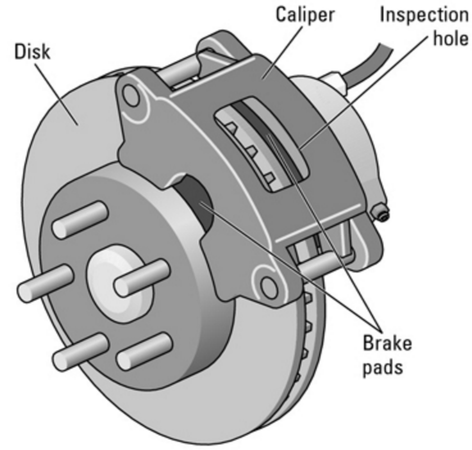
Disk Brake



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.

Three cases will be considered:

Axisymmetric Model with UDF

```
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.

Axisymmetric Model without UDF

```
discbrake_std_cax4t.inp
```

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, given in the appendix to this assignment. How does this impact the outcome? What physics are missing? Are there hotspots or potential material issues that result?

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).

As UDFs are not supported in the student version of Abaqus nor on the EWS Windows machines, you have two options for completing the axisymmetric model with UDF:

- *EWS Linux*: you will need to load the modules `abaqus` and `intel-composer` first, and manually locate the `ifort` command. The Fortran source file should be located in the same directory as the input file. On the command line, specify the user keyword; *i.e.*,

  ```
  $ abaqus job=JOB.INP user=UDF.F
  ```

- *SEAGrid*: Create an account first and download the application. Follow the steps as given in class, but include the Fortran source file as well. (Use Comet, not Gordon or another machine.)

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

a) Problem description (shape, grid, etc.)—interpret the input files
b) Details of simulation and numerical parameters
c) Observations of numerical behavior (mesh artifacts, etc.)
d) Discussion of resulting physics as discussed above, including labeled plots

The report should be formatted with 1.5-line spacing, 1-inch margins on all sides, and set in 10.5–12-point serif font. All figures and tables (if any) should be numbered and have labels and captions. Cheng (2006) exemplified how to document numerical simulations for this type of problem. Submit an electronic copy to the TA by 5:00 p.m. on the due date.

Reference

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

Appendix

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: