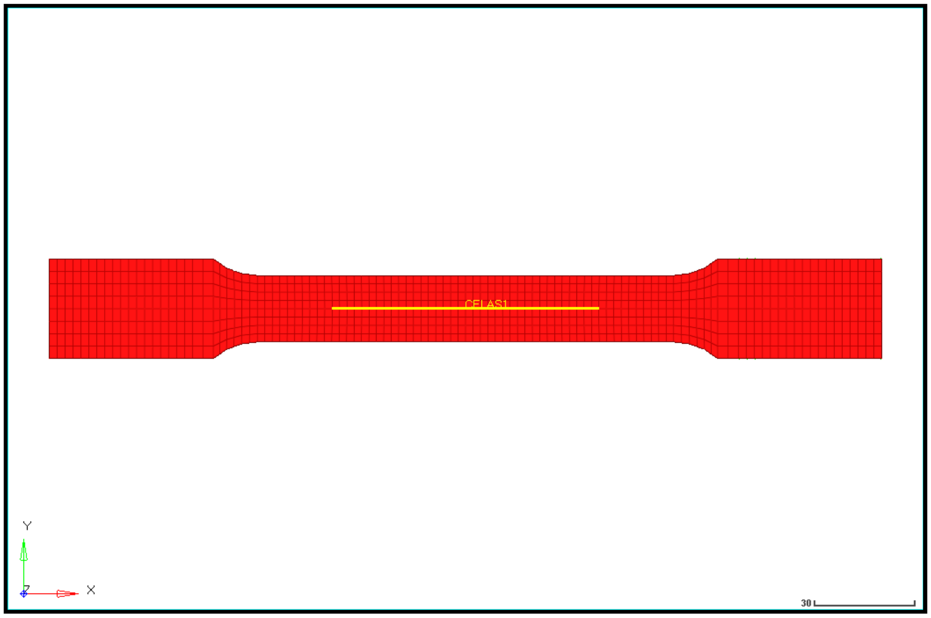
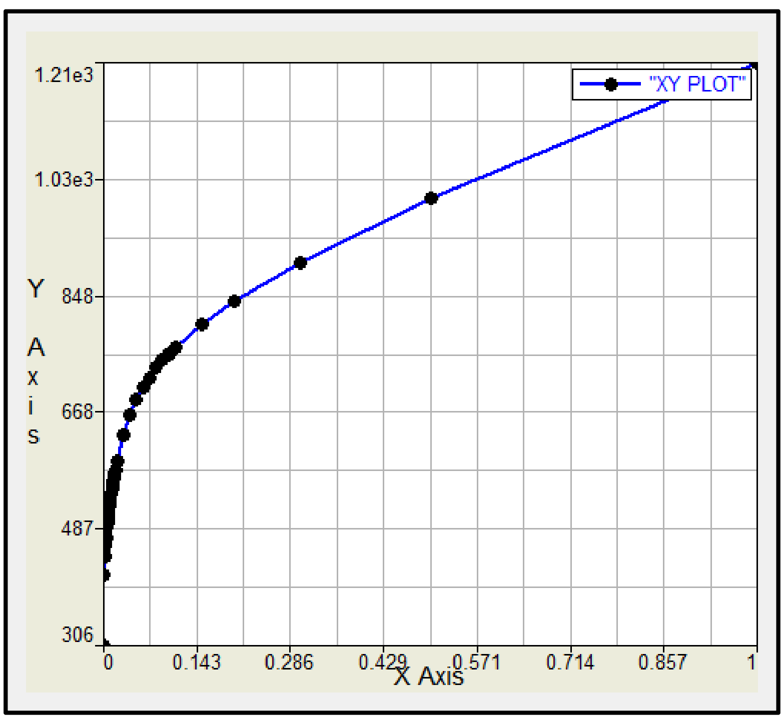
Exercise 3c: Small-Strain Plasticity in a Test Coupon

The purpose of this exercise is to illustrate the setup of a small-strain plasticity enforced displacement loading on a test coupon bar. The model is a solid bar composed of first order hexagonal elements which will be fixed at one end and subjected to an enforced displacement in-plane.





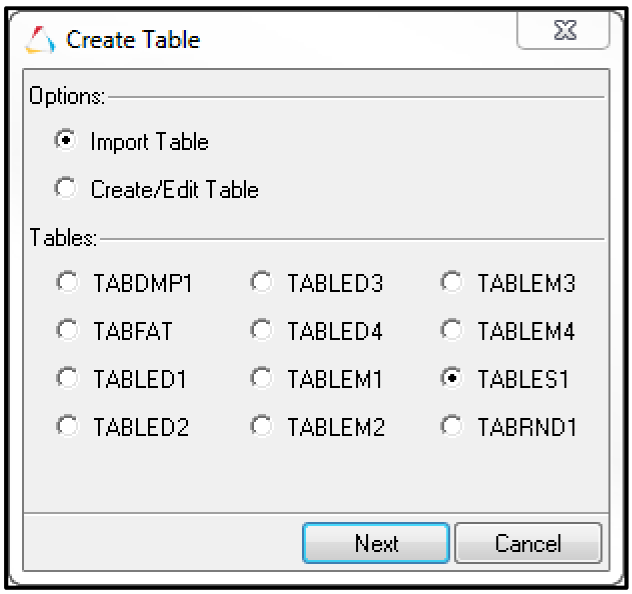
Problem Setup

You should copy the file: `tension_sm_str.fem`

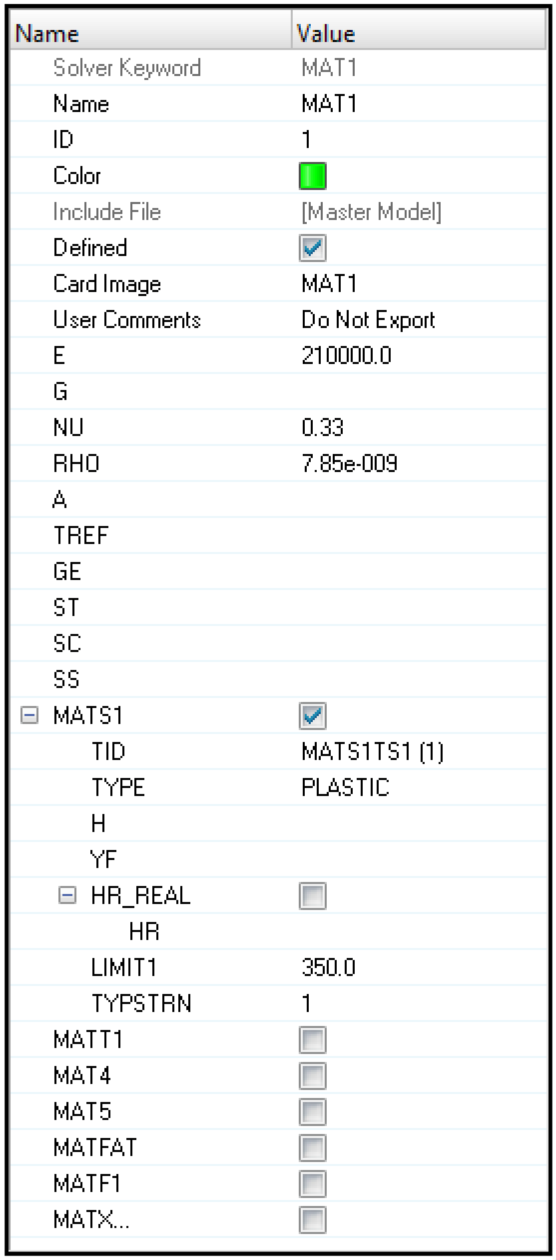
Step 1: Import the model: `tension_sm_str.fem` into HyperMesh Desktop

Step 2: Import the table with the plasticity curve and add a new MAT1 material with plastic material behavior

1. Enable the **Utility** tab by checking the menu drop-down ***View* > *Browsers* > *HyperMesh* > *Utility***.
2. Click on the **Utility** tab in the Browser area.
3. Ensure that the **FEA** button option at the bottom of the Browser area is selected.
4. In the **Utility** tab, select the **TABLE Create** utility. This opens the **Create Table** dialog box.
5. Select ***Import Table*** as the option and click on the table type `TABLES1`.



6. Click ***Next***.
7. Browse for and select the `plasticity.csv` file in the model directory. Name the new table `MATS1TS1` and click ***Apply*** to import the table.
8. Close the **Import TABLES1** window.
9. In the **Model** tab, click on the existing ***MAT1*** material named `MAT1` to load it into the **Entity Editor**.
10. Activate the ***MATS1*** option to put in plasticity parameters and set the card values as shown below:



Step 3: Create a solid property card and apply it to the solid elements

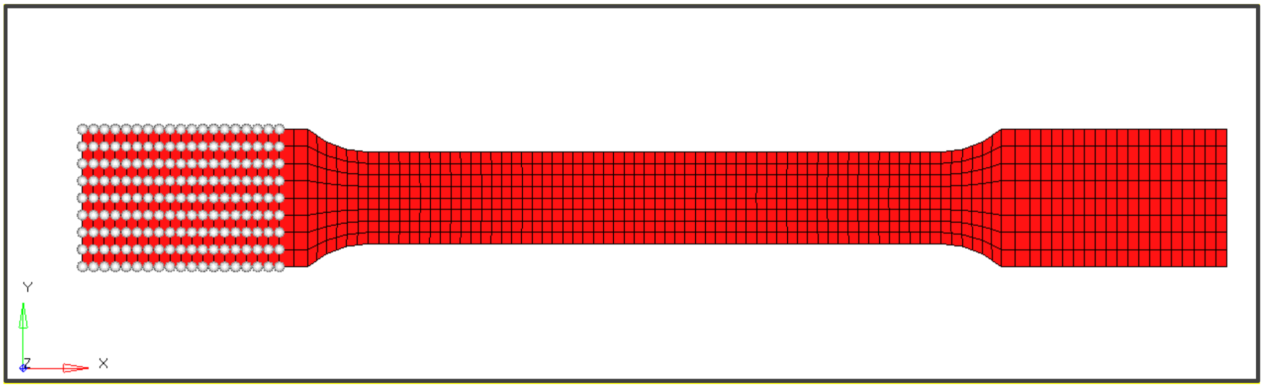
1. Create a new ***PSOLID*** property named `PSOLID` that references the material card `MAT1`.
2. Assign this property to all of the elements in the `TENSIONTEST` component.

Step 4: Create the boundary conditions and loading conditions

1. Create a new load collector named `SPCs`.
2. In the **constraints** panel, select node `14598` ***by id*** and constrain DOFs `1-6`.

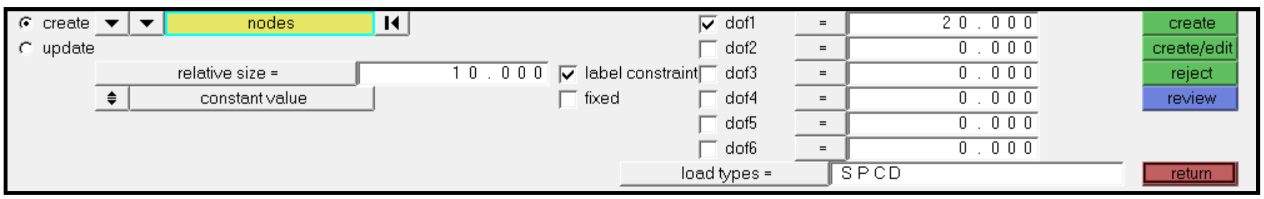
This node is the independent node of the RBE2 and will constrain all nodes connected to that rigid element.

3. Using a window selection, select all of the nodes shown in the image below. Constrain them in DOFs `1-3`.



4. Create a new load collector named `LOAD_20`.

5. In the **constraints** panel, set the ***load type*** to `SPCD` and create a constraint in DOF `1` on node `14598` of ***magnitude*** `20.0`.



6. Create a new `NLPARM` load collector named `NLPARM` with ***NINC*** of `20`.

Step 5: Create the output requests for the measurement set

1. In the **control cards** panel, enter the `GLOBAL_OUTPUT_REQUEST` section.

2. Ensure that the following outputs are requested:

Output Type	Format	Option
`DISPLACEMENT`	`H3D`	`ALL`
`SPCF`	`H3D`	`ALL`
`ELFORCE`	`H3D`	`SID (2)`

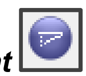
Step 6: Add an NLOUT load collector to enable 100 increments of intermediate step results and set the NLOUT option to the load collector in GLOBAL_CASE_CONTROL

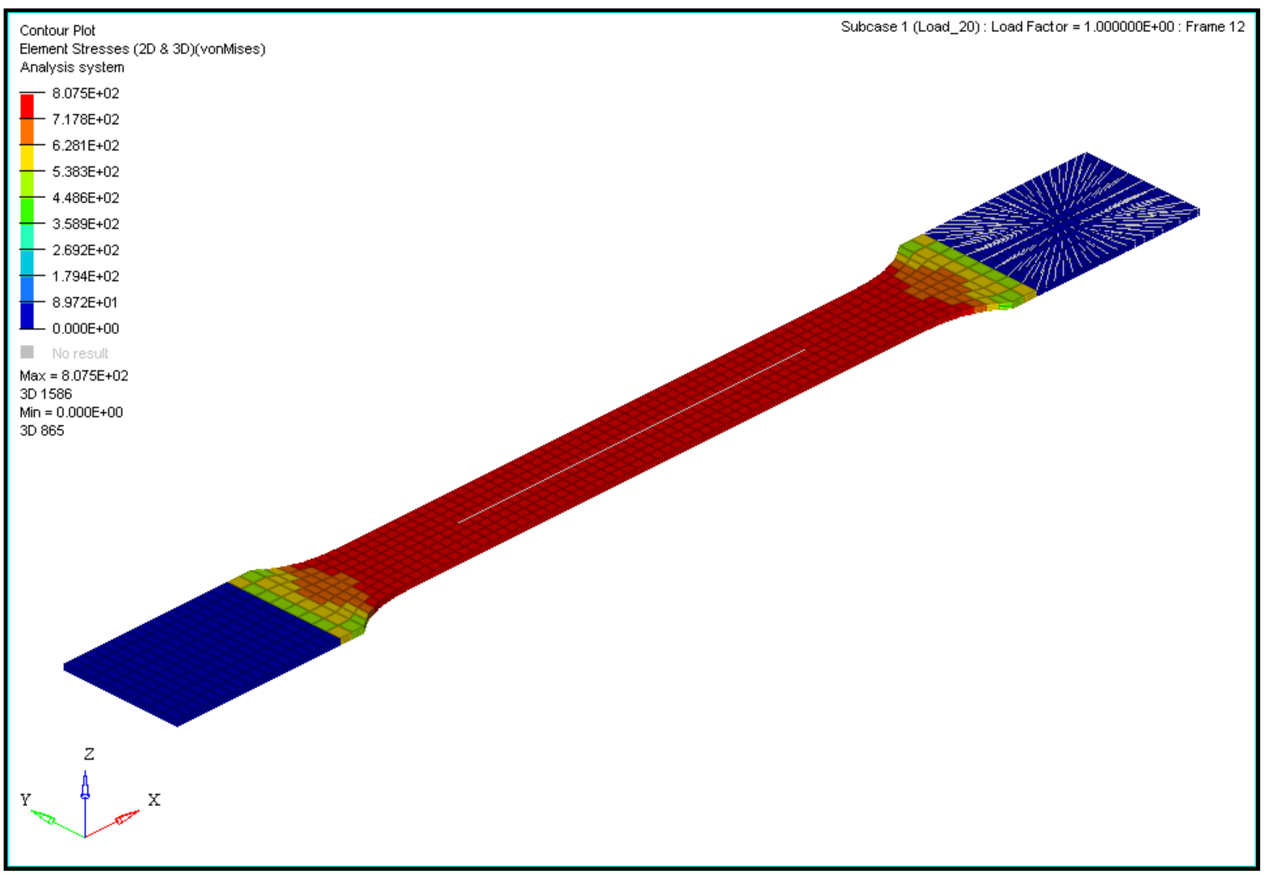
Step 7: Create a new `Non-linear quasi-static analysis` load step named `Load_20`. Set the *SPC* to `SPCs`, the *LOAD* to `LOAD_20`, and the *NLPARM* to `NLPARM`.

Step 8: Run the analysis with 4 CPUs

1. In the ***OptiStruct*** panel under the **Analysis** page, ensure that the ***export options:*** is set to `all`, the ***run options:*** is set to `analysis`. Name the file `tensile_sm_str.fem` and enter the ***options*** `-optskip -out -cpu 4.`
2. Click ***OptiStruct*** to run the nonlinear analysis.

Step 9: Review the results in HyperView

1. When the run has completed, from the ***OptiStruct*** panel, click on the ***HyperView*** button to launch the post-processing report `tensile_sm_str.mvw` which was automatically created from the analysis.
2. Set the animation type to ***Transient***  and click the ***Play*** button  to view the displacement results. Overlay the contours of various results onto the load case animation to view how they change as the enforced displacement scales.



A contour of element stresses of the model at its last enforced displacement (20 units)