history of load as described above, but tells ABAQUS how to apply the load during this time step. If you specify AMPLITUDE=RAMP, the load is applied smoothly, while if you say AMPLITUDE=STEP, the load is applied at once.

```
*STATIC
1.0,1.0
```

We tell ABAQUS that this is a quasi--static analysis. The stress fields are in static equilibrium throughout the history of load. To choose a static analysis, use the *STATIC key word. The first number on the following line suggests an initial value for the time increment that ABAQUS should take while calculating the deformation in this step. Since we expect the plate to deform elastically in this step, it makes sense to take a time increment equal to the step size -- ABAQUS should be able to go straight to the solution at the end of the step, without taking little steps to get there. The second number specifies the time *interval* for this load step. The step starts at time $t=0$ and ends at time $t=1$, so the time interval is 1.

Two additional optional parameters are also available - the third number specifies a minimum value for the time increment, and the last number specifies a maximum value. We have not used these parameters here.

```
*DLOAD, AMPLITUDE=HIST
EDGE, P2, -82.E06
```

Now we specify the loading applied to the plate. We select distributed loads(pressure) acting on face 2 (P2) of all the elements in set EDGE. We take the load magnitude to be 82 MPA, so that when it is scaled by HIST, the stress reaches 82MPA at time $t=1$. Note that, by definition, *DLOAD defines pressure (i.e. compressive normal stress) to be positive, so we apply tensile loading by making the pressure negative.

```
*EL FILE, POSITION=AVERAGED AT NODES
S,E
```

Next, we specify what variables we'd like printed to the history file for post processing. We are going to print all stress components (S) and strain components (E). These variables are normally only computed at element integration points, so we use the *EL FILE keyword to ask ABAQUS to print them to a file. However, we are really interested in values of stress and strain at the nodes in this case, so we set the POSITION=AVERAGED AT NODES flag to have ABAQUS calculate the varibles at nodes.