



Altair

HyperWorks

HM-9000: Exporting Data for Fatigue Analysis

This tutorial demonstrates how to write an input file for a given fatigue solver using the options available on the **Fatigue** panel.

Model Files

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

Step 1: Retrieve the model file.

For this tutorial, retrieve the file, `keyhole.hm`. This file contains a finite element (FE) model, for which an analysis has already been conducted, to obtain the stress/strain information for durability loads of interest.

1. From the menu bar, click **File > Open > Model**.
2. Open the `keyhole.hm` file.

Step 2: Load the results file.

1. From the menu bar, click **File > Load > Results**.
2. Open the `keyhole.res` file.

Step 3: Export data and write a fatigue solver input deck.

1. Open the **Fatigue** panel by clicking **fatigue** from the **Post** page.

The screenshot shows the Fatigue panel with the following settings:

- FE Analysis Type: static/modal
- Output File Format: ascii
- output file: (empty field)
- data group = (text input field)
- select simulation: all
- Buttons: write, save failed, return

2. Toggle **FE Analysis Type** to **static/modal**.

Results contained in `keyhole.res` were obtained from linear statics analysis.

Note: Select the **transient dynamic** option if a dynamic finite element analysis was used to obtain the stress/strain results for the model.

3. Toggle **Output File Format** to **ascii**.

Note: Select the **binary** option if the fatigue solver allows a binary input file. For more information on fatigue solvers and acceptable input file formats, see the **Fatigue** panel documentation.

- Click **browse...** and locate the file folder you want store the file in, then enter a name for the output file.

Note: This output file is the input file for the fatigue solver.

- Click **data group =** and select any of the data groups that you want to write to the output file.

Data groups are organized based on whether nodal or elemental results are available in the results file.

- Set **select simulation** to **all**.

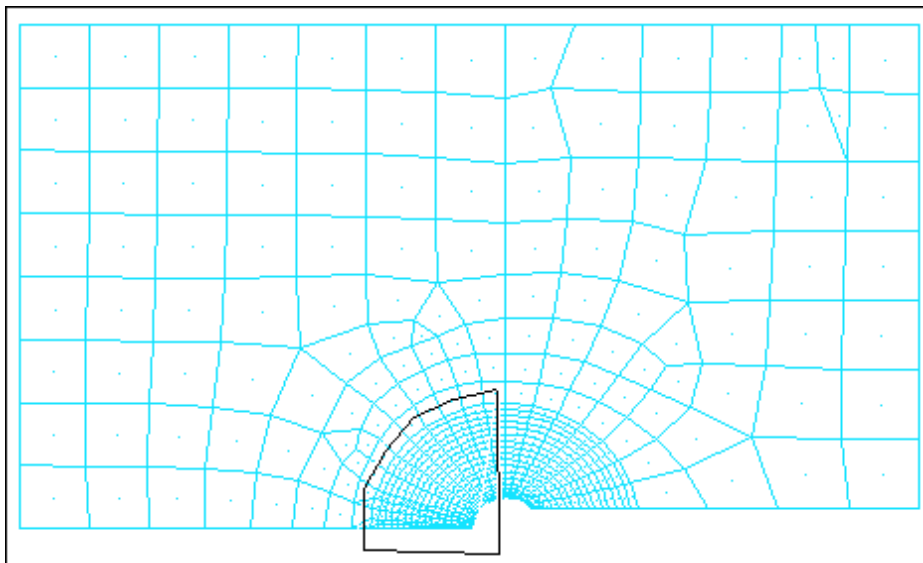
This specifies the data in the results file that is written to the output file. In this case, selecting **all** writes the stress/strain data for the selected nodes or elements for all loadcases represented in keyhole.res.

Note: You can write out stress/strain information for **all** of the time steps, or you can choose a **range** from a starting time step to an ending time step, or you can choose a **selection** of time steps manually from the available list.

The next sub-step is to select the entities for which the finite element analysis results file is written.

Note: The type of entity you select is based upon the data group you selected. Select **nodes** if the data group you selected refers to nodal results. Similarly, select **elements** if the data group you selected refers to elemental results. Select **sets** to choose a predefined entity set comprised of nodes or elements corresponding to a data group with nodal/elemental results, respectively. If the data group results and the entity type are not the same, HyperMesh displays the error message, "Results file doesn't contain nodal values."

- Set the selector to **elems**.
- Click **elems >> by window**.
- Draw the window as shown in the following image.



- Click **interior**.
- Click **select entities**.

12. Click **write**.

An ASCII file is written to the selected directory.

You can read this file into the appropriate fatigue solver to complete the fatigue analysis.