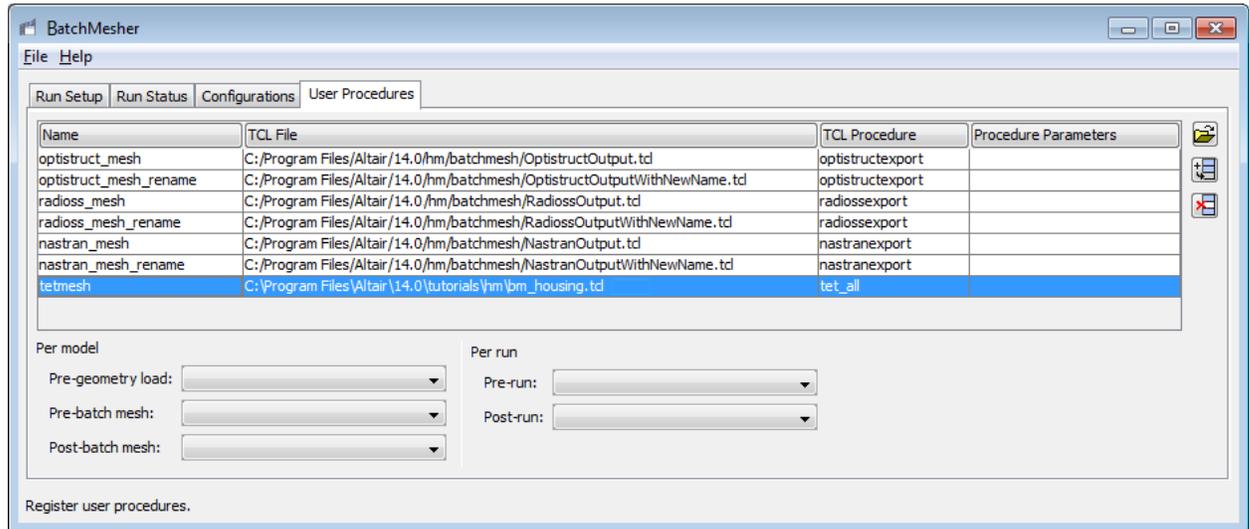


2. On the right-hand side of BatchMesher, click . BatchMesher adds a new entry to the table of configurations.
3. Double-click the **Mesh Type** field, and then enter a name for the mesh type.
4. Double-click the **Criteria File** field, and then click .
5. In the **Select Criteria File** dialog, select the `bm_housing.criteria` file.
6. Double-click the **Parameter File** field, and then click .
7. In the **Select Parameter File** field, select the `bm_housing.param` file.
8. Click the **Run Setup** tab.
9. Click the **Mesh Type** field. The new mesh type you created is now available for selection.

Step 3: On the **User Procedures** tab, set up a simple script to perform a tetramesh on the housing.

1. Click the **User Procedures** tab.
2. On the right-hand side of BatchMesher, click . BatchMesher adds a new entry to the table of user procedures.

3. Double-click the **TCL File** field, and then click .
4. In the **Select Tcl File** dialog, select the `bm_housing.tcl` file.
5. Click the **TCL Procedure** field, and then select **tet_all** from the list of procedures.
6. Double-click the **Name** field, and then enter a name for the procedure. For example, `tetmesh`.



7. Click the **Run Setup** tab.
8. Click the **Post-Mesh** field. The new post-batch meshing script you created is now available for selection.

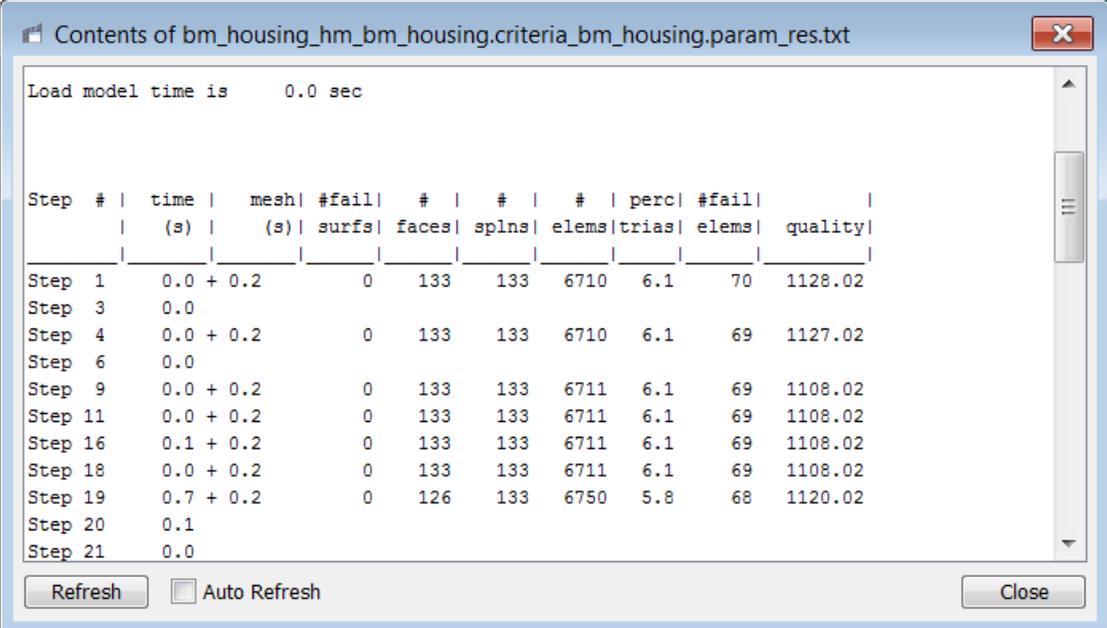
Step 4: On the **Run Setup** tab, begin defining a configuration for the batch mesh run.

1. For each geometry file, click the **Mesh Type** field and select the predefined mesh types as follows:
 - `bm_housing.hm`: use the mesh type you defined in Step 2.
 - `part1.hm`: use **Crash 5mm**.
 - `part2.hm`: use **Crash 5mm**.
2. For the Geometry file `bm_housing.hm`, click the **Post-Mesh** field and select the procedure you created in Step 3.

Note: This will run the tetramesher on all available shell elements after batch meshing.

Geometry File	Geom Type	Mesh Type	Pre-Geom Load	Pre-Mesh	Post-Mesh
...\bm_housing.hm	HyperMesh	Tetmesh			tetmesh
...part1.hm	HyperMesh	Crash 5mm			
...part2.hm	HyperMesh	Crash 5mm			

- Click **Submit** to initiate the run, or click **Submit At** to submit the job at a specified time. BatchMesher switches to the **Run Status** tab. As the three jobs run, the **Status** changes from **Working** to **Pending** to **Done**.
- Obtain more details on a single job when its status is **Working** by selecting the job and then clicking **Show Details**. A detailed summary appears with a table containing information about the model during each step of the batch meshing process, such as the number of failed elements and the quality index.



Load model time is 0.0 sec

Step #	time (s)	mesh (s)	#fail	#surfs	#faces	#splns	#elems	#trias	#elems	quality
Step 1	0.0 + 0.2		0	133	133	6710	6.1	70	1128.02	
Step 3	0.0									
Step 4	0.0 + 0.2		0	133	133	6710	6.1	69	1127.02	
Step 6	0.0									
Step 9	0.0 + 0.2		0	133	133	6711	6.1	69	1108.02	
Step 11	0.0 + 0.2		0	133	133	6711	6.1	69	1108.02	
Step 16	0.1 + 0.2		0	133	133	6711	6.1	69	1108.02	
Step 18	0.0 + 0.2		0	133	133	6711	6.1	69	1108.02	
Step 19	0.7 + 0.2		0	126	133	6750	5.8	68	1120.02	
Step 20	0.1									
Step 21	0.0									

Refresh Auto Refresh Close

- Once a job's status is **Done**, click **Load Mesh**. BatchMesher loads the mesh into HyperMesh for model interrogation.
- Obtain an overall run status of all the jobs by clicking **Run Details** once all of the jobs have been meshed.
- While the jobs are running, you can pause or cancel them. If you pause a job, it can be resumed immediately or you can have it resume at a specific time.
- Once the BatchMesher session has been setup with file directories and mesh types, you can save it as a config file and load it at a future time.
- It is also possible to load an entire set of models that have already been batch meshed in order to take advantage of the **Load Mesh** option in the **Run Status** tab.
- If you make modifications to your criteria or parameter files, you can submit a run again. BatchMesher will place the new files in a separate sub-directory.

Step 5 (Optional): Edit the criteria and parameter files.

- Click the **Configurations** tab.
- Right-click on the **Criteria File** or **Parameter File** field that you wish to edit and select **Edit File** from the context menu. The **Criteria and Parameters Files Editor** dialog appears.

- From the **Criteria** tab, you can set the target element size, element criteria, and the method that is used to calculate the values. You can also select the **Advanced Criteria Table** check box to enabled additional options that give you more control over the intermediate QI values, however, it is usually not necessary to edit these options in order to obtain a good quality mesh. The **Use min length from timestep calculator** check box is also available for explicit solver models. If you select this check box, the overall minimum element size will be set by this option and the top element checks will be disabled.
- From the **Parameters** tab, you can set all of the meshing controls over various geometric features. Parameters are grouped into sections; you can click the small downward-arrows next to each section to show and hide that section. The number of parameters is extensive; for more details, see the [Parameters Editor](#) topic in the BatchMesher help.

