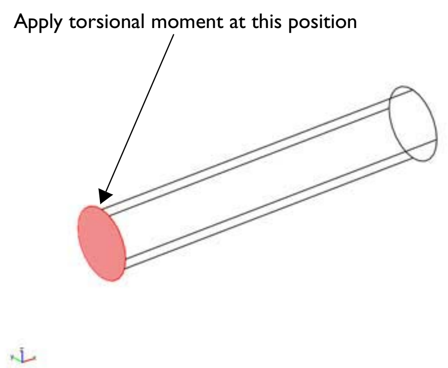
Moments in the Continuum Application Modes

The continuum application modes do not have rotational degrees of freedom, which makes the specification of moments somewhat cumbersome. To specify moments, you can apply a stress distribution which corresponds to the moment.

EXAMPLE: TORSIONAL MOMENT ON A CYLINDER

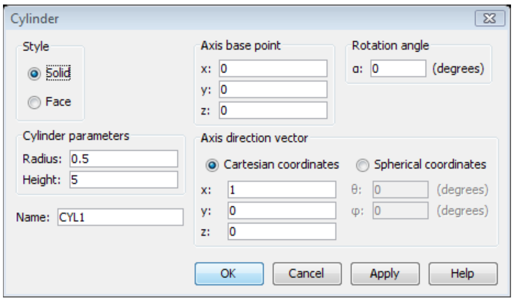
The following steps show how to apply a torsional moment at one end of a cylindrical axle in the following figure.



1. In the **Model Navigator** select **3D** from the **Space dimension** list.
2. In the list of application modes select **Structural Mechanics Module> Solid, Stress-Strain>Static analysis**; then click **OK**.

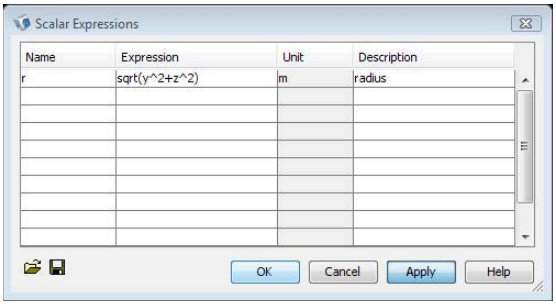
   Continue by creating a cylinder:
3. In the **Draw** toolbar click the **Cylinder** button.

4 In the dialog box that opens, enter `0.5` as the **Radius**, `5` as the **Height**, and define the axis of the cylinder along the *x*-axis by entering `1` in the **x** edit field and `0` in the **y** and **z** edit fields in the **Axis direction vector** area. Click **OK**.



Next define the radial location as a scalar expression, which you can later use in the load expression.

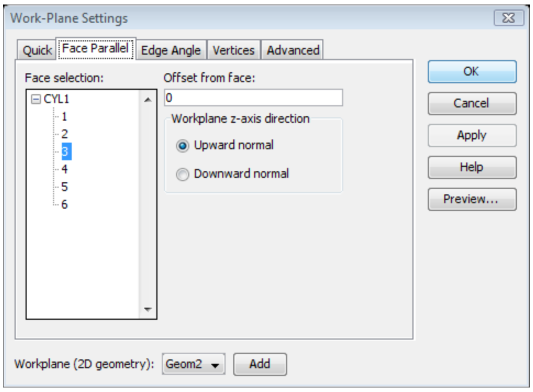
5 Choose **Options>Expressions>Scalar Expressions**. Enter `r` in the **Name** column and `sqrt(y^2+z^2)` in the **Expression** column. Click **OK**.



Create a work plane that you can use to define a cylindrical coordinate system.

6 Select **Draw>Work-Plane Settings**.

7 On the **Face Parallel** page in the **Work-Plane Settings** dialog box select Face 3 from the **Face selection** list. Click **OK**.



8 A new geometry appears with the work plane that you just created. Switch back to the geometry containing the cylinder by clicking the Geom1 page in the Drawing area.

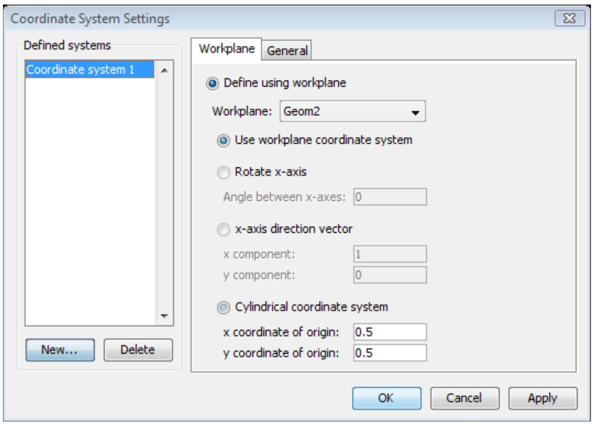
Continue by defining a cylindrical coordinate system with the help of the work plane you just created.

9 Select **Options>Coordinate Systems**. Click **New** in the **Coordinate System Settings** dialog box.

10 Click **OK** in the **New Coordinate System** dialog box.

11 Back in the **Coordinate System Settings** dialog box make sure that **Define using work plane** is selected as well as **Geom2** in the **Work plane** list.

12 Click the **Cylindrical coordinate system** button. Enter `0.5` for both the *x*- and *y*-coordinates of the origin, because these are the coordinates of the center of the circular face in the coordinate system of the work plane. Click **OK**.



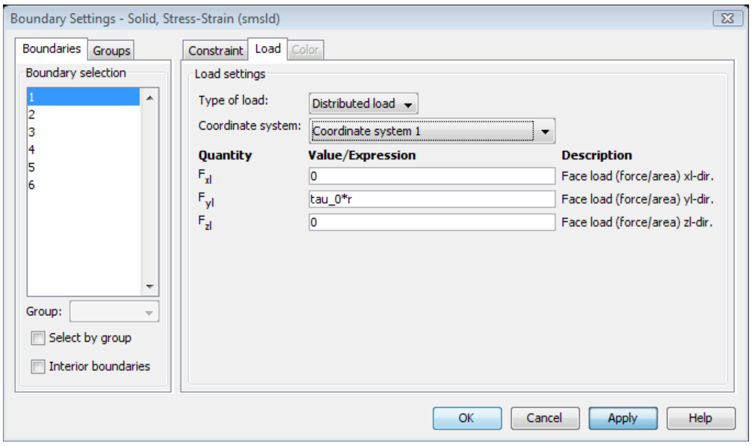
Now you can define a shear stress distribution in the tangential direction, which is zero at the center and reaches its maximum at the surface of the axle.

13 Select **Physics>Boundary Settings** to open the **Boundary Settings** dialog box.

14 Select Boundary 1 from the **Boundary selection** list.

15 On the **Load** page select **Coordinate system 1** (the one you just created) from the **Coordinate system** list.

16 Enter `tau_0*r` in the **F_{yl}** edit field. Click **OK**.

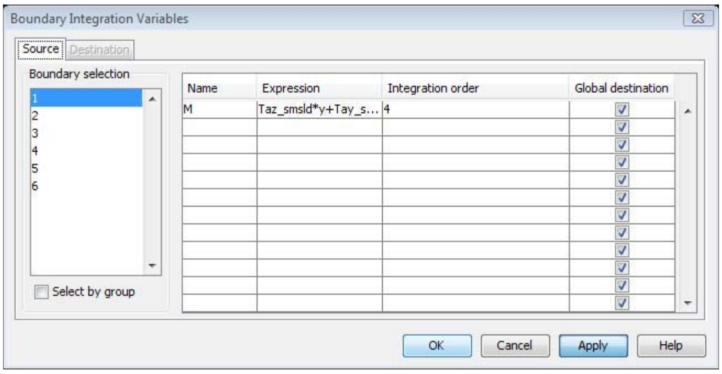


Because the integral of the stress distribution over the boundary must equal the moment to satisfy equilibrium, you can easily specify a value for the moment based on the stress distribution.

17 Define the boundary integral for the moment. Select **Options>Integration Coupling Variables>Boundary Variables**.

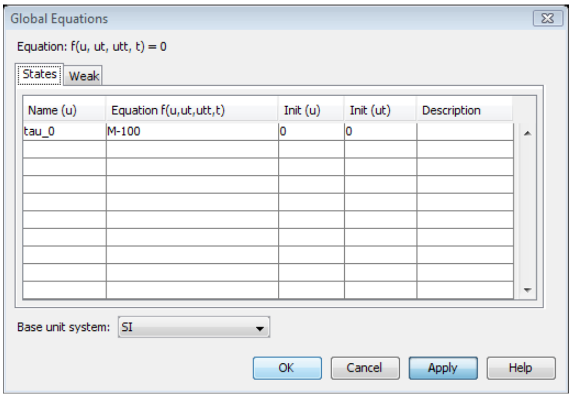
18 In the dialog box that opens select Boundary 1, which is the one where you have defined the load.

19 Enter `M` in the **Name** column and `Taz_smsld*y-Tay_smsld*z` in the **Expression** column. Click **OK**.



20 In this last step you specify a value for the moment through defining an equilibrium equation. Choose **Physics>Global Equations** to open the **Global Equations** dialog box.

21. Enter `tau_0` in the **Name** column and `M-100` in the **Equation** column. The value 100 is the magnitude of the applied moment. This entry defined the equation $M - 100 = 0$, that is, $M = 100$.



22. Constrain the other end face of the cylinder by fixing it before solving the model.

    After solving the model you can visualize the moment on the cylinder by an arrow plot of the global force on the boundaries.
23. Click the **Plot Parameters** button on the Main toolbar.
24. On the **General** page clear the **Slice** check box and select the **Arrow** check box.
25. Click the **Arrow** tab to switch to the **Arrow** page.
26. From the **Plot arrows on** list box select **Boundaries**.
27. From the **Predefined quantities** list box select **Global force**.
28. From the **Arrow type** list box select **3D arrow**.
29. From the **Arrow length** list box select **Normalized**.
30. Click **OK**.