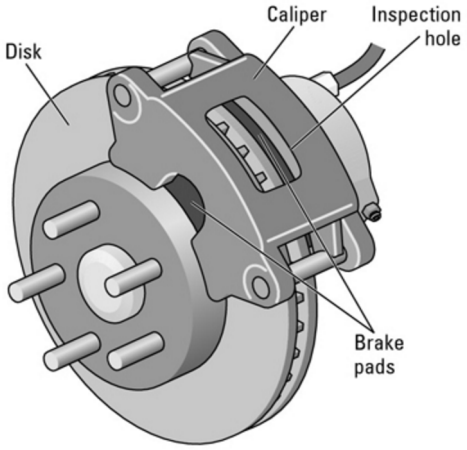
Disk Brake (25 pts)



Vehicles have used disk brakes for decades due to their superior thermal performance (they can be cooled more readily than drum-style brakes). The disk or rotor is often simply of cast iron. You will use an Abaqus guided example to simulate the effect of heating on this brake and the resulting thermal distribution due to friction as the brake pads press upon a rotating disk. (The solution is steady-state, not transient, however.) You may acquire the files used in this simulation from the course web site.

Axisymmetric Model

```
discbrake_std_cax4t.inp
discbrake_std_cax4t.f
```

You should closely examine the input file (either using CAE or a text editor) and report the geometry (including consequences of applying axisymmetry to this problem as compared to the real disk brake model on this page), the boundary conditions and loads, the mesh and element type, and any other pertinent facts about the model. You may find it profitable to examine the associated Fortran user-defined function which defines the friction function; you will also need to load it in Abaqus/CAE to submit the job. After simulating the piece, you should plot and report the stress, the temperature distribution, and any deformation.

3D Model

```
discbrake_3d.inp
```

This model lays out a more fully realized (and hence more computationally expensive) simulation. You should similarly examine and report the geometry, the loads, the elements, etc. After simulation, plot and report the same results as before in both radial and axial cross-section (*i.e.*, cutting perpendicularly to the rotor and cutting in the same plane as the rotor at the midpoint).

You will document your simulations in a 10–12 page report (with figures) containing the sections:

a) Problem description (shape, grid, etc.)
b) Details of the simulation and numerical parameters
c) Observations of numerical behavior (residual convergence rate, mesh behavior, etc.)
d) Discussion of the resulting physics as discussed above, including plots

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

Reference

Abaqus Example Problems Manual, 5.1.1 Thermal–stress analysis of a disk brake.

Addendum

The user-defined friction model in the two-dimensional model doesn't work properly on the Windows installation of Abaqus since there is no Fortran compiler available. You need to update the interaction property `INT` in the *Interaction* module. It should match the settings for the 3D model: