

ANSYS - 3D Curved Beam - Problem Specification

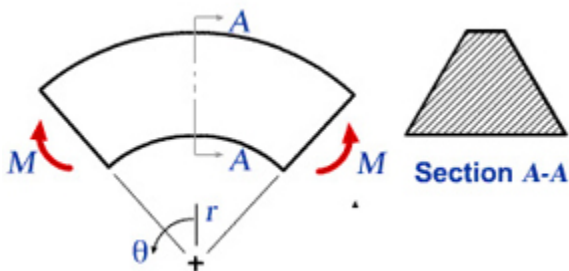
Author: Rajesh Bhaskaran, Cornell University

Problem Specification

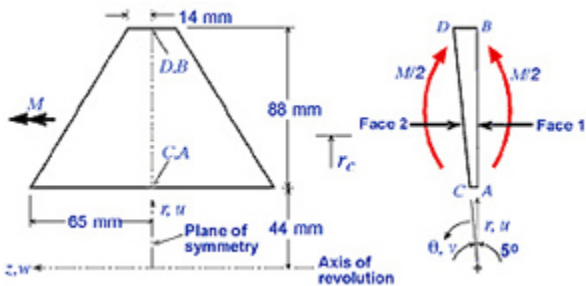
- 1. Start-up and preliminary set-up
- 2. Specify element type and constants
- 3. Specify material properties
- 4. Specify geometry
- 5. Mesh geometry
- 6. Specify boundary conditions
- 7. Solve!
- 8. Postprocess the results
- 9. Validate the results

Problem Specification

The problem considered here is the curved beam of uniform trapezoidal cross-section in example 6.15 of Cook et al. The beam is bent in its own plane by moments M . The problem is not axisymmetric because displacements have circumferential as well as radial and axial components. So we use 3D solid elements rather than axisymmetric elements. The geometry can nevertheless be described in cylindrical coordinates.



We would like to obtain the stresses for the trapezoidal cross-section AA shown above. Stresses in the curved beam do not vary with θ , so we can reduce the model and analyze only a typical slice between two closely spaced radial planes as shown below. The angle between AB and CD is taken to be 5 deg. as suggested by Cook et al.



The bending moment M must be applied indirectly in the reduced model since we don't know *a priori* the circumferential stress distribution it produces on the cross-section. Instead, we'll prescribe *displacements* such that radial plane sections remain plane and a pure moment load acts on the model i.e. no net force acts on it. The moment M can be computed from the stress distribution on the cross-section obtained from FEA. Stresses scale linearly with the applied moment. So the stresses associated with a prescribed moment M_p can be obtained by multiplying the computed stresses by the ratio M_p/M .

The z-constant plane containing A,B,C and D is a symmetry plane. So only half the cross-section needs to be modeled.

Boundary Conditions

The nodal d.o.f. in the radial (u), circumferential (v), and axial (w) directions are constrained as follows:

Face 1	Face 2
$u=0$ at node A	.
$v=0$ at all nodes	$v=0.0001(r_c-r)$ at all nodes
$w=0$ along AB	$w=0$ along CD

All remaining d.o.f. are unrestrained. Setting $u=0$ at A prevents rigid body motion in the r -direction. Setting $v=0$ on face 1 nodes prevents circumferential motion of face 1. Setting $w=0$ on ABCD imposes symmetry about the middle r - plane. The above BC on face 2 nodes causes face 2 to remain plane as it rotates about a z -parallel axis at $r=r_c$. The factor 0.0001 is arbitrarily chosen. At the outset, the appropriate value of r_c is not known. The right value of r_c will give a pure bending load so that the radial reaction R_A at node A is zero. Two preliminary FE analysis with guess values of $r_c=60\text{mm}$ and $r_c=70\text{mm}$ were done. The respective R_A values turn out to be 2001N and 357N. By linear extrapolation, $R_A=0$ when $r_c=72.2\text{mm}$. So we'll use $r_c=72.2\text{mm}$ in our analysis. (Since this is a pedagogical exercise, I've decided to be nice and give you the r_c value to use. In the real world, you'd of course have to figure it out yourself).

[Go to Step 1: Start-up and preliminary set-up](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)