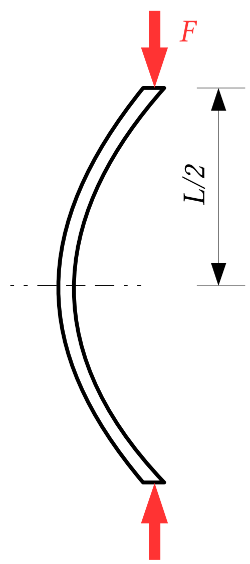
***Buckling of a Bar under Compressive Loading* (20 points)**

Objectives:

1. Use APDL to carry out a multistage simulation including a restart.
2. Learn how the buckling analysis type works in APDL.

Consider a bar under a variable loading force F. Determine the critical buckling load under axial loading.



L = 200 in
$E = 30\times10^6$ psi
$\nu = 0.3$
A = 0.5 in×0.5 in = 0.25 in^2
w = 0.5 in (width)
$F_0 = 1$ lb$_f$
`PLANE182`

Mesh a symmetric model with ten elements and subject it to a symmetry boundary condition at one end and to the loading force at the other (make sure to distribute this load between n nodes). For the model solution, we require several calculations.

First, we use a `STATIC` analysis with all output prepared for a restart. Issue the command `rescontrol,linear,all,1`; look up `rescontrol` in the documentation and summarize what this line does. Also turn on `Sol'n Controls→Sol'n Options→Basic→Calculate prestress effects`. Solve the system once and check your output, including stress intensity `SINT`.

Next, we will use those data to perform a perturbation analysis (this is why we issued the `rescontrol` command). `Restart` the analysis and you should be asked for the load step/substep numbers and an action. Set the action to `perturb` for the linear perturbation analysis; in the next dialog, we want a `buckling` analysis which keeps all mechanical loads. Solve this system. (There are a lot of steps here. If you get really stuck, we have a script prepared which the TA or I can use to provide specific hints.)

Finally, we have to get the buckling mode. In the solution expansion options, use a `subspace iteration` with `1` mode. Expand the mode (`MXPAND`) if the option to do so presents itself—else don't worry about it. Then view the `Results Summary`. The frequency of the mode is the critical buckling load in our force units, lb$_f$.

The critical buckling load reported by Stepan Timoshenko in his classic *Strength of Material* is 38.553 lb_f. How close can you get to this value?

Report

Document your simulation in a 2–5 page report (with figures) containing the sections:

- Problem description (specimen shape, grid, etc.)
- Numerical values (element parameters including any nonstandard keyopts, number of nodes, boundary conditions, etc.)
- Observations of numerical behavior (mesh behavior, etc., as pertinent)
- Discussion of the physics (stress maxima, largest x and y components of stress and displacement, largest displacement, etc.)

Include the following plots in your report as appropriate, with data from each case:

- Mesh of model
- Deformed result with outline of undeformed original (with scale factor noted)
- Contour plot of von Mises stress
- Line plot of von Mises stress along a vertical and a horizontal intersecting line

These plots will largely come from the first `STATIC` simulation, with the later simulations providing the critical buckling load.

The report should be formatted with 1.5 line spacing, 1 inch margins on all sides, and set in 11 point serif font. Figures and tables should be numbered with labels and captions.