Altair HyperWorks™

2022

FE Configuration File

The FE configuration file (`feconfig.cfg`) is used to define custom welds such as ACM (Area Contact Method) and other special types.

Weld definitions are solver dependant (Nastran, LS-DYNA, and so on). The weld definition in the file includes the type of weld to create and the surrounding connector to shells. The specific solver template for the type of weld must be loaded before the welds can be created using a connector entity. When HyperWorks launches, it searches for `feconfig.cfg` in the following locations, listed in search order:

1. The product installation path
2. HW_CONFIG_PATH (a specified environment variable)
3. Your home directory (in UNIX, for example)
4. The current working directory.

By default, the `feconfig.cfg` file from the `<install_directory>/hm/bin` directory is loaded in each of the panels related to each connector type, such as Spot, Seam and Area. It is not recommended to have more than one config file of the same name, even in different directories, as the results can be unpredictable as to which one will be used by default.

Weld Definition Template

```
CFG <SOLVER> <USER_FE_TYPE> <USER_FE_NAME>
*filter <FILTER_TYPE > [<FILTER_TYPE>]
*style < STYLE_TYPE> <STYLE_NUM>
*head
<HM_FE_CONFIG> <HM_FE_TYPE> <RIGID_FLAG>
*body <BODY_FLAG>
<HM_FE_CONFIG> <HM_FE_TYPE> <LENGTH_LOCATION_FLAG> [<DOFS>]
[<HM_FE_CONFIG> <HM_FE_TYPE> <LENGTH_LOCATION_FLAG> [<DOFS>]]
*post <POST_SCRIPT_NAME>
```

Figure 1. Weld Definition Template

Where,

CFG

Keyword to start a custom weld definition.

SOLVER

The solver template for which FE needs to be created.

Supported solvers are: Abaqus, ANSYS, LS-DYNA, Nastran, OptiStruct, PAM-CRASH, or PAM-CRASH 2G.

USER_FE_TYPE
: A unique (with respect to a solver) user defined configuration type ID. Customer-defined CFGs should use numbers greater than 10,000 to ensure no collisions with future native HM CFGs.

USER_FE_NAME
: The user-specified name for the FE configuration. The specified name is saved and displayed in the Connector Browser.

Note: This should be the first line in the custom weld definition.

*filter

Allows only the specified connector types to realize the configuration.

For example, *FILTER spot seam indicates that this configuration can be realized only by the spot and seam connector types. In addition, this option is used as a filter when displaying FE configurations in the type = field of respective realize panels.

*filter lines also set which panel the CFG is visible in.

CE_TYPE
: The connector type, spot, bolt, seam, area, and so on.

*style

Indicates that the configurations have specific behaviors associated during realization, and that they are native types.

Note: The style definition line for these configurations must not be edited.

For example, *style bolt 1 indicates that this is a bolt connection of type 1 that creates a specific bolted connection between the parts.

STYLE_TYPE
: The connector style name, such as "adhesive", "bolt", "acm", "quad", "continuous", and so on.

STYLE_NUM
: The connector style number:

- Adhesive:

  "1" mesh independent adhesive nodes tie to shells with RBE3/RBE2.

  "2" force shell gap length on. Adhesive (HEXA element) shares nodes with shell at co-incident locations.

- Bolt:

  "0" normal bolt: "wagon wheels" in the holes.

  "1" symmetrical spider bolt.

  "2" unsymmetrical spider bolt: the middle node is biased towards one hole.

  "3" cylinder bolt: ties together all nodes within virtual cylinder.

- ACM:

  "1" share the nodes of HEXA element for consecutive layers (> 2T) and the length of HEXA is average of part thickness.

  "2" HEXA elements in consecutive layers have unique nodes and the length of HEXA is average of part thickness.

  "3" share the nodes of HEXA elements for consecutive layers and the length of HEXA is the gap distance between parts.

  "5" HEXA nodes are not equivalenced with the shell nodes/washer nodes, if a washer is present.

- Quad

  "1" create two sets of QUAD4 elements, first along projection direction and second at an orientation determined by average part thickness.

  "2" create one set of QUAD4 elements at an orientation determined by average part thickness.

***head**

The string head is required to specify that a rigid is to be created to connect the weld node to the surrounding shell element.

*head lines must be followed with at most one HM_FE_CONFIG line.

HM_FE_CONFIG

The config for the rigid currently supported.

The various types supported are "bar2", "bar3", "equations", "gap", "hex8"(3D), "plot", "mass"(0D), "rigid", "rigidlink", "rbe3", "rod", "spring", "weld", "quad4"(2D seam only), or "penta6"(3D adhesive only).

HM_FE_TYPE

Unique (with respect to a solver) user defined configuration type id defined in the solver template.

RIGID_FLAG

Defines the number and arrangement of rigids.

"0" is a single rigid

"1" is multiple rigids

"2" is multiple rigids to outer shell nodes (for 2D bolt washers only)

"3" is multiple rigids to outer alternate shell nodes (for 2D bolt washers only)

"10" is multiple rigids with a 0 length leg connecting with body (for bolt only)

"12" is multiple rigids to inner and outer shell nodes (for 2D bolt washers only)

"13" is multiple rigids to inner and outer alternate shell nodes (for 2D bolt washers only)

DOFS (Optional) — Degrees of freedom of the rigid (1-6).

*body

Specifies that a weld is to be created to connect the link entities added to the connector.

*body lines may be followed by one or more HM_FE_CONFIG lines

BODY_FLAG — The body flag is used to calculate the length of the weld. If the body flag = 0, the length is calculated based on the distance between the connecting layers (link entities). If the body flag = 1, the length is calculated based on the average thickness of the connecting layers (link entities).

HM_FE_CONFIG — The config for the rigid currently supported. The various types supported are "bar2", "bar3", "equations", "gap", "hex8"(3D), "plot", "mass"(0D), "rigid", "rigidlink", "rbe3", "rod", "spring", "weld", "quad4"(2D seam only), or "penta6"(3D adhesive only).

HM_FE_TYPE — The solver defined type for the HyperMesh config. For example, CBUSH is of config spring and type 6. The type number is defined in respective solver templates and differs, based on the solver.

LENGTH_LOCATION_FLAG

0D element details.

"0" place the 0D element along the proposed 1D element path. If this 0D element is the only config given in the *body, then it is placed at the center of the proposed 1D element path.

"1" has the same behavior as "0" except only a single 0D element is created even if multiple bodies are created (as happens in >2T welds)

"2" place the 0D element at the connector location.

1D element details.

"0" force zero length welds.

>"0" but <"1" denote a percentage of the distance between shells the length of a given weld should be. To create series welds, all the PERCENT_LENGTH_FLAG variables for a given *body must add up less than or equal to 1.0. Example: 0.33 or 0.50

"1" force each body weld to have a length equal to the distance between the shells, which can be used for parallel welds.

"2" place the 1D element at the connector location, with both nodes coincident.

"3" place multiple (thickness-1) 1D elements at the connector location connected end-to-end, with all nodes coincident.

3D element details.

"0" force a floating hexa element to have a length equal to half the distance specified by the BODY_FLAG.

"1" force the hexa element's length to be equal to the full distance specified by the BODY_FLAG.

DOFS (Optional)

The degrees of freedom (1-6) of the rigid.

*post

*post lines are optional, but if specified, they must be followed by the name, excluding path, of a valid TCL script with a `.tcl` extension. HyperMesh searches for the TLC script in the locations and order specified below:

1. Current working directory
2. Users home directory
3. Paths in HW_CONFIG_PATH environment variable
4. Installation directory
5. `hm/bin` directory
6. `hm/scripts` directory
7. `hm/scripts/connectors/` directory

This post script will be automatically executed post FE realization, and can be used to edit weld properties, attributes, and other solver specific details.

FE Specification Rules

Each solver will have a specific definition so the same user-defined types can be repeated for each solver.

The head and the body definition must begin with a "*" to define rigid and weld definitions.

Multiple solid element combinations are not currently supported. Therefore, an ACM can have only one hexa weld element specified in the definition.

1D and 3D element combinations are not supported.

The total length of series welds cannot exceed 1.0 (100 percent). Hence there cannot be three welds specified in series having a length factor of 0.5 (50 percent) each.

Series and parallel weld element combinations are not supported.

Series welds are not supported where the link entities are coincident. Series welds are not created when the distance between the connecting link entities is zero.

User comments should start with a hash character "#".