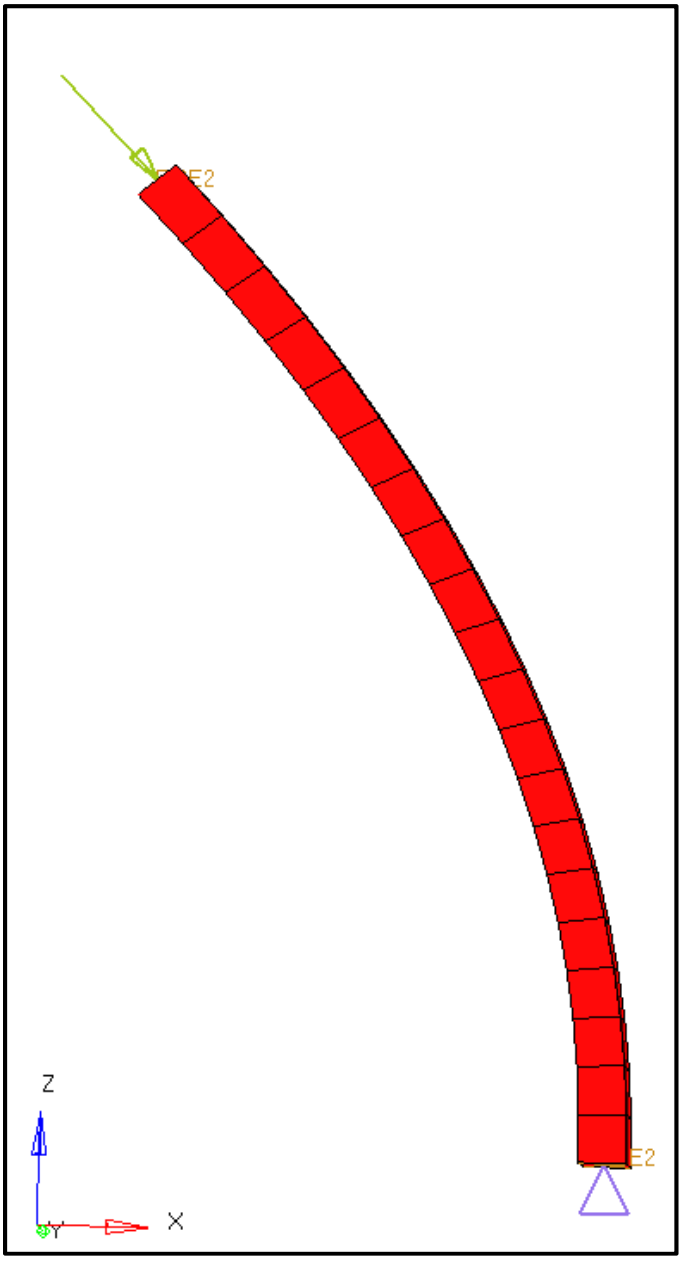
Exercise 4b: Follower Forces on a Curved Rod

This exercise demonstrates the differences between non-follower and follower forces for a curved bar under a load directed into the bar parallel to the axis at one end. The material models, including plasticity effects, the property, the applied force, and the load steps will be created by the user.



Problem Setup

You should copy this file: `cvrod.fem`

Step 1: Import the file `cvrod.fem` into HyperMesh Desktop

Step 2: Create an elastic material with the following values:

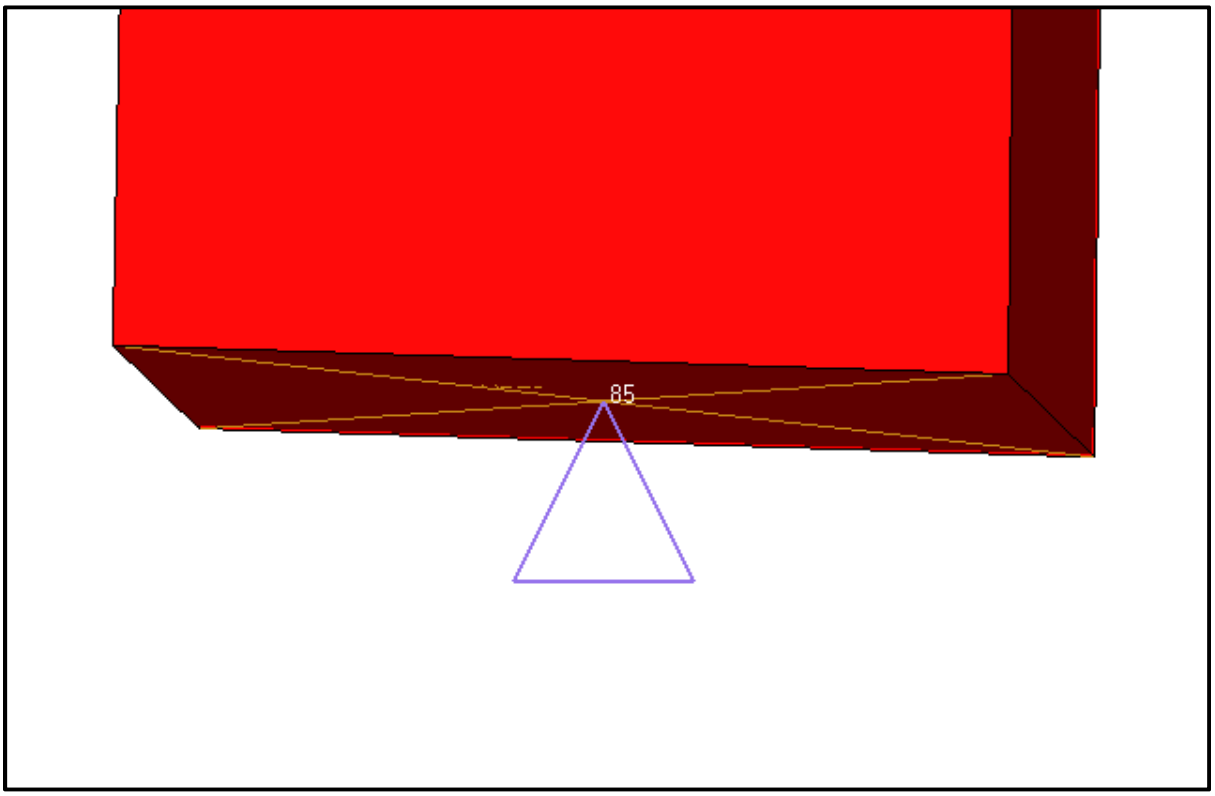
3. ***Name***: Material
4. ***Type***: `MAT1`
5. ***E***: 2.1e5
6. ***NU***: `0.3`
7. ***RHO***: 7.85e-9

Step 3: Create a new PSOLID property named `PSOLID`

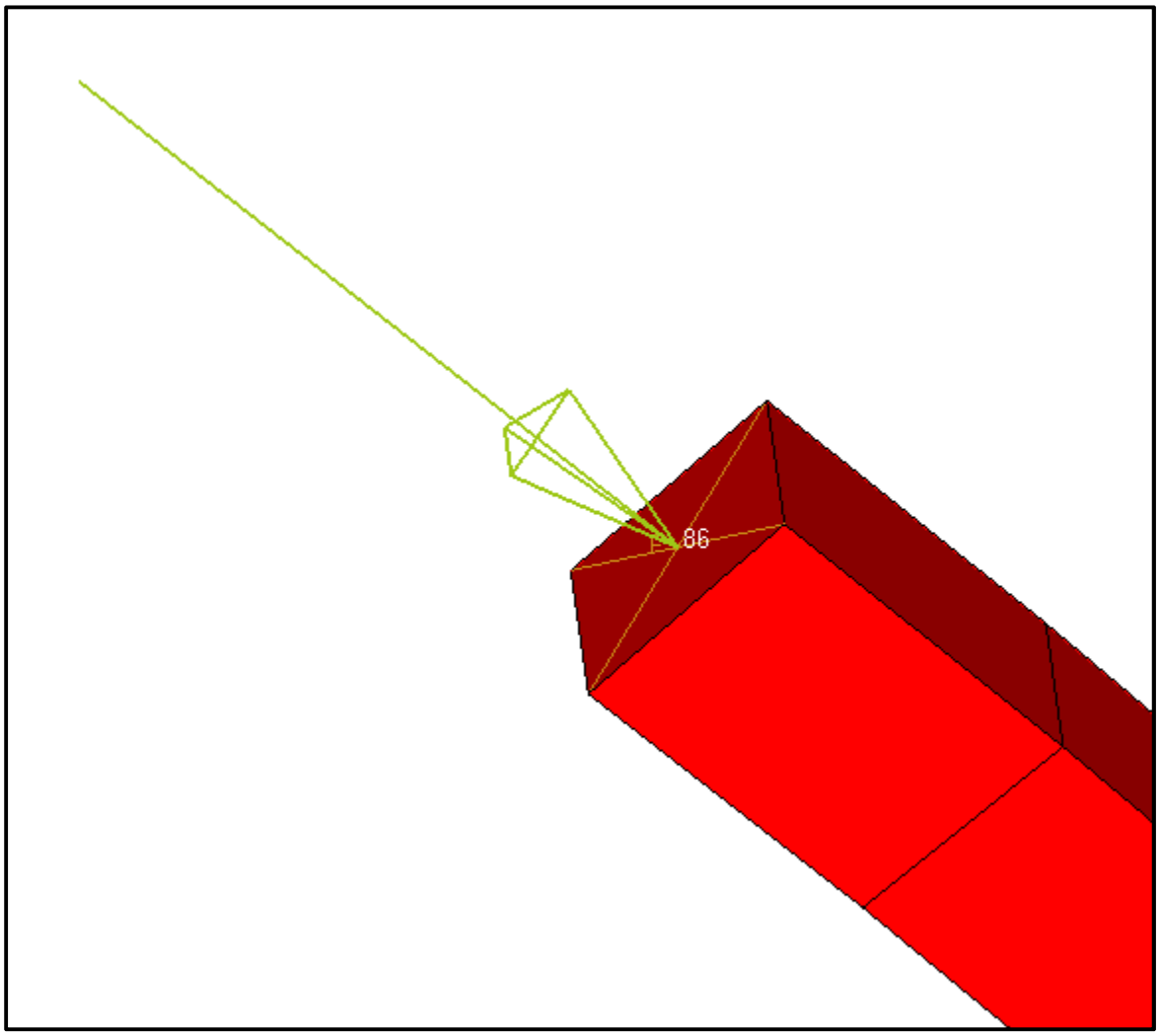
Step 4: Assign the newly created material card to the `PSOLID` property card

Step 5: Assign the `PSOLID` property to the `PSOLID_1` component

Step 6: In a new load collector named SPC, constrain node 85 in DOFs `1-6`



Step 7: In a new load collector named `Load`, create a force on Node 86 with the following components: `{70.253,0.0,-71.165}`



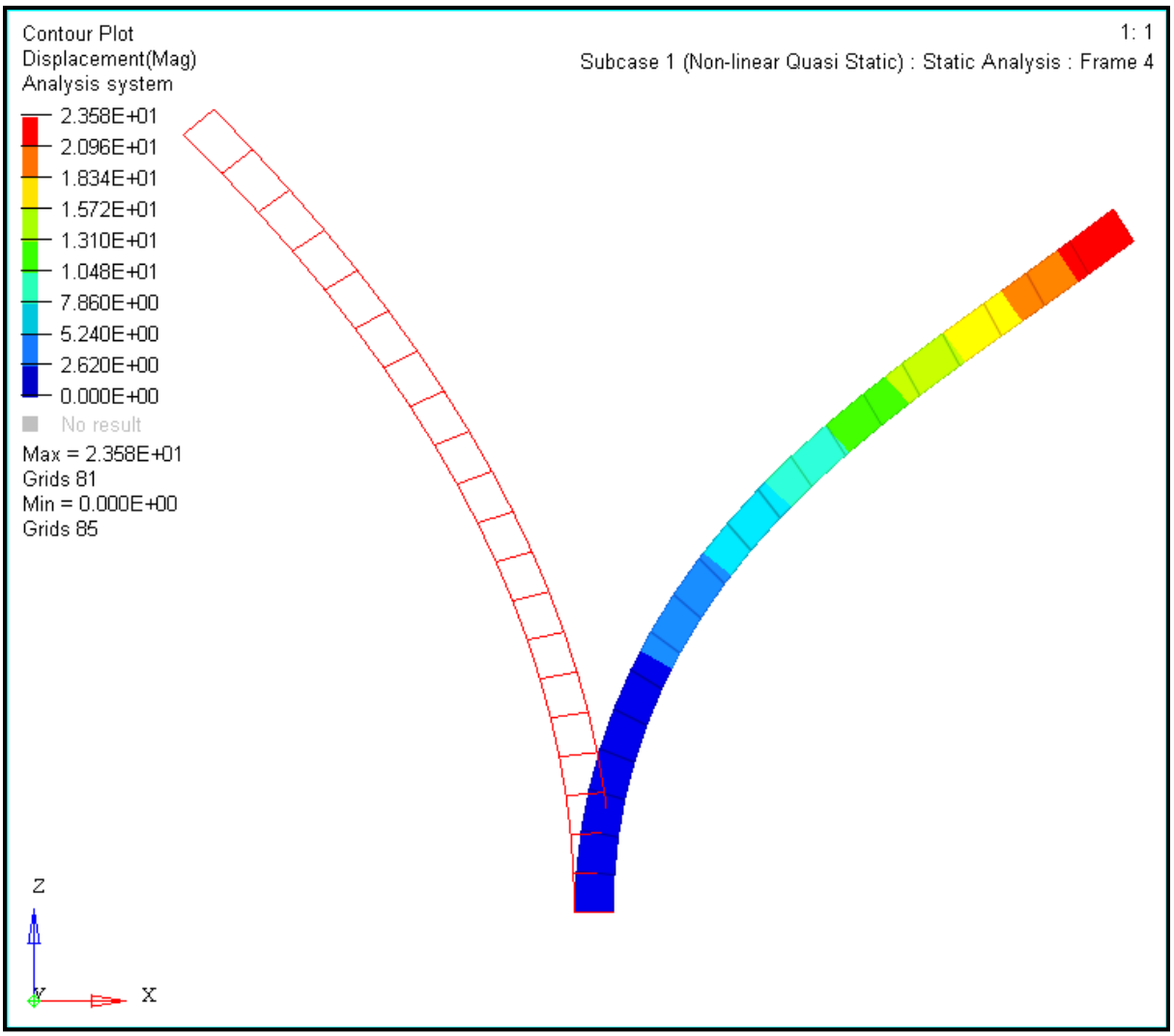
Step 8: Create an NLPARM load collector named `NLPARM` with an *NINC* of `10`

Step 9: Enable hash assembly, large displacement, and the unsymmetric solver using the control cards `PARAM, HASHASSM (YES)` as well as `PARAM, LGDISP (1)` and `PARAM,UNSYMSLV (YES)`.

Step 11: Create a new `Non-linear quasi-static` analysis loadstep with *SPC* set to `SPC`, *LOAD* set to `Load`, and *NLPARM* set to `NLPARM`.

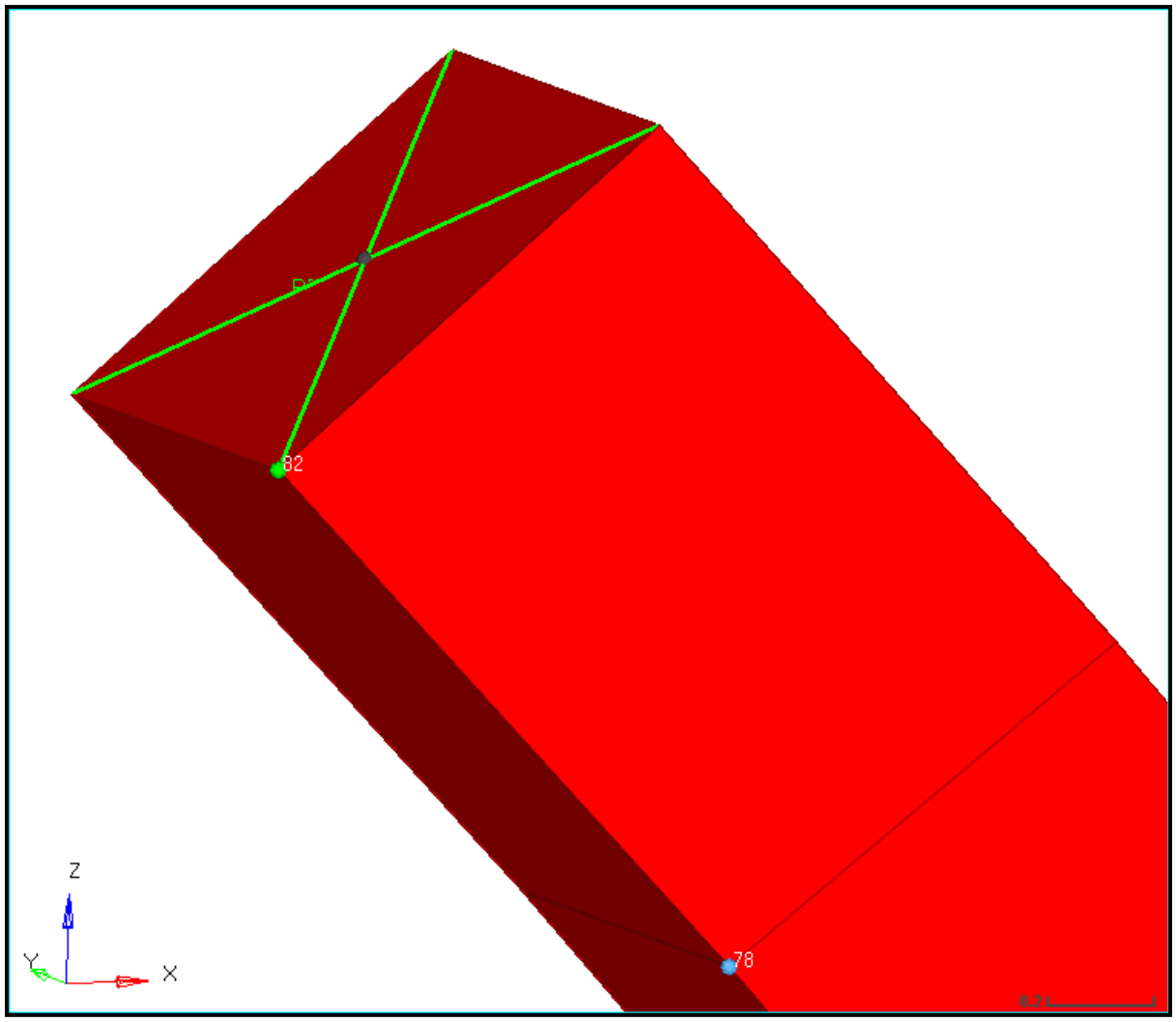
Step 12: Run the analysis as `cvrod_nf.fem`

Step 13: Review the results in HyperView



Step 7: Return to HyperMesh Desktop to replace the force definition and enable follower forces

1. Delete the existing load in the `Load` load collector.
2. Make the `Load` load collector the current load collector.
3. In the **forces** panel, set the ***load types=*** to `FORCE1` and use the nodes entity selector to select node `86`.
4. Enter a ***magnitude=*** of `100` and select node `82` as ***N1*** and node `78` as ***N2*** as shown below:



5. Click ***create*** to create the FORCE1 definition.

   Tip: This definition is identical to the original force in vector and magnitude, and is defined for load collector 2 which is the **Load** load collector, but unlike the previous analysis using the FORCE card, this can make use of the follower force settings.

6. In the control cards, set ***PARAM, FLLWER*** to 1.

Step 8: Save the FE model file as `cvrod_ff.fem` and rerun in OptiStruct

Step 9: Review the results in HyperView