



UNIVERSITY OF
CAMBRIDGE



DEPARTMENT OF
ENGINEERING

The ABAQUS FAQ

THE ABAQUS FAQ	0
1. GENERAL QUESTIONS	2
2. JOBS	5
3. ELEMENTS	10
4. ABAQUS - MESH	14
5. ABAQUS - MATERIALS.....	16
6. ABAQUS - BOUNDARY CONDITIONS	21
7. LOADING.....	25
8. ABAQUS - PROCEDURES.....	34
9. ABAQUS - ANALYSIS.....	35
10. OUTPUT	41
11. ABAQUS/POST - GENERAL	47
12. ABAQUS/POST - CONTOURS.....	52
13. ABAQUS/POST - MESH PLOTS.....	55
14. ABAQUS/POST - XY PLOTS	58
15. ABAQUS/POST - VECTOR PLOTS.....	61
16. ABAQUS/POST - PATH PLOTS	62
17. ABAQUS/POST - VIEWS.....	63
18. ABAQUS/POST - HARDCOPY	64
19. ABAQUS/PLOT.....	65
20. PRE PROCESSING USING PATRAN.....	66
21. POST PROCESSING USING PATRAN.....	67
22. PRE PROCESSING USING FEMGV.....	68
23. POST PROCESSING USING FEMGV.....	69
24. ABAQUS - ERRORS	72

1. General Questions

Q1.1 : How much disk space do I need?

Minimum of 10 MBytes to run simple examples if using PATRAN for Pre or Post processing. Probably about 20 MBytes for medium sized problems.

The amount of disk space required depends on the numbers of nodes/elements present in the mesh and the output frequency requested for outputting data to the *.fil, *.dat and *.res files.

The following ABAQUS files are created in the user's home directory.

*.dat, *.fil, *.res, *.msg, *.sta, *.log, *.job, *.023 (deleted at the completion of the ABAQUS job).

Q1.2 : What type of analysis can I do?

The procedures available in ABAQUS are listed below (in alphabetical order) :

- BUCKLE
 - COUPLED TEMPERATURE-DISPLACEMENT (steady state and transient)
 - COUPLED THERMAL-ELECTRICAL (steady state and transient)
 - DYNAMIC
 - FREQUENCY
 - GEOSTATIC
 - HEAT TRANSFER (steady state and transient)
 - MASS DIFFUSION (steady state and transient)
 - MODAL DYNAMIC
 - RANDOM RESPONSE
 - RESPONSE SPECTRUM
 - SOILS, CONSOLIDATION
 - SOILS, STEADY STATE
 - STATIC
 - STEADY STATE DYNAMICS
 - VISCO
-

Q1.3 : What output options (hard copy) are there?

With ABAQUS 5.7 any size plots (including A3 and A4) can be obtained.

If using PATRAN or FEMGV hard copies of any size (including A4 and A3) can be obtained.

Q1.4 : What pre and post-processing programs are available other than ABAQUS/Post?

PATRAN and FEMGV are available in the teaching system.

Both these programs read the results file *.fil for post processing.

If you are post processing using FEMGV and have written results at element Gauss points then you need to write the gauss point co-ordinates as well. Include the following lines in the ABAQUS input file :

```
*EL FILE, POSITION=INTEGRATION POINTS  
S  
E  
COORD
```

Otherwise it will not be possible to use the FEMGV program to post process the element gauss point results ie FEMGV would not know the location of the gauss points.

Q1.5 : Is ABAQUS/Pre available?

No. ABAQUS/Pre is not available in the teaching system. However PATRAN which provides the same functionality and which has the same look and feel of ABAQUS/Pre is available in the teaching system. Type **patran** to run the PATRAN program.

Q1.6 : Is ABAQUS/Explicit available?

ABAQUS/Explicit is only licenced on **tw900**. You need to rlogin to tw900 to run this version of ABAQUS. ABAQUS/Standard is also available on tw900.

Q1.7 : What on-line documentation are available and how do I access it?

Type **abaqus57 doc** in the CUED teaching system to access the full set of the ABAQUS Users' manual (Volumes I, II and III).

At present this is the only on-line documentation available except for [ABAQUS Release Notes](#) (100 pages) and [ABAQUS Site Guide](#) (80 pages).

Q1.8 : What units are used in ABAQUS?

There is no inherent set of units used in ABAQUS. It is up to the user to decide on a consistent set of units and use that units. Typical sets of units :

	1	2
Length	- metres	mm
Force	- Newtons	Newtons
Time	- second	second
Mass	- Kg	tonne (**)
Density	- Kg/m ³	tonne/mm ³
Stress	- N/m ²	N/mm ² (= MPa)
Young's Modulus	- N/m ²	N/mm ² (= MPa)

** 1 tonne = 1000 kilograms

Decide on the units before you start preparing your data. This is critical. If you start typing in the nodal co-ordinates that means you have already decided on what units to use for the **Length parameter. This only leaves the choice for the units of **Force**.**

It is not a good idea to choose mm for length, Newton for force and then specify the Young's modulus in KN/m².

2. Jobs

Q2.1 : How do I run small jobs?

Use the following command

```
abaqus job=job-id interactive
```

Example : abaqus job=plate

Q2.2 : How do I go about running many small jobs?

Create a batch file (say **aba.run**) with one line per analysis as shown below :

```
abaqus job=analysis-a interactive
abaqus job=analysis-b interactive
abaqus job=analysis-c interactive
abaqus job=analysis-d interactive
```

Then make the file an executable using the following unix command :

```
chmod u+x aba.run
```

Then type **aba.run** to execute the ABAQUS jobs one at a time while you are logged ON. This is only suitable for small jobs which only take a few minutes to run. These jobs will run one at a time and in sequence.

This is preferable to submitting all the jobs at the same time (for example typing the above commands directly at the terminal without the **interactive** parameter). This will put a strain on the server and its resources and inconvenience the other users as well.

For medium to large jobs use the **batch** command available in the CUED teaching system.

Q2.3 : How do I run large jobs using batch?

In the CUED teaching system use the batch command. To run the job on tw500 or tw900 servers use :

```
batch -QX -mbao "abaqus job=job-id interactive"
```

To run a job specifically on the tw900 server use :

```
batch -QN -mbao "abaqus job=job-id interactive"
```

Example :

```
batch -QN -mbao "abaqus job=plate interactive"
```

The progress of the submitted job can be monitored using the **batchq** command. Use the **batchrm** command to delete any batch jobs you had submitted before these are run, if you change your mind. See the man pages on **batchq**, **batchrm** for more details.

Example : Type **man batchq**.

Q2.4 : How do I run ABAQUS/Post?

Using the following command :

```
abaqus post job=job-id
```

Example : abaqus post job=cantilever

Q2.5 : How do I run ABAQUS/Plot?

Using the following command :

```
abaqus plot job=job-id device=cps or hgl or x11
```

Use **device=x11** to view the plots on the screen. Use one of the other options (cps for colour postscript or hgl for hpgl) to create a hard copy of the plot.

Example : abaqus post device=cps job=cantilever

Q2.6 : How do I get copies of the ABAQUS examples input data files (*.inp)?

The ABAQUS datafiles used in the examples manual can be found in the **/export/abaqus/samples/exastd** directory. Similarly the datafiles in the verification manual can be found in **/export/abaqus/samples/verstd** directory. These files will have the extension name **inp**. All file names are in lowercase. use the **cp** command to copy the relevant file.

```
Example : cp /export/abaqus/samples/exastd/1010101.inp .
```

This will copy the 1010101.inp file to the current directory.

If these directories don't exist then can use the **abaqus fetch** command.

Example : abaqus fetch job=1010101

Q2.7 : How do I run a ABAQUS job which uses a user subroutine?

Using the following command :

```
abaqus job=job-id
```

As can be seen this is no different from running a standard abaqus job. The user subroutine itself can be embedded in the abaqus input file. Here it is illustrated with the **umat** subroutine.

```
<....part of the abaqus input file ....>
      .....
      .....
*END STEP
*USER SUBROUTINES
  SUBROUTINE UMAT(.....)
      .....
```

```
.....  
END  
.....
```

Alternatively the user subroutine can be in a separate file (say **my_material.f**) and the **INPUT** parameter is set to that file name.

```
<....part of the abaqus input file ....>  
.....  
.....  
*USER SUBROUTINES, INPUT=my_material.f  
.....  
.....
```

Q2.8 : How do I run a user written post processing program which accesses the *.fil file?

Using the following command :

```
abaqus make job=job-id user=name-of-file  
Example : abaqus make job=cantilever user=displ
```

This will compile the user program in a file called displ.f and then create an executable called cantilever.x. Type cantilever.x to run this program.

Q2.9 : How do I find out about the different execution procedures that are available with ABAQUS?

Type abaqus help and this will list all the abaqus execution procedures. These are listed below :

Execution Procedure for ABAQUS/Standard and ABAQUS/Explicit

```
abaqus job=job-name [ analysis | datacheck | continue | help | recover |  
                    convert={restart|select|all} |  
                    information={environment|local|memory|release|status} ]  
                    [ input=input-file ] [ user=source-file ]  
                    [ oldjob=oldjob-name ] [ fil={append|new} ]  
                    [ globalmodel=results file-name ] [ double ]  
                    [ memory=memory-size ] [ buffer=buffer-size ]  
                    [ interactive | background | queue=queue-name ]  
                    [ cpus=number-of-cpus ] [ scratch=scratch-dir ]  
                    [ subcomplex=subcomplex-name ]
```

Note: subcomplex is only valid on the Convex Exemplar

Execution Procedure for ABAQUS/Post

```
abaqus post [ job=job-name ]  
            [ restart=restart-name ] [ input=input-file ]  
            [ device=device-name ] [ display=display-name ]  
            [ geometry=widthXheight+xpos+ypos ]  
            [ memory=memory-size ] [ buffer=buffer-size ]
```

Execution Procedure for ABAQUS/Abares

```
abaqus abares job=job-name [ restart=restart-name ]  
    [ beginstep=step-number ] [ endstep=step-number ]  
    [ increment={all|endstep|final|none|integer-list} ]
```

Execution Procedure for ABAQUS/Plot

```
abaqus plot job=job-name [ input=input-file ] [ device=device-name ]  
    [ frame={all|integer-list} ] [ options=options-file ]
```

Execution Procedure for ASCII translation of results (.fil) files

```
abaqus ascfil job=job-name [ input=input-file ]
```

Execution Procedure for on-line documentation

```
abaqus doc
```

Execution Procedure for ABAQUS/Append

```
abaqus append job=job-name oldjob=oldjob-name input=input-file
```

Execution Procedure for ABAQUS/Fetch

```
abaqus fetch job=job-name [ input=input-file ]
```

Execution Procedure for ABAQUS/Findkeyword

```
abaqus findkeyword [ job=job-name ] [ maximum=maximum-matches ]
```

Execution Procedure for ABAQUS/Make

```
abaqus make job=job-name [ user={source-file|object-file} ]
```

Q2.10 : How do I find what the current settings are for the environment variables?

Type `abaqus info=environment` and this will list all the current setting of the ABAQUS environmental variables.

memory, local, release, status are other options on which you can get more information on.

Q2.11 : How do I change the current settings of the environment variables?

Create a file called `abaqus.env` in the directory from which ABAQUS is run

which contains lines of environment variables you want to change set equal to new values. However make sure that the computer on which you are running ABAQUS can support the changes. For example you can increase the memory used by ABAQUS. But you cannot increase this beyond what is available in the computer.

Example : Include the following lines in the abaqus.env file to increase the size of post_buffer and the post_memory used by ABAQUS/Post.

```
post_buffer="1000000"  
post_memory="3000000"
```

Q2.12 : Can I run long jobs on the twgs?

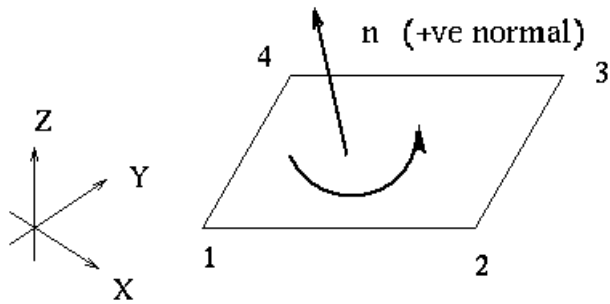
No. is is not a good idea to use the twgs to run long ABAQUS jobs. It is better to use the batch system on one of the faster servers (example : tw900) for long jobs.

Also because of the limited resources (memory, swap space, scratch disk space) large jobs also cannot be run on the twgs.

3. Elements

Q3.1 : How do I find the positive normal of a shell element?

For shells the positive normal is given by the right-hand rule going around the nodes of the element in the order they are given in the element-nodal connectivity data line (datalines which follow the keyword *ELEMENT line).



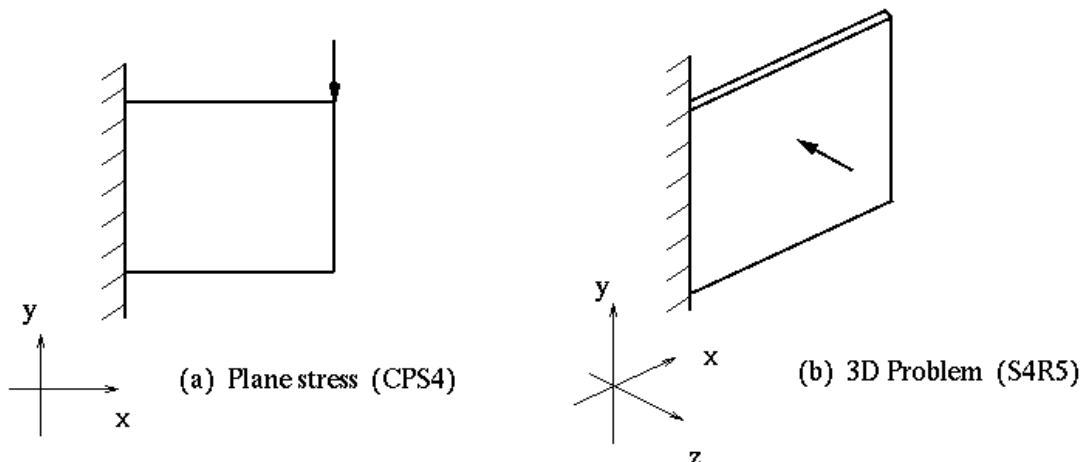
If the ABAQUS analysis has been completed then one could use the following command in ABAQUS/Post to get a visual check of the positive normal.

```
set, fill=on  
draw, normals
```

Q3.2 : What is the difference between a shell element and a 2D solid element?

The 2D Planar elements (For example : CPE4, CPE8 - plane strain analysis and CP4, CPS8 - plane stress analysis) are only used in situations where the loading is confined to the plane of the elements. The elements only have planar variables (d.o.f) u_x, u_y .

Shell elements are needed for out-of-plane loading. Consider a square plate subjected to a loading normal to the plane of the plate. This requires shell elements and use of plane stress/plane strain type of elements would be inappropriate under these circumstances.

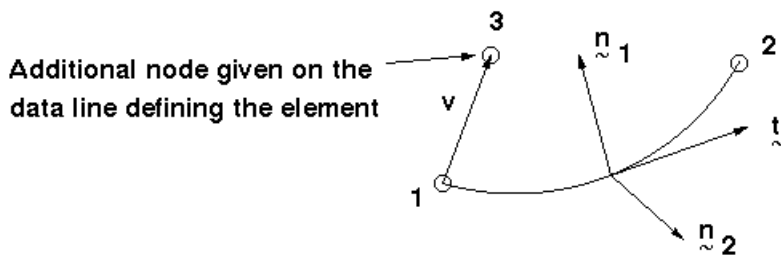


Shell elements can also be used where the loading is planar but the material is made of composites. Since shell elements by definition allow for through thickness variation of material properties these are the appropriate elements to be used in these cases.

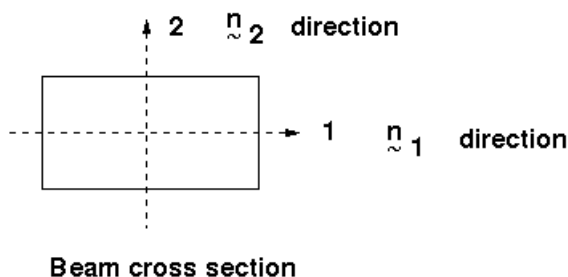
Q3.3 : How do I specify the local-1 direction for a beam in space?

See the section 6.1.2 of the **Getting Started with ABAQUS/Standard** manual for a description of this.

There are a number of ways of specifying this. In the following figure **n1** represents the local-1 direction. Consider a beam element of rectangular cross-section. Then local-1 direction is the direction parallel to the width (or base of the element). See Figure 15.3.4-1 and the figure in Page 15.3.9-15 of the ABAQUS User's manual (Ver 5.7).



Orientation of the beam element tangent, \tilde{t} , and beam section axes, \tilde{n}_1 and \tilde{n}_2



If the beams all lie in the X-Y plane then by default the negative Z axis is taken as the local-1 direction. The following are the different methods available in specifying the local-1 direction.

1. Specify an additional node in the element entry for the beam. If a third node is specified then the direction connecting node 1 to 3 defines the **v** direction in the above figure. This direction is used as an approximate **n1** direction. ABAQUS then defines **n2** direction as $\mathbf{t} \times \mathbf{v}$. Having determined **n2**, ABAQUS defines the actual **n1** direction as $\mathbf{n2} \times \mathbf{t}$. To summarise as long as **v** lies in the same plane as the **t** and **n1** vectors no errors are introduced.
2. Specify the approximate **n1** direction on the element section option. Then ABAQUS uses the same procedure as above (method 1) to calculate the **n2** direction first and then re-calculates the **n1** direction again which it uses in the analysis.

```
*BEAM SECTION, SECTION=I, ELSET=<element set name>, MATERIAL=<material>
-0.6, 2.4, 3.0, 2.0, 0.2, 0.2, 0.2
<n11>, <n12>, <n13>
```

```
*BEAM GENERAL SECTION, ELSET=<element set name>,
SECTION=<section type>, MATERIAL=<material name>
<cross-section dimensions> or <section engineering properties>
<n11>, <n12>, <n13>
<Young's modulus (E)>, <torsional shear modulus (G) >
```

If both the additional node and the **n1** direction were specified as part of the section properties then the additional node takes precedence.

As mentioned earlier ABAQUS calculates the **n2** direction from the **t** and the approximate **n1** directions. There are two methods that can be used to override this.

1. Give the components of **n2** as the 4th, 5th and the 6th data values following the nodal coordinates on the data lines of the *NODE option.
2. Use the *NORMAL option.

If both methods are used then *NORMAL takes precedence. When **n2** direction is specified using one of the above methods the beam element tangent **t** is calculated as **n1** x **n2**.

After the ABAQUS analysis has been completed the following commands can be used to check visually the **t**, **n1** and **n2** directions.

```
*SET, FILL=ON
*VIEW, VIEW=(x1,y1,z1),UP=(x2,y2,z2)
Example : *VIEW, VIEW=(1,1.5,2), UP=(0,1,0)
*DRAW, NORMALS
```

n1 (beam 1-axis) is shown in blue. **n2** (beam 2-axis) is shown in red. **t** (beam tangent) is shown in white.

Q3.4 : Why do you need to specify the local-1 direction for a beam in space?

Consider a non-circular cross-section (example : rectangular section). The bending stiffness is affected by which way the width is oriented. local-1 direction fixes the orientation without any ambiguity.

Q3.5 : I am having a problem interpreting the stress output from a shell element?

If you are using the shell elements (example : S4, S8R5) then you need to be aware that the stresses are defined in local directions which in turn are dependent on the orientation of each element w.r.t the global axes.

The [Shell element sign convention](#) explains the sign convention for the local directions.

4. ABAQUS - Mesh

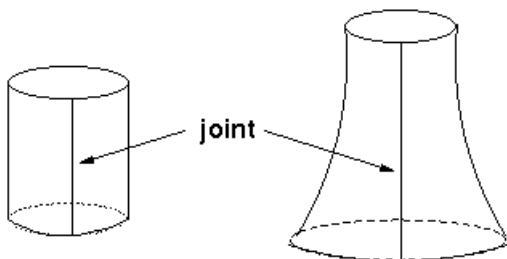
Q4.1 : I have created a mesh in two parts separately and have ended up with two sets of node numbers along a common edge. Is there any alternative to editing the *.inp file to replace one set of node numbers by the other set in the element-nodal connectivity list ie element entries?

Yes. Use the **TIE** option available under MPC (Multi Point Constraint) to tie the respective nodes as illustrated in the figure below :

```
*MPC
TIE, 100, 200
TIE, 101, 201
.....
TIE, 104, 204
```

100	101	102	103	104
200	201	202	203	204

This would then treat these pair of nodes as identical ie. they will have the same nodal variables. This can also be useful in situations where the mesh used in an axisymmetric analysis is generated from a single surface wrapped around (rather than using the axisymmetric elements available in ABAQUS). See figure below. Then the **TIE** option could be used along line joining the 2 edges (and the 2 sets of nodes).



If there are several such nodes these could be grouped together into sets and the TIE option specified with a single data line. Here the corresponding nodes should appear in the correct order.

```
*NSET, NSET=TOP
100, 101, 102, 103, 104
*NSET, NSET=BOT
200, 201, 202, 203, 204
*MPC
TIE, TOP, BOT
```

Q4.2 : I have created a mesh but the scale used is wrong. Is it possible to scale the mesh with ABAQUS?

This is being investigated.

5. ABAQUS - Materials

Q5.1 : How do I find what material properties are needed for a particular analysis ?

Read the relevant section in Chapter 6 : Analysis Types (User's manual Vol. I). This gives an overview about the analysis and has more information about the material properties.

Read also the following sections in Chapter 9 : Materials Introduction of the ABAQUS User's manual.

- Section 9.1.1 - Material Library : Overview
- Section 9.1.2 - Material Data Definition
- Section 9.1.3 - Combining Material Properties

Section 9.1.3 lists the material model combination tables. Several models are available to define the mechanical behaviour (elastic, plastic).

Some material options require the presence of other material options. Some exclude the use of the other material options. For example *DEFORMATION PLASTICITY completely defines the material's mechanical behaviour and should not be used with *ELASTIC.

Once you have all the relevant keywords to define the material properties consult the keyword section - Chapter 23 (User's manual Volume III) for each of the keywords. This will explain what data is required for each of the keyword.

Q5.2 : What material properties need to be specified in a thermal-electrical analysis ?

Referring to Section 9.1.3 of the ABAQUS User's manual you will require the **heat transfer** properties as well as the **electrical** properties. These are listed below :

- Heat Transfer properties
 - *CONDUCTIVITY
 - *LATENT HEAT
 - *SPECIFIC HEAT
 - *HEAT GENERATION
- Electrical properties
 - *DIELECTRIC
 - *ELECTRICAL CONDUCTIVITY

- *JOULE HEAT FRACTION
- *PIEZOELECTRIC

This forms the complete set of properties. If Piezoelectric elements are not used then *PIEZOELECTRIC and *DIELECTRIC properties will not be required.

If only the steady state heat transfer response is of interest then *SPECIFIC HEAT properties are not required. Similarly if there are no phase changes involved then *LATENT HEAT is not required.

*JOULE HEAT FRACTION is used to specify the fraction of electrical energy that will be released as heat.

Example problem 3.2.24 - thermal-electrical modeling of an automotive fuse illustrates the thermal-electrical analysis.

ABAQUS allows for redundant material properties to be specified. It will simply ignore the material properties not required for the current analysis.

Typical example of material properties :

```
*MATERIAL, NAME=ZINC
*CONDUCTIVITY
0.1121, 20.0
0.1103, 100.0
*ELECTRICAL CONDUCTIVITY
16.75E3, 20.0
12.92E3, 100.0
*JOULE HEAT FRACTION
1.0
*DENSITY
7.14E-6
*SPECIFIC HEAT
389.0
```

Q5.3 : What material properties need to be specified in an analysis using temperature-displacement elements ?

Referring to Section 9.1.3 of the ABAQUS User's manual you will require the **heat transfer** properties as well as the **mechanical** properties. These are listed below :

- Mechanical properties
 - *ELASTIC
 - Additional properties which may be required : example plastic
- Heat Transfer properties
 - *CONDUCTIVITY
 - *LATENT HEAT

- *SPECIFIC HEAT
- *HEAT GENERATION

Q5.4 : What material properties need to be specified in an analysis using piezoelectric elements?

Referring to Section 9.1.3 of the ABAQUS User's manual you will require the **electrical** properties. These are listed below :

- Electrical properties
 - *DIELECTRIC
 - *ELECTRICAL CONDUCTIVITY
 - *JOULE HEAT FRACTION
 - *PIEZOELECTRIC

Q5.5 : What material properties need to be specified in modeling concrete with reinforcements?

Use the concrete model available with rebar to model the reinforcements.

Section 4.2.7 of the ABAQUS Example's manual gives an example of the collapse analysis of a concrete slab subjected to a central point load.

The data file for that example is [collapse example](#).

```
*CONCRETE
  3000., 0.          abs. value of compressive stress, abs. value of plastic strain.
  5500., 0.0015    "          "          "
```

```
*FAILURE RATIOS
  1.16, 0.0836
```

This is used to define the shape of the failure surface (see section 7.6.7 of the ABAQUS USER's manual).

The first parameter is the **ratio** of the ultimate **biaxial compression** stress, to the **uniaxial compressive** stress. Default is 1.16.

The second parameter is the absolute value of the **ratio** of **uniaxial tensile stress** at failure to the **uniaxial compressive stress** at failure. Default is 0.09.

Tension Stiffening

```
*TENSION STIFFENING
  1., 0.
```

0., 2.E-3

First parameter is the fraction of remaining stress to stress at cracking. The second parameter is the absolute value of the direct strain minus the direct strain at cracking.

This defines the retained tensile stress normal to the crack as a function of the deformation in the direction of the normal to the crack.

Shear Retention

*SHEAR RETENTION

Not used for this example.

Reinforcement modelling

*REBAR is used to model the reinforcement.

```
*REBAR, ELEMENT=SHELL, MATERIAL=SLABMT, GEOMETRY=ISOPARAMETRIC, NAME=YY  
SLAB, 0.014875, 1., -0.435, 4
```

```
*REBAR, ELEMENT=SHELL, MATERIAL=SLABMT, GEOMETRY=ISOPARAMETRIC, NAME=XX  
SLAB, 0.014875, 1., -0.435, 1
```

Here **SLAB** is the element name or name of the element set that contains these rebars. The geometry is ISOPARAMETRIC. Other choice is SKEW. ELEMENT can be BEAM, SHELL, AXISHELL or CONTINUUM type. The following are the other parameters specified :

- cross-sectional area of the rebar.
- spacing of the rebars in the plane of the shell
- position of the rebar. Distance from the reference surface. Here the mid-surface is the reference surface and the minus sign indicates that the distance is measured in the opposite direction to the direction of positive normal. The positive normal is defined by the right hand rule as the nodes are considered in an anti-clockwise sequence.
- edge number to which rebars are similar.

Q5.6 : What material properties need to be specified in using the deformation plasticity model ?

See section 11.2.9 of the users' manual (Vol. I). See also section 23.4.7 of the users' manual (Vol. III), keyword section.

$$E \varepsilon = \sigma + \alpha \left(\frac{|\sigma|}{\sigma^0} \right)^{n-1} \sigma$$

where

- σ is the stress;
- ε is the strain;
- E is Young's modulus (defined as the slope of the stress-strain curve at zero stress);
- α is the "yield" offset;
- σ^0 is the yield stress, in the sense that, when $\sigma = \sigma^0$
 $\varepsilon = (1 + \alpha) \sigma^0 / E$
- n is the hardening exponent for the "plastic" (nonlinear) term; $n > 1$.

*DEFORMATION PLASTICITY

$E, \nu, \sigma^0, n, \alpha, t$

For example :

*DEFORMATION PLASTICITY

1.E3, 0.3, 2., 3, 0.396

Here the data line contains the Young's modulus, Poissons ratio, Yield stress, Exponent, Yield offset respectively. If it is necessary to define the dependence of these parameters on temperature then the 6th parameter will be the temperature. Then repeat the dataline for different temperatures as required.

6. ABAQUS - Boundary Conditions

Q6.1 : How do I change the boundary conditions at some of the nodes?

Re-specify the boundary condition for only the nodes for which the boundary condition has changed with the OP=NEW option.

```
*BOUNDARY, OP=NEW
  1, 1,, 2.5
  2, 1,, 2.5
  3, 1,, 2.5
```

In the above example the x-displacement is changed to be 2.5 units at nodes 1, 2 and 3.

Q6.2 : How do I completely re-define the boundary conditions?

Same as above (see the answer to question 6.1).

Q6.3 : How do I release a previously fixed d.o.f.?

Re-specify all the fixities except for the ones to be released (in the step in which these fixities are to be released) with the parameter OP=MOD.

Q6.4 : How do I apply a prescribed displacement?

Use the keyword ***BOUNDARY** and in the data line specify the node number, variable (d.o.f.) number and the magnitude of the prescribed displacement. It will require one dataline per variable that is being prescribed.

```
*BOUNDARY, OP=NEW
  1, 1,, 2.5
  2, 1,, 2.5
  3, 1,, 2.5
```

In this example nodes 1, 2 and 3 are applied a displacement of 2.5 units in the direction of the first axis (usually X axis).

Q6.5 : Are there any default boundary conditions representing "pinned" and "encastred" nodes that can be used?

Yes. The following is the list of named constraints.

```
ENCASTRE  Constraint on all displacements and rotations at a node.
PINNED    Constraint on all translational degrees of freedom.
```

XSymm Symmetry constraint about a plane of constant x coordinate.
YSymm Symmetry constraint about a plane of constant y coordinate.
ZSymm Symmetry constraint about a plane of constant z coordinate.
XASymm Antisymmetry constraint about a plane of constant x coordinate.
YASymm Antisymmetry constraint about a plane of constant y coordinate.
ZASymm Antisymmetry constraint about a plane of constant z coordinate.

Example :

```

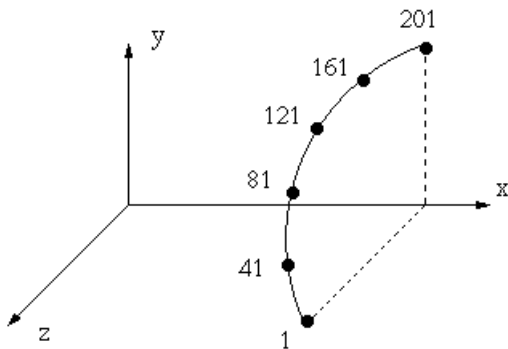
*NGEN, NSET=FIXED
1, 10
*BOUNDARY
FIXED, ENCASTRE
  
```

Here a node set which consists of 10 nodes grouped together in node set FIXED is assigned the ENCASTRE boundary condition.

Example :

```

*NODE
  1, 134.0,  0.0,  28.5
201, 134.0,  28.5,  0.0
**
*NGEN, LINE=C,NSET=CLAMPED
1, 201, 40
**
*BOUNDARY
CLAMPED, XSymm
  
```

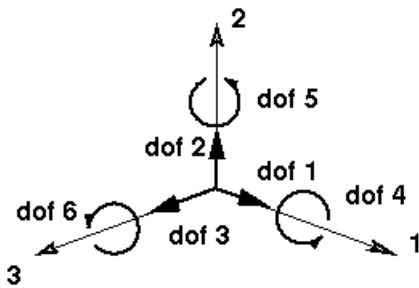


A node set called CLAMPED which consists of nodes 1, 41, ..., 201 which lie along a circular arc is first created. Then the XSymm boundary condition is specified.

Q6.6 : What are the variable numbers for the different nodal variables?

Following is a list of the more common variable numbers.

- 1,2,3 - x,y,z displacement respectively (ux, uy, uz)
- 1,2 - r,z displacement in an axisymmetric analysis (ur, uz)
- 4,5,6 - Rotation about x,y,z axes respectively (phi_x, phi_y, phi_z)
- 6 - Rotation in the r-z plane for axisymmetric shells
- 7 - warping amplitude (for open section beam elements)
- 8 - Pore pressure
- 9 - Electric potential
- 11 - Temperature
- 12 - Second temperature (for shells or beams)
- 13 - Third temperature (for shells or beams)

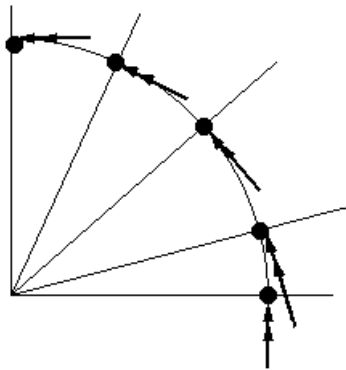


Q6.7 : Is it possible to connect different element types together in the same mesh ?

Yes there are no restrictions in connecting different element types together in a mesh. Any restriction you are likely to come across is inappropriate use of element types in certain analysis (procedures).

Q6.8 : Is it possible to apply boundary conditions w.r.t local axes?

Yes it is possible to apply boundary conditions w.r.t local co-ordinate axes. Consider the situation where you want to apply a rotation at a set of nodes which lie along the periphery of a circular arc.



Consider the figure where nodes 1 to 5 along the outer radius are to be subjected to a rotation of 1 unit. Then a local cylindrical co-ordinate system transformation is set up first. The nodes 1 to 5 are formed into an element set called CID1. The transform keyword is used to apply the transformation to the node set CID1 as follows :

```
*TRANSFORM, TYPE=C, NSET=CID1
      0.,      0.,      0.,      0.,      0.,      1.
**
*NSET, NSET=CID1
      1,      2,      3,      4,      5
**
**
** rotation boundary condition
```

```
**  
*BOUNDARY CONDITION, OP=NEW  
  1, 5, 1.  
  2, 5, 1.  
  3, 5, 1.  
  4, 5, 1.  
  5, 5, 1.
```

The applied rotation is interpreted according to the transformation.

Q6.9 : How do I ensure that the vertical displacements are the same along a line of nodes which are free to move?

Consider the situation where the nodes 1040, 1023, 1006, 989 are to have the same vertical (y-direction) displacement as node 1046. Define a nodal set (say) VERT which includes all the nodes except for 1046. Then use the *EQUATION keyword as shown below :

```
*NSET, NSET=VERT  
  1040, 1023, 1006, 989  
**  
*EQUATION  
  2  
  VERT, 2, 1.0, 1046, 2, -1.0
```

The 2 in the second line represents the number of terms in the equation. The 2 after the nodeset and node number is the d.o.f for the y-displacement. This is followed by the coefficient for the equation.

Q6.10 :Is it possible to remove a specified MPC during the latter part of an analysis?

No.

7. Loading

Q7.1 : How do I apply point loads?

Use the following statements for example. The data lines after the keyword line *CLOAD contain the **Node number** and the **Components of loads** in the **X, Y and Z** directions respectively. One dataline per node.

```
*CLOAD
 100, 0., -100., 0.
 101, 0., -80., 0.
```

If the same load is to be applied to a number of nodes then these nodes can be made into a set and single data line could be used as follows :

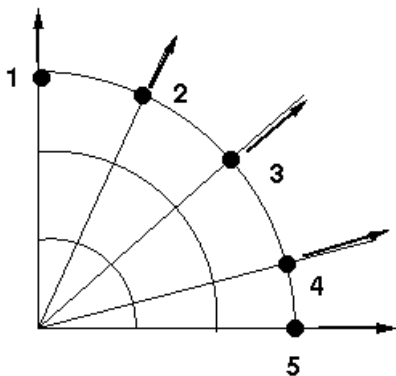
The nodes listed are grouped together in a set called **NDLSTA**.

```
*NSET,NSET=NDLSTA
 100, 110, 120, 130, 140
 .....
```

```
*CLOAD
 NDLSTA, 0., -100., 0.
```

Q7.2 : Is it possible to apply point loads w.r.t. to local co-ordinates?

Yes it is possible to apply point loads w.r.t to local co-ordinates.



Consider the figure where nodes 1 to 5 along the outer radius are to be subjected to a radial outward force of 1 unit. A local cylindrical co-ordinate system transformation is set up first. The nodes 1 to 5 are grouped into an element set called CID1. The transform keyword is used to apply the transformation to the node set CID1 as follows :

```
*TRANSFORM, TYPE=C, NSET=CID1
      0.,      0.,      0.,      0.,      0.,      1.
**
*NSET, NSET=CID1
      1,      2,      3,      4,      5
**
** radial force
**
*CLOAD, OP=NEW
```

```
1, 1, 1.
2, 1, 1.
3, 1, 1.
4, 1, 1.
5, 1, 1.
```

The applied loads are interpreted according to the transformation.

Q7.3 : How do I apply a uniformly distributed load?

Use the following lines of input to define a distributed load. Here 2D solid elements are used for this example.

```
*DLOAD
40, P3, 50. (Element 40 Side 3 is subjected to 50 units of loading)
TOP, P3, 10. (Side 3 of all elements in set TOP is subjected to 10 units of loading)
```

For shell elements subjected to pressure loading use the following statements.

```
*DLOAD
SHLGB, P, 100.
```

Here the shell elements in element set **SHLGB** are subjected to 100 units of pressure loading in the direction of the positive normal to the shell. To apply the loading in the opposite direction make the load magnitude negative.

Q7.4 : How do I apply a non-uniformly distributed normal load?

Use the user subroutine DLOAD to specify a non-uniformly distributed load. See section 25.2.5 of the ABAQUS User's manual (version 5.7).

Q7.5 : How do I apply a varying shear load?

No. It is not possible to directly specify a varying shear load. The user has to calculate the equivalent set of point loads using numerical integration and then specify it using the *CLOAD option.

Q7.6 : How do I apply a gravity loading?

First the elements subjected to the gravity loading are grouped together in an element set. Then use the following statements. Also the density should be specified as part of the *MATERIAL record and then these properties assigned to the above element set.

```
*MATERIAL, NAME=STEEL
**
*ELASTIC, TYPE=ISO
3.E+7, 0.3
**
*DENSITY
```

7800.

```
*DLOAD  
PLATE, GRAV, 9.8, 0.0, 0.0, -1.0
```

Here **PLATE** is the element set subjected to gravity loading which is defined by **GRAV**. For earth's gravity 9.8 is the actual magnitude. This is followed by the components of the gravity vector in X, Y and Z directions respectively. Here gravity acts in the negative Z direction.

For the axisymmetric case only the component 2 should be non-zero (-1).

Q7.7 : How do I apply a centrifugal loading?

Define the element set to be subjected to the centrifugal loading. The magnitude is calculated as density multiplied by the square of the angular velocity (in radians/second). For axisymmetric problems the axis of rotation must be the global Z-axis, which must be specified as 0.0, 0.0, 0.0, 0.0, 1.0, 0.0.

```
*DLOAD  
PLATE, CENT, 225., 0.0, 0.0, 0.0, 0.0, 1.0, 0.0
```

The value of 225 is arrived at from a density value of 9 and an angular velocity of 5 radians/sec.

Q7.8 : How do I apply a thermal loading ie heat input?

Section 19.4.3 of the ABAQUS 5.7 users manual (Vol. II) gives an overview of the type thermal loading that can be applied.

1. Concentrated heat flux prescribed at nodes (***CFLUX**)
2. Distributed heat flux prescribed at element faces (***DFLUX**)
3. Body heat flux per unit volume (***DFLUX**)
4. Boundary convection defined on element faces (***FILM**)
5. Boundary radiation defined on element faces (***RADIATE**)

***CFLUX**

```
*CFLUX  
56, 11, 10.  
57, 11, 10.
```

Nodes 56 and 57 are applied a heat flux of 10 units. The number 11 represents the temperature d.o.f.

***DFLUX**

```
*DFLUX  
100, SPOS, 10.
```

A uniform surface flux of 10.0 per unit area is applied to the top face (SPOS) of element 100 which is a general heat transfer shell element.

***FILM**

Prescribing boundary convection :

Heat flux on a surface due to convection is governed by :

where $q = h(\theta - \theta^{\circ})$
q is the heat flux across the surface.
h is a reference film coefficient.
 θ is the temperature at this point on the surface, and
 θ° is a reference sink temperature value.

Example

A uniform, time-dependent film condition can be defined for face 2 of element 3 by :

```
*AMPLITUDE, NAME=sink
0.0, 0.5, 1.0, 0.9
*AMPLITUDE, NAME=famp
0.0, 1.0, 1.0, 22.0
...
**
*STEP
*HEAT TRANSFER
...
*FILM, AMPLITUDE=sink, FILM AMPLITUDE=famp
3, F2, 90.0, 2.0
```

A uniform, temperature-dependent film coefficient and a time-dependent sink temperature can be defined for face 2 of element 3 by

```
*AMPLITUDE, NAME=sink
0.0, 0.5, 1.0, 0.9
*FILM PROPERTY, NAME=filmp
2.0, 80.0
2.3, 90.0
8.5, 180.0
...
**
*STEP
*HEAT TRANSFER
...
*FILM, AMPLITUDE=sink
3, F2, 90.0, filmp
```

General

In the element type section in the ABAQUS user's manual the type of loading that can be applied is listed under each element type category. This type of loading is only applicable to the following element types :

1. Heat transfer elements
2. Coupled thermal-electrical elements

3. Coupled temperature-displacement elements

These categories of elements can be found under the following broader group of element types :

1. 1-D Solids (Only heat transfer elements are available)
2. 2-D Solids
3. 3-D Solids
4. Axisymmetric Solids
5. Shell Elements

Look for the heading **Distributed heat fluxes** under the element group and these list the type of heat input that can be specified.

2-D and 3-D Solids

- BF - Heat body flux per unit volume
- Sn - Heat surface flux per unit area into face **n**

Add NU after the load type if the heat input is non-uniform.

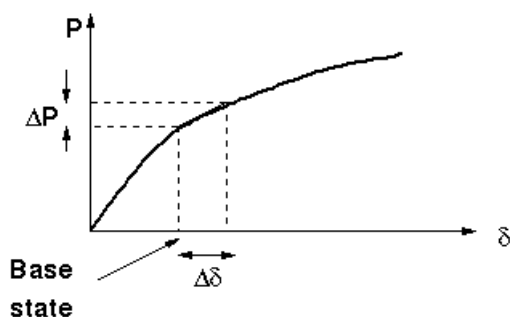
Shell Elements

- BF - Heat body flux per unit volume
- SNEG - Heat surface flux per unit area into the bottom face of the element.
- SPOS - Heat surface flux per unit area into the top face of the element.

Add NU after the load type if the heat input is non-uniform.

Q7.9 : Is the loading specified in a step the "total" load or an "incremental" load?

For an analysis which consists of solely **general** steps the loads are considered to be **total**. For a **perturbation** step the load specified is treated as **incremental** ie ABAQUS will seek a response to the specified load about the **base state**. Here the base state is the state at the end of the last general step.



The output (displacements, stresses) will reflect this interpretation. These (displacements, stresses) are calculated as changes from the base state.

Q7.10 : How do I remove the loading applied in a previous step?

This is illustrated with the *CLOAD type of loading. Specify with the option OP=NEW without any data lines.

```
*CLOAD,OP=NEW
```

The same applies to other type of loading (*DLOAD).

If the analysis involves loading and subsequent unloading then the load-unload time history can be specified using the *AMPLITUDE option and then this is associated with the loading.

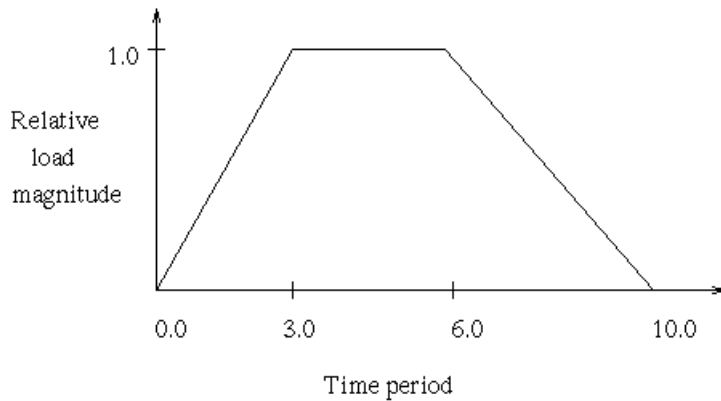
In a simple linear elastic static analysis all that is needed is the load magnitude. But in a more complex analysis - dynamics or nonlinear analysis it might be necessary to define loading that varies with time. One way ABAQUS allows for this is by using the *AMPLITUDE option (User's manual section 19.1.2, Vo. II, version 5.7).

```
*AMPLITUDE, NAME=UPDOWN
  t1, f1, t2, f2, t3, f3
```

Here the time and the load scale factors are listed in pairs. For example :

```
*AMPLITUDE, NAME=UPDOWN
0., 0., 3.0, 1., 6.0, 1., 10., 0.
....
....
**
** step 1
**
*STEP, INC=50
*STATIC
0.1, 5.0,
*DLOAD, AMPLITUDE=UPDOWN
TOP, P3, 1000.
*END STEP
**
** step 2
**
*STEP, INC=50
*STATIC
0.1, 5.0,
*DLOAD, AMPLITUDE=UPDOWN
TOP, P3, 1000.
*END STEP
```

This first defines a time variation which rises linearly from a value of 0.0 at time zero to 1.0 at a time 3.0, remains at that value until a time of 6.0, then ramps down to 0.0 at a time of 10.0 as shown in figure.



This time variation is called UPDOWN, and is referred to on a *DLOAD option, where it governs the behaviour of a pressure load applied on side 1 (load type P1) of a set of elements called TOP. This magnitude will vary through time according to the amplitude definition, so that, for example, at a time 1.5 the pressure will be 500. The magnitudes specified in the AMPLITUDE option are used as scaling factors.

You will notice that because of the absence of the **perturbation** option in the *STEP keyword this is a general analysis where time accumulates. Because the step time is only 5.0 the loading of 1000 is applied and the unloading phase has not begun. In the second step which is also of duration of 5.0 units of time in the first 1.0 unit of time the loading remains unchanged. Then the loading is gradually reduced and at the end of the step it has reached the unloaded stage.

From the above example it should be clear that the *AMPLITUDE specification if necessary can span across more than one step in a general analysis.

If you are using the *AMPLITUDE option in a perturbation step then the time range cannot straddle more than one step. However it is perfectly acceptable to use *AMPLITUDE when the time used is the step time. The example below shows such a step. The amplitude definition UPDOWN remains unchanged. The following step has a step time of 10 units which is the same as the time range specified in the amplitude definition.

```
**
** step 1
**
*STEP , PERTURBATION , INC=100
*STATIC
0.1,10.0,
*DLOAD , AMPLITUDE=UPDOWN
TOP , P3 , 1000.
*END STEP
```

Q7.11 : What do I do to maintain the load at the same level as in a previous step?

Either re-specify the data input for the previous step or completely omit the load data input. If the loading was specified using *CLOAD keyword then omit these.

Q7.12 : How do I change part of the previously applied loading?

Specify the loading with the OP=MODIFY option.

```
*CLOAD, OP=MODIFY
 25, 0., -25.
```

Then the loading is modified for the listed nodes ONLY. At all other nodes the loading applied in the previous step using the *CLOAD keyword (for this example) remains unchanged.

Q7.13 : How do I remove a single load which forms of group of loading which has already been applied in a previous step?

In the step in which you want to remove the single load re-specify the group of loading except for the load to be removed with the OP=NEW option. Here it is assumed that the type of loading is the same for the whole group (example : *CLOAD).

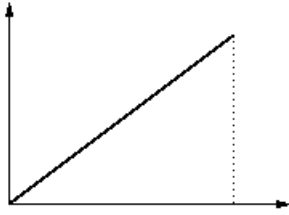
```
....
** step in which the group of loading is applied
**
*CLOAD
 25, 0., -25.
 50, 0., -25.
 75, 0., -25.
100, 0., -25.
....
** step in which the load applied to node 25 is removed
*CLOAD, OP=NEW
 50, 0., -25.
 75, 0., -25.
100, 0., -25.
```

Q7.14 : Is it possible to specify a line load along a line of nodes?

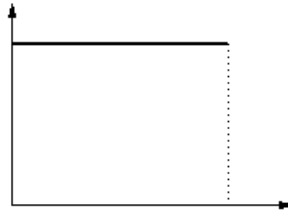
No it is not possible to specify directly a line load in ABAQUS. The user needs to calculate the **equivalent** set of point loads for the nodes along that line and then specify it using *CLOAD keyword. This requires the use of virtual work principle and the shape function defining the displacements.

Q7.15 : What are RAMP and STEP loading?

RAMP and STEP define how and when the loading is applied during a given step. The following figure shows the difference between the two.



(a) RAMP



(b) STEP

Which of these two forms the default option depends on the procedure used for the step and the type of loading. See section 10.1.1 of the ABAQUS User's manual (version 5.5) for further explanation.

Q7.16 : How do I apply a pressure loading to a shell element (S8R5) which is in the global X direction?

Use the following statements.

```
*DLOAD  
SHELL, BX, pressure/t
```

SHELL is an element set consisting of all the shell elements subjected to pressure. The loading should be specified as body force. Therefore divide the pressure by the thickness of the shell elements.

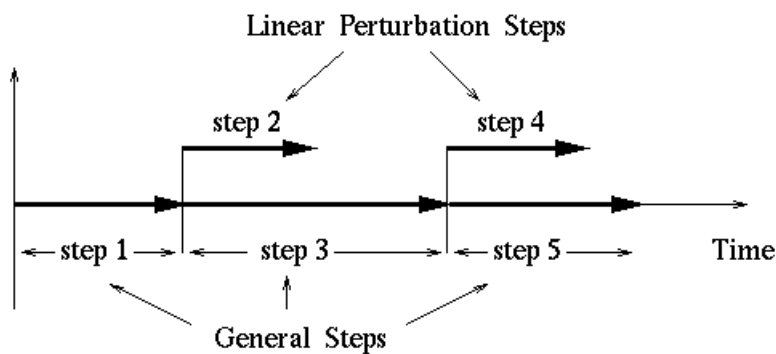
8. ABAQUS - Procedures

Q8.1 :

9. ABAQUS - Analysis

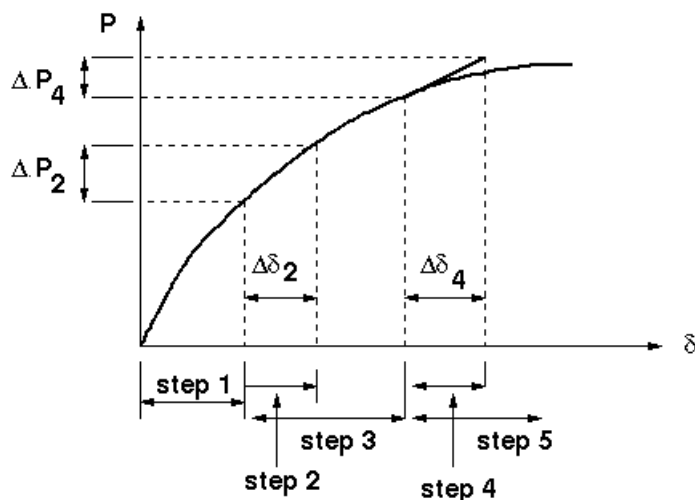
Q9.1 : What is the difference between "General" and "Perturbation" steps?

Most complex analysis are likely to have a sequence of **steps**. For example consider an analysis with four steps (see figure below). In step 1 the structure could be subjected to a static loading. Step 2 could be a frequency step. Step 3 could be further loading and step 4 another frequency step. The frequency steps 2 and 4 would be dependent on the state reached at the end of the previous steps (namely 1 and 3). This is generally true for any type of analysis with more than one step. The behaviour of the structure is dependent on the state at the end of the previous step.



An analysis step during which the response is nonlinear is called **general** analysis step. An analysis step during which the response is linear is called a **linear perturbation** step. A linear perturbation analysis step provides the linear response of the system about the **base state** ie the state at the end of the last nonlinear analysis step prior to the linear perturbation step. In the above example the state at the end of step 1 forms the base state for the frequency response in step 2.

Also because the step 2 is a perturbation step it has no influence on the response in step 3 which only depends on the results at the end of step 1. Similarly for step 5 the base state is the state at the end of step 3.



The following procedures are purely linear perturbation procedures (The references

made are to the User's manual version 5.7):

- BUCKLE (Section 6.2.3)
- FREQUENCY (Section 6.3.4)
- MODAL DYNAMIC (Section 6.3.5)
- RANDOM Response (Section 6.3.8)
- RESPONSE SPECTRUM (Section 6.3.7)
- STEADY STATE DYNAMICS (Section 6.3.6)

A **STATIC** step could be made into a perturbation step by including the **PERTURBATION** parameter on the ***STEP** keyword.

```
*STEP, PERTURBATION  
*STATIC
```

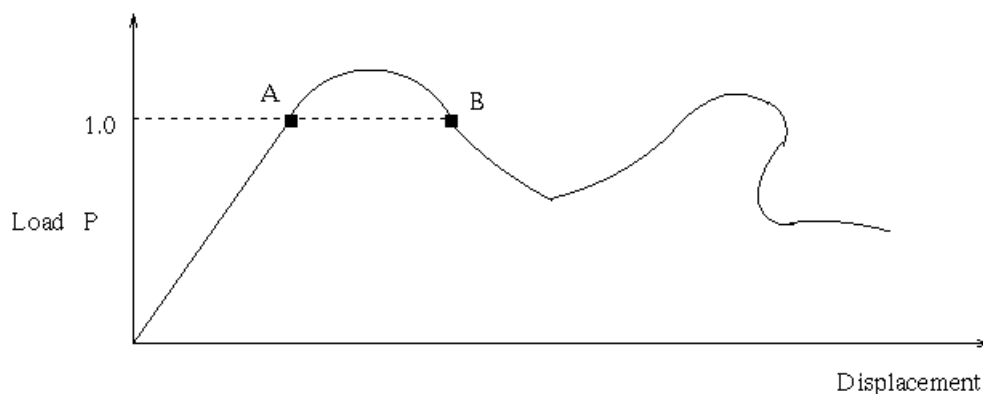
Omitting this parameter implies that a general ***STATIC** analysis is required.

Each step has **step time**. The **total time** is the accumulation of the step times of all the **general steps** in an analysis. The step times of all perturbation steps are omitted from this.

Similarly all results quantities are accumulated values from all the general steps in an analysis. The results output in the perturbation steps are incremental values about the base state.

Q9.2 : How do I carry out a non-linear analysis where there is a possibility of instability?

Geometrically nonlinear static problems frequently involve buckling or collapse behaviour, where the load-displacement response shows a negative stiffness. The static equilibrium states during the unstable phase of the response can be found by using the "modified Riks method". This method is selected by including the **RIKS** parameter on the ***STATIC** keyword (Section 6.2.4 of the User's manual version 5.7).



In a large deformation analysis the effect of geometric nonlinearity can be significant. Use the **NLGEOM** option with the ***STEP** keyword to take into account the changes in

geometry during the analysis. Then the stiffness matrix is calculated using the current configuration ie using the current position of the nodes.

The Riks method cannot be used when the only loading is by applied moments and/or applied rotations. However the bending moment can be represented by a couple ie set of point loads acting in the opposite direction, separated by a suitable distance.

A Riks step cannot be followed by another step in the same run. Subsequent steps must be analysed using the *RESTART option (Section 7.1.1 User's manual version 5.7).

```
*STEP, INC=10, NLGEOM
*STATIC, RIKS
```

Q9.3 : In trying to restart an analysis from a step which did not complete I am getting the same error message?

In the *RESTART keyword line add the **END STEP** parameter. Example :

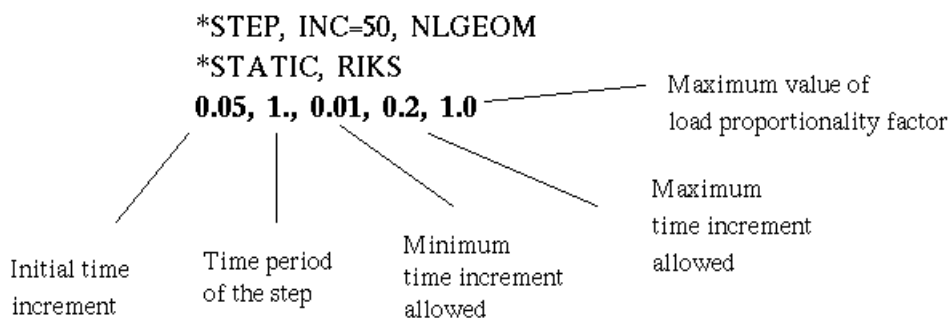
```
*RESTART, READ, STEP=2, INC=10, END STEP, WRITE
```

Q9.4 : I am interested in the dynamic response (ie oscillations) due to heating of a thermocouple. How do I go about analysing this problem?

This analysis has to be carried out in 2 parts. First part is a heat transfer analysis. Then carry out a dynamics analysis with the temperatures from the first analysis as input. The element types for the 2 analyses will be different.

Q9.5 : I have carried out a static analysis with the RIKS method and I find that the time at the end of the step exceeds the step time. What do I do?

Check the dataline for the *STATIC keyword includes the **load proportionality factor**. This is the 5th parameter in the data line. This should be set to 1.



Q9.6 : What is the significance of the "minimum" and "maximum" time increments in a step?

This is explained with reference to the figure for question 9.5 above.

In the above example the maximum time increment allowed is 0.2. This will not be exceeded in any increment in this step. By default automatic time stepping will be used. This means that the ABAQUS program will choose the largest time increment on efficiency and other grounds.

In a given increment if convergence is not possible it will re-solve with a reduced time step. ABAQUS will repeat this procedure until convergence is reached in a given increment. However the analysis will be terminated if ABAQUS has to reduce the time step to a value less than the minimum time step (0.01 for this example). These parameters sets limits and are means of controlling what happens in a given step. If any one of the limits is reached the step is terminated. Irrespective of which limit is reached first the step will be terminated.

Q9.7 : In carrying out a frequency analysis I am getting warning messages about negative eigenvalues. What do I do?

Check the boundary conditions if any are specified. Check the material properties.

Check that original shape of the elements in the mesh are reasonable. Triangular elements must be nearly equilateral (and aspect ratios should not exceed 3). Similarly quadrilateral elements should not exceed an aspect ratio of 10.

If no errors are detected then draw the different eigenmodes and check that the mode shapes are realistic. If so the warning messages can be ignored. However in general warning messages should not be routinely ignored. These may have some implications on the results from the analyses.

Q9.8 : Can I apply any loading in a frequency step?

Any applied loading during a frequency step (using *FREQUENCY procedure) will be ignored.

However if *STEP, NLGEOM is used in a general analysis then the initial stress stiffness effect of any loading applied in a prior step will be taken into account in the eigenvalue extraction.

Therefore to include the effect of any pre-loading apply the loading in the first step with *STEP, NLGEOM option.

If initial stresses prescribed by *INITIAL CONDITIONS, TYPE=STRESS are to be considered in the frequency extraction then include this in a static step (step 1) which

includes the NLGEOM option. Specify the appropriate boundary conditions and loading in the same step. Then use *FREQUENCY in step 2.

Q9.9 : I want to use the displacement obtained from one analysis to modify the geometry (to change the nodal position) and carry out a separate analysis. Is this possible?

Yes it is possible. This is catered for by the *IMPERFECTION keyword.

Let us assume that the analysis-ID of the analysis from which the displacements (to be used as imperfection) is "static1". This could be anything and used here for illustrative purposes. Run this analysis and this will create the results file static1.res among other output files.

The second analysis should have the following input. Only the essential ones are shown.

```
*HEADING
Second analysis which uses the displacement from the previous analysis
as an imperfection.
*NODE
Data lines to define the initial "perfect" geometry
.....
*IMPERFECTION, FILE=static1, STEP=, INC=, NSET=
  1,
....
**
*STEP, NLGEOM
*STATIC, RIKS
...
*CLOAD and/or *DLOAD
data lines to specify loading
*END STEP
```

In the above specify the as the STEP number and the if the displacements from the previous analysis are to be scaled. Otherwise specify it as 1.

INC and NSET are the other optional parameters. If the displacements at the end of particular increment rather than the end of the step (from the previous analysis) is to be considered then set the INC parameter. Similarly if the displacements of only a subset of the nodes in the mesh is to be considered as the imperfection then define a node set and specify it using the NSET parameter.

If the previous analysis is a buckling or a frequency analysis then the *IMPERFECTION key word is as follows :

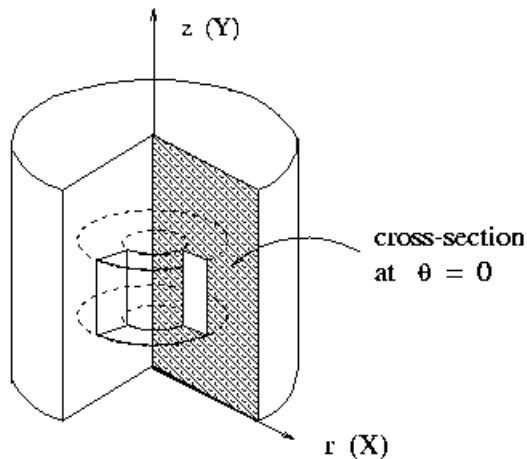
```
*IMPERFECTION, FILE=buckle1, STEP=, NSET=
,
,
as many lines as required modes to be used.
```

In this case the imperfection is a linear combination of the selected mode shapes.

Q9.10 : In an axisymmetric analysis are there any

restrictions in the choice of the axis of symmetry?

Yes, Only the z axis can be the axis of symmetry. In an axisymmetric analysis r-z-theta forms the 1-2-3 axes. The radial axis is considered to be horizontal going from left to right. The z-axis (the axis of symmetry) is the axis pointing vertically upwards. The 3rd-axis is directed towards the user, normal to the rz plane.



The radial distance is measured from the z axis and $r=0$ at the z axis. Only the radial section is meshed and ABAQUS integrates this section through 360 degrees for any calculation which involves the volume.

10. Output

Q10.1 : How do I suppress output being written to the *.dat file?

Use the following statements in the abaqus input file (*.inp).

```
*EL PRINT, POSITION=INTEGRATION POINTS, FREQUENCY=0
**
*EL PRINT, POSITION=CENTROIDS, FREQUENCY=0
**
*EL PRINT, POSITION=NODES, FREQUENCY=0
**
*NODE PRINT, FREQUENCY=0
```

This will suppress any element or nodal output being written to the printed output file (*.dat).

Q10.2 : I want to print element stresses averaged at nodes as well as the Gauss points. How do I do that?

Use the following statements in the abaqus input file (*.inp).

```
*EL PRINT, POSITION=INTEGRATION POINTS, FREQUENCY=n
S
*EL PRINT, POSITION=AVERAGED AT NODES, FREQUENCY=n
S
```

This will print separate tables consisting of the stresses at Gauss points and the values averaged at the nodes for every nth increment.

Q10.3 : I want to print a single table with stresses and strains at Gauss points. How do I do that?

Use the following statements in the abaqus input file (*.inp).

```
*EL PRINT, POSITION=INTEGRATION POINTS, FREQUENCY=n
S,E
```

This will print a single table which contains the stresses and strains at the Gauss Points.

Q10.4 : I would like to print the displacements w.r.t. local co-ordinates. How do I do that?

First of all you need to specify the local co-ordinate system using the *TRANSFORM keyword. This is then applied on a selected set of nodes.

```
*TRANSFORM, TYPE=C, NSET=CID1
0., 0., 0., 0., 0., 1.
```

```

**
*NSET, NSET=CID1
      1,      2,      3,      4,      5

```

The implication of the use of *TRANSFORM is that any input **concentrated force and moment** you apply to this node set will be interpreted in the local co-ordinate system.

Then use the following statement in the ABAQUS input file.

```

*NODE PRINT, GLOBAL=NO, FREQUENCY=n
U

```

This in fact the default output option. If necessary the displacements w.r.t. the global axes can be obtained by using GLOBAL=YES in the *NODE PRINT statement.

However if you want to use the displacements w.r.t local co-ordinates in some graphical output (ie drawing contour of one component of displacement w.r.t. local co-ordinate) then it will not be possible even if you specify the following data in the ABAQUS input file.

```

*NODE FILE, GLOBAL=NO, FREQUENCY=n
U

```

The reason is *NODE FILE controls the data written to the *.fil (results) file and changing this has no effect on any contours. Contours are produced from the *.res (restart) file which is always written in global coordinates. PATH plots are also generated from the *.res file and so will again will always be in the global coordinate system.

However you can use FEMGV to produce contour plots of displacements w.r.t local co-ordinates because FEMGV reads the results from the *.fil for post processing. So depending on whether you included GLOBAL=YES or NO the contours of the displacements will be different if using FEMGV.

Q10.5 : I would like to print the displacements w.r.t. local co-ordinates. How do I do that?

This is done using the ORIENTATION option in the *SOLID SECTION keyword and the specification fo the *ORIENTATION keyword as illustrated below :

```

**
** square_plate
**
*SOLID SECTION, ELSET=SQUARE_P, MATERIAL=STEEL,ORIENTATION=CYL
      5.,
**
*ORIENTATION, SYSTEM=CYLINDRICAL,NAME=CYL
0., 0., 0., 0., 0., 1.
3, 0

```

Here a local cylindrical co-ordinate system is specified. Then use of the following output option

```

**
*EL PRINT, POSITION=INTEGRATION POINT, FREQ=1
S

```

produces the following output in the (*.dat) file. Note the presence of OR next to the integration point which indicates use of the local co-ordinate system with the

*ORIENTATION option.

ELEMENT	PT	FOOT-	E11	E22	E12
NOTE					
1	1	OR	-2.9537E-08	-2.7172E-09	3.5417E-08
1	2	OR	-3.4626E-08	6.6933E-09	7.9072E-09
1	3	OR	-5.5444E-09	-1.3702E-08	3.0689E-08
1	4	OR	-1.2208E-08	-2.7172E-09	2.6730E-08

OR: *ORIENTATION USED FOR THIS ELEMENT

Q10.6 : In running a ABAQUS analysis I can't find the *.msg file?

This would indicate that data errors must have been detected during the pre (data checking) stage of the ABAQUS run. The detected errors can be found in the printed output file (*.dat).

Q10.7 : In running a ABAQUS analysis the *.dat file has terminated abruptly. Whats wrong?

When ABAQUS completes a run irrespective of whether it had detected any errors or not there will always be a summary at the end of the printed output file (*.dat). The last statement lists the CPU time used for the particular stage (pre, analysis). If you find that the *.dat file has come to an abrupt end then the most likely reason is that in creating the various output files you disk quota has been exceeded.

If you find that there is a difference between the usage and allocation of disk quota you need to remember that one of the ABAQUS files (*.023) is created in the directory from which ABAQUS is run and gets deleted when it completes its run. This could explain the discrepancy.

Q10.8 : How do I generate a customised output file?

If none of the *NODE PRINT and *EL PRINT input statements with its option cater for your requirement (example : you want the nodal displacements printed to a greater accuracy than it is printed in the *.dat file) then you can

1. Write a post processing program to read the *.fil file and write out the results to a separate file after the ABAQUS run has completed. A typical post processing routine is listed in section 5.1.3 of the ABAQUS User's manual (Version 5.7).
 2. Use one of the user subroutines (example : USDFLD or UVARM) to write the required results to a separate file during the ABAQUS run.
-

Q10.9 : How do I monitor the progress of a long analysis?

A useful output file is the status file (*.sta). This is useful in checking the progress of a long ABAQUS run and also to monitor the state of the analysis, while it is running. There is one line of output for each increment of each step which is written to the status file at the end of each increment.

By including the following statement in the *.inp file one can monitor the progress of a single variable (d.o.f) at a single node.

```
*MONITOR, NODE=11, DOF=2
```

Then the y-displacement (uy) at node 11 will be written to the *.sta file so that the user can monitor its progress during a ABAQUS run. The contents of a typical status file is listed below.

SUMMARY OF JOB INFORMATION:

STEP	INC	ATT	SEVERE DISCON ITERS	EQUIL ITERS	TOTAL ITERS	TOTAL TIME/ FREQ	STEP TIME/LPF	INC OF TIME/LPF	DOF MONITOR	IF RIKS
1	1	1	0	1	1	1.00	1.00	1.000		
MONITOR NODE: 11 DOF: 2										
2	1	1	0	1	1	1.01	1.000E-02	1.0000E-02	-27.9	
2	2	1	0	1	1	1.02	2.000E-02	1.0000E-02	-21.7	
2	3	1	0	1	1	1.03	3.000E-02	1.0000E-02	-14.0	
2	4	1	0	1	1	1.04	4.000E-02	1.0000E-02	-4.05	
2	5	1	0	1	1	1.05	5.000E-02	1.0000E-02	7.47	
2	6	1	0	1	1	1.06	6.000E-02	1.0000E-02	17.7	
2	7	1	0	1	1	1.07	7.000E-02	1.0000E-02	24.4	
2	8	1	0	1	1	1.08	8.000E-02	1.0000E-02	28.6	
2	9	1	0	1	1	1.09	9.000E-02	1.0000E-02	30.0	
2	10	1	0	1	1	1.10	.100	1.0000E-02	26.7	

Type **tail -10 analysis-id.sta** and this will display the last 10 lines of the status file.

Example : **tail -10 plate.sta**.

You can also use the the following command which will display the contents of the status file as it gets updated. You need to use the CTRL/C to quit when the analysis is completed.

tail -f plate.sta.

Q10.10 : I have run an analysis and the *.res file has not been created. Whats wrong?

Check whether the following statement has been included in the *.inp file.

```
*RESTART,WRITE,FREQUENCY=n
```

Q10.11 : In running a ABAQUS analysis using the user subroutine option the printed output is terminated after the datacheck stage and there are

no report of any errors?

This can happen if the user subroutine you have written had syntax or some other errors. Check the *.log file for clues to the errors. Example :

```
PROGRAM ABORTED : IEEE OVERFLOW  
PROCEDURE TRACEBACK ...
```

Q10.12 : In running a ABAQUS analysis with the user subroutine option the output file written to in the user subroutine couldn't be found. Whats wrong?

The specified file is created and then deleted with the run-time files. This can happen when the file-name used does not include the full path name.

Example : `open(26, file='user.out',status=new)`

Specify the full path name and then this file will not be deleted at the end of the ABAQUS run. To find out the full path name use the **pwd (which represents "present working directory") command in the Unix system.**

Example : If typing "pwd" yields the following :
`/usersn/ug/92/92abc/abaqus`

Then append the file name to the directory as follows :

```
open(26, file='/usersn/ug/92/92abc/abaqus/user.out',status=new)
```

Q10.13 : I have used the cylindrical co-ordinate system to specify the nodal position using *NODE with SYSTEM=C. Does it mean that the displacements printed at the end of the analysis will be in terms of the cylindrical co-ordinate system?

No. the displacements printed will still be in terms of the global axes. Use *NODE with SYSTEM=C is just for ease of specifying the position of nodes in a cylindrical co-ordinate system. It does not mean that the nodes belong to a different local co-ordinate system.

Q10.14 : What is the default option for printing stresses and strains in the printed output (*.dat) file?

In the absence of *EL PRINT keyword ABAQUS will print the stresses and strain at all the integration points for all the elements in the mesh for every increment(which is the default option). This can be a substantial amount of output resulting in very large printed output (*.dat file).

11. ABAQUS/Post - General

Q11.1 : Is it possible to use the same set of commands repeatedly for post processing different analyses?

Yes. The commands you type in during a session of ABAQUS/Post is written to a file called **abaqus.jnl**. Before starting another session of ABAQUS/Post (which would overwrite the commands of the previous session) make a copy of this file (say) **analysis.jnl**. Then this file can be used in the next session by typing :

```
input, file=analysis.jnl
```

after starting ABAQUS/Post. It is also possible to edit and then re-use it.

Q11.2 : Is it possible to cut open and develop a cylindrical mesh and produce contour plots on the developed flat surface?

No. It is not possible to do this with ABAQUS/Post. However it is possible with FEMGV. See answer to the question [Q23.4](#).

Q11.3 : Is it possible to use the displacements in terms of local co-ordinates in post processing an analysis with an axisymmetric mesh and non-axisymmetric loading?

No. It is not possible to do this with ABAQUS/Post. However it is possible with PATRAN and FEMGV.

Q11.4 : Is it possible to choose your own layout for multiplots other than the 1,4,9 frames per page option provided by ABAQUS/Plot?

The simple answer is no. However with a bit of effort one could choose ones own layout and then produce a hardcopy of it.

Q11.5 : Is it possible to take a section along a line of nodes and plot the stress distribution for a selected eigenmode in a frequency analysis?

No. It is not possible to do this with ABAQUS/Post. However it is possible with FEMGV. See answer to the question [Q23.6](#).

Q11.6 : Is it possible to produce a contour plot of the stress distribution for a selected eigenmode in a frequency analysis?

No it is not possible to produce a contour plot of stress or any other output parameter for any of the eigenmodes in a frequency analysis. However this is possible with FEMGV and PATRAN.

Q11.7 : Is it possible to produce a plot of the applied boundary conditions?

Use

```
SET, BC DISPLAY=ON  
DRAW
```

to display the boundary conditions. Use

```
SET, BC DISPLAY=OFF
```

to switch it off.

Q11.8 : Is it possible to produce a plot of the applied loading?

Yes but only for concentrated loads specified with the *CLOAD keyword. Use

```
SET, LOAD DISPLAY=ON  
DRAW
```

to display the loading. Use

```
SET, LOAD DISPLAY=OFF
```

to switch it off.

Q11.9 : Is it possible to draw only a subset of the mesh?

Yes use the DETAIL command with the names of the element sets.

```
*DETAIL, ELSET=FLANGE
*DRAW
```

```
*DETAIL, ELSET(FLANGE, BOLT)
*DRAW
```

```
*DETAIL, ELSET=(FLANGE, 45, 50)
*DRAW
```

To display only elements 5, 10 and 15 :

```
*DETAIL, ELSET=(5,10,15)
*DRAW
```

The **DETAIL** command is very useful and also has the following options parameters: **NSET**, **REBAR**, **SURFACE**, **MAXIMUM** and **MINIMUM**. **NSET** for node sets and nodes, same as for **ELSET** with elements. **NSET** can be used with **ELSET**.

```
*DETAIL, ELSET=SHOCK, NSET=FRONT
```

This displays the element set **SHOCK** and that part of node set **FRONT** that overlaps **SHOCK**.

REBAR to display the position of reinforcements modelled using **REBAR**.

SURFACE can be used to specify surface names and surface set names. See the **ABAQUS/Post** manual for more details.

Q11.10 : After viewing a subset of the mesh what is the command to restore the view of the whole mesh?

Use the command **DETAIL** command without any options :

```
*DETAIL
*DRAW
```

Q11.11 : Is it possible to find the location a particular node?

Yes, use the **DETAIL** command. For example to find the position of the node 45 :

```
*DETAIL, NSET=45
*DRAW
```

Q11.12 : Is it possible to find the nodes associated with a particular element?

Yes. Here again the **DETAIL** command is used :

```
*DETAIL, ELSET=24
*DRAW
```

Q11.13 : When using the dynamic plotting capabilities of ABAQUS/Post using the mouse the mesh has disappeared from view. How can I restore the mesh view?

Try the following command :

```
zoom, reset  
draw
```

If the mesh view is dis-orientated use the following command to restore it to the default setting.

```
view, v=(0,0,1), up=(0,1,0)  
draw
```

Q11.14 : How can I find which variable I can use in which type of plot and from which file?

Consult the chapter on **Output Variable Identifiers** in the ABAQUS/Post manual.

The variables are printed under separate tables depending on whether it an element integration point variable, element section variable or nodal variable or some other type of variable. Some variables are analysis dependent. The first task is to locate the variable in one of these tables.

Then look for the filled circle under the heading COPA and VECT. If found under the COPA heading then a contour or path plot of the variable (which could be a scalar quantity or a vector component) in question can be obtained. Similarly look under VECT for a vector plot.

The other three categories (XYFI, XYRE, XYSE) indicate which file if possible can be used in a READ CURVE command for any given variable.

Q11.15 : How do I plot the surface normals for a pair of surfaces in a contact analysis?

First of all create a surface set which consists of the pair of surfaces in contact. Then use the DETAIL and DRAW commands :

```
*surface set, name=blsurf  
>blank_t, punch  
> CR
```

Here CR represents the RETURN key

```
*zoom, reset  
*detail, surface=blsurf  
set,vector length=0.002  
*set, vector tip scale=0.003
```

```
*draw, normals
```

This example is taken from the Chapter on **Contact** in the **Getting started with ABAQUS/Standard** book.

Q11.16 : Is it possible to colour the different components differently in a complex mesh?

Use the following commands :

```
*set, fill=on  
*color set, elset=flange, color=red  
*color set, eslet=web, color=blue  
*color set, elset=diaphragm, color=green  
*draw
```

It is also possible to colour a visible face of an element set.

```
*color set, elset=all, face=3, color=blue  
*draw
```

This can be switched off with the following command :

```
*color set, elset=all, face=3, color=off  
*draw
```

12. ABAQUS/Post - Contours

Q12.1 : How do I get a labelled contour plot?

Use the command

```
set, c labels=on
```

Q12.2 : How do I increase the size of the contour legend ?

Use the command

```
set, c legend size=0.5
```

Q12.3 : How do I change the position of the contour legend ?

Using the command

```
set, c legend=(fx,fy)
```

Example : `set, c legend=(0.1,0.5)`

which defines the position of the top left hand corner of the contour legend as fraction of the page size. This will position the legend near the left hand side and half way up the page.

Q12.4 : How do I remove the axes and title from the display ?

Use the commands

```
set, c axes=off  
set, c title=off
```

Q12.5 : How do I add my own title to the display?

Use the command

```
title, origin=(0.2,0.3), size=0.04, title=In-line Title Example
```

Q12.6 : How do I change the size of the text used in the title of the contour plot ?

Use the command

```
set, c title=0.05
```

Q12.7 : How do I change the position of the title?

Use the following command to specify the start position of the title.

```
set, c title=(fx,fy)
```

Example : `set, c title=(0.1,0.3)`

Q12.8 : How do I get a contour plot of the unsmoothed stresses?

Use the command

```
set, c quilt=on  
contour, var=s11
```

Q12.9 : How do I get the contours plotted on a deformed mesh plot ?

Use the command

```
set, c magnification=100  
draw, d
```

and then issue the contour drawing command. Example

```
contour, name=stress, variable=mises
```

Q12.10 : In plotting the contours on the deformed mesh how do I change the scale used for the displacements ?

Use the command

```
set, c magnification=100  
draw, d
```

Q12.11 : Is it possible to draw a contour plot of the displacement magnitude (not a contour plot of the individual displacement component)?

No, it is not possible.

Q12.12 : What are the commands to specify the

minimum an maximum values of a contour plot?

```
set, c max=1.E5  
set, c min=0.0
```

Q12.13 : What is the command to specify the number of contour intervals?

```
set, c level=10
```

13. ABAQUS/Post - Mesh Plots

Q13.1 : How do I plot the mesh?

Use the command

```
draw
```

Q13.2 :How do I generate a wireframe mode of the mesh?

Use the commands

```
set, hide=off  
set, fill=off (default)  
set, outline=element (default)  
draw
```

Q13.3 : How do I generate a hidden line plot of the mesh?

Use the command

```
set, c hide=on  
draw
```

Q13.4 : How do I generate a shaded view of the mesh?

Use the command

```
set, fill=on  
draw
```

Q13.5 : How do I generate a view of the mesh with shrunken elements? ?

Use the commands

```
set, shrink=0.2  
draw
```

To reset it subsequently type

```
set, shrink=0.
```

Q13.6 : How do I generate a plot of the mesh with just the mesh outline?

Use the command

```
set, outline=perimeter
```

To restore drawing of element boundaries, type

```
set, outline=element
```

Q13.7 : How do I get a plot of the deformed mesh only?

Use the commands

```
set, undeformed=off  
draw, deformed or draw, d (in short)
```

Q13.8 : How do I get a plot of the mesh with node and element numbers?

Use the commands

```
set, el numbers=on  
set, n numbers=on  
draw
```

Q13.9 : How do I get only the node and element numbers for a subset of the mesh?

Use the command

```
detail, elset=seta or detail, nset=401
```

Q13.10 : How do I display the loading and boundary conditions?

Use the commands

```
set, bc display=on (for boundary conditions)  
set, load display=on (for loading)  
draw
```

Q13.11 : How do I change the scale used for the displacement in plotting the deformed mesh?

Use the command

```
set, d magnification=100  
draw, displaced
```

Q13.12 : What is the significance of the displacement magnitudes in a FREQUENCY analysis ie eigenmode analysis?

They have no special significance. The displacements are normalised to give the largest displacement to be unity.

Q13.13 : I have used the rebar option to model the reinforcement in concrete in an analysis. How do I display the position of the rebars?

Use the command

```
detail, rebar  
draw
```

Q13.14 : How do I mark the position of the nodes?

Use the following command:

```
set, node=on  
dr
```

It can be switched off using :

```
set, node=off
```

14. ABAQUS/Post - XY Plots

Q14.1 : How does one produce a stress vs strain plot for an integration point in a non-linear analysis?

Use the following commands

```
verify curve
read curve, name=stress, variable=mises, element=1, integration point=4
read curve, name=strain, variable=peeq, element=1, integration point=4
define curve, operation=combine, name=strsstrn
> strain, stress
>
display curve
> strsstrn
>
```

Create the curves of variation stress and strain for the duration of the analysis separately and then combine the two curves to get the stress-strain plot. In combining the first specified curve parameter is used for the x-axis (strain in the current example).

Q14.2 : Is it possible to read in results from elsewhere (experimental, analytical) and use it to compare it with ABAQUS results?

Yes an ascii file containing a table of rows and column of numbers can be read into ABAQUS/Post. Use the command

```
read curve, name=userdata, file=results.dat, xcol=1 ycol=4
```

In the above example the results will be read from a file called **results.dat** and the values in the first column will be used as the x values and the values in the 4th column will be used as the y values. The values in the first 3 columns will be ignored. Needless to say that each line should have at least 4 values. The created curve will be given the name **userdata**. This can then be used in subsequent plotting using the **display curve** command.

Q14.3 : Is it possible to write out the data from a XY plot into a file for use elsewhere, example : Matlab?

Yes it is possible. Use the command

```
set, xy print file=results.out
print curve, xdisp
```

Here the results of the curve **xdisp** which exists is written to a file called **results.out**.

Q14.4 : I want to create a force vs displacement plot at a node which is subjected to external point load. How do I do it?

In the ABAQUS input file include the following statements :

```
*NODE FILE, NSET=NDWF, FREQUENCY=n  
U  
CF
```

Here the node set NDWF is assumed to include the node which is subjected to external point loads and for which the plot is required. CF represents such point loads.

Then in ABAQUS/Post use the following command.

```
results, file=platen  
read curve, name=force21, node=21, var=CF2  
read curve, name=disp21, node=21, var=U2  
define curve, operation=combine, name=fcedsp  
> disp21, force21  
>  
display curve  
> fcedsp  
>
```

Q14.5 :I want to create a force vs displacement plot at a node which has a prescribed non-zero displacement. How do I do it?

In the ABAQUS input file include the following statements :

```
*NODE FILE, NSET=NDWF, FREQUENCY=n  
U  
RF
```

Here the node set NDWF is assumed to include the node which is given a prescribed non-zero displacement and for which the plot is required. RF represents the reaction force.

Then in ABAQUS/Post use the following command.

```
results, file=platen  
read curve, name=force21, node=21, var=RF2  
read curve, name=disp21, node=21, var=U2  
define curve, operation=combine, name=fcedsp  
> disp21, force21  
>  
display curve  
> fcedsp  
>
```

Q14.6 : I want to create a pressure at an element face vs nodal displacement plot where the element in question is subjected to external pressure

loading. How do I do it?

In the ABAQUS input file include the following statements :

```
*NODE FILE, NSET=NDWF,FREQUENCY=n
U
*EL FILE, FREQUENCY=n
LOADS
```

Here the node set NDWF is assumed to include the node which is associated with the element side subjected to pressure loading and for which the plot is required. LOAD represents the pressure applied to element faces.

Then in ABAQUS/Post use the following command.

```
results, file=platen
read curve, name=press15, element=15, variable=LP3
read curve, name=disp21, node=21, var=U2
define curve, operation=combine, name=prsdsp
> disp21, press15
>
display curve
> prsdsp
>
```

Here the element face 3 is subjected to the pressure load.

Q14.7 : I want to create a plot of the sum of the reactions at a set of nodes given a prescribed displacement against the displacement at the nodes. How do I do it?

```
results, file=platen
read curve, name=react, node=fend, variable=RF2, sum
read curve, name=disp, node=101, var=U2
define curve, operation=combine, name=fd
> disp, react
>
display curve
> fd
>
```

Here the node set **fend** contains the group of nodes which are specified a prescribed displacement (which is non-zero). The **sum** at the end of the read curve command will sum up the reactions at all the nodes within the **fend** node set.

Node 101 is assumed to be part of the **fend** node set. Here the prescribed displacement is applied in the y direction. Hence the use of RF2 and U2.

Q14.8 : How do I find the names of the curves created so far during a session of ABAQUS/Post?

Use the following command :

```
show, curve attributes
```

15. ABAQUS/Post - Vector Plots

Q15.1 : How do I generate a displacement vector plot?

Use the command

```
vector plot, variable=u
```

Q15.2 : What are the other output parameters that can be used in a vector plot?

See the ABAQUS/Post manual (pages 4.1.1-4) for a list of variables that can be used in a vector plot. Apart from displacements, reaction forces, principal stresses and strains can be used in a vector plot.

```
vector plot, variable=rf  
vector plot, variable=sp (principal stresses)
```

For the case of principal stresses the colour blue represents compression and yellow represents tension.

use the following commands to adjust the length of the arrowheads used in the plots

```
set, v tip scale=0.1  
set, v length=0.1 (fraction of the longest model dimension)
```

Q15.3 : Is it possible to produce a vector plot of the principal stresses?

Yes. Use the command :

```
vector plot, var=sp
```

Q15.4 : How do I change the length of the arrows representing the vectors in a vector plot?

See the answer to Q15.2.

16. ABAQUS/Post - Path Plots

Q16.1 : Is it possible to take an arbitrary line section through a 2D model and display the variation of a selected output parameter along that line?

No. It is not possible for 2D models. However this is possible with the FEMGV program. See the answer to the question [Q23.5](#).

However the above is possible with 3D models using ABAQUS/Post. For example use the command

```
path, var=mises, name=test5, start=(-70,50,20), end=(70,50,20), absolute
```

This creates a curve set names **test5** that gives the variation of von Mises stress in solid elements along a straight line in space. The straight line is defined by giving the **start** and **end** points.

17. ABAQUS/Post - Views

Q17.1 : How do I change the direction of the view?

Use the command

```
view, view=(a,b,c), UP=(d,e,f)
view, view=(0,0,1), UP=(0,1,0) (for example)
```

The UP option defines which direction should be up. This is an optional parameter.

Alternatively use the left mouse button (hold and drag) to rotate the view. The following command can be used to regain the original view.

```
reset, all
draw
```

Q17.2 : Are there any dynamic view capabilities available with ABAQUS/Post?

Yes. The mouse buttons can be used to manipulate the views. Mouse buttons when you hold and drag has the following functionality.

Left - rotates view.

Middle - pans the view. To reset use the command **reset, fit**

Right - zooms in and out. To reset use the command **reset, fit**

Q17.3 : How do I get the default view after using the mouse buttons to change the view? ?

To reset after using the middle or right mouse button use the command **reset, fit**.

Q17.4 : How do I zoom in ?

Use the right mouse button (hold and drag) towards and away from the origin. Alternatively type the following commands :

```
zoom, cursor
```

```
draw
```

Q17.5 : How do I reset after using zoom in?

Use the command

```
zoom, reset
```

Q17.6 : How do I generate an animation?

Use sequence, capture.

18. ABAQUS/Post - Hardcopy

Q18.1 : How do I get a colour postscript output?

Specify **device=cps** in the command line.

```
abaqus plot job=job-id device=cps
```

Q18.2 : How do I create a hpgl output file? ?

Specify **device=hgl** in the command line.

```
abaqus plot job=job-id device=hgl
```

Q18.3 : How do I create an encapsulated postscript output file for inclusion in a latex document ?

The **device=cps** option actually creates an encapsulated postscript file. If the *.mpl file has more than 1 plot then use the option to create separate output files when running ABAQUS/Plot.

Q18.4 : Is it possible to create a A3/A2/A1 size output from a ABAQUS analysis? ?

Yes. This is possible with ABAQUS/Post (version 5.7).

It is also possible with PATRAN and FEMGV.

Q18.5 : Is it possible to create a plot of size 200 mm x 100 mm from a ABAQUS analysis? ?

Yes it is possible with version 5.7.

19. ABAQUS/Plot

Q19.1 : After creating a neutral plot file (*.mpl) using ABAQUS/Post I was trying to view the generated plots but there weren't any?

Make sure that you have exited from running ABAQUS/Post or issued a **set, hard copy=close** command **before** starting ABAQUS/Plot and accessing the neutral plot file created by ABAQUS/Post.

20. Pre processing using PATRAN

Q20.1 :

21. Post processing using PATRAN

Q21.1 : Is it possible to use PATRAN for post processing even if it was not used for pre-processing ie not used to generate the mesh?

Yes. Create a new database using the **abaqus.db** template. Click on the radio button marked **Analysis** and change the `Action' from **Analyse** to **Read Both**. Default option is **Read Results** if you had used PATRAN to generate the mesh.

Then follow the instructions given in the **Beginner's Guide to PATRAN**.

22. Pre processing using FEMGV

Q22.1 :

23. Post processing using FEMGV

Q23.1 : Is it possible to have a 2-plot layout instead of the 1, 4 plot layouts?

No. It is not possible to have 2-plot layout with FEMGV.

Q23.2 : Is it possible to write out the results of a XY Plot to an external ascii file?

Yes. Use the following commands.

```
UTILITY TABULATE OUTPUT PRINTFILE OPEN file-name  
< Then issue commands to draw the plot >  
PRESENT GRAPH LINE OLD  
CLOSE file-name
```

Here use an appropriate file name instead of file-name. If no file name is specified the data is written to a file called **fgv1st.lst**.

Q23.3 : Is it possible to read in the results of a XY Plot from an external ascii file?

No it is not possible to read in results from an external ascii file which contains x and y values to be used in XY Plots.

Q23.4 : Is it possible to cut open and develop a cylindrical mesh and produce contour plots on the developed flat surface?

Yes it is possible. Use the following commands to develop the surface.

```
VIEW DEVELOP X (develop model about X axis. Similarly about other axes)
```

Then issue commands to plot the contours.

It is also possible to cut open non-developable surfaces using this option.

The command to revert back to the original surface ...

Q23.5 : Is it possible to take an arbitrary line section through a 2D model and display the variation of a selected output parameter (example : von Mises stress) along that line?

Yes. Use the following commands.

```
CONSTRUCT LINE FREE /CURSOR  
< This is followed by 2 cursor clicks to indicate 2 points to define line >  
< Then issue commands to draw the plot >  
PRESENT GRAPH LINE OLD
```

Q23.6 : Is it possible to take an arbitrary section through a 3D model and display contours of a selected output parameter (example : von Mises stress) on that section?

Yes. Use the following commands.

First choose the right direction of view for making a cut.

```
EYE DIRECTION X Y Z
```

Then make the cut.

```
CONSTRUCT XSECTION /CURSOR  
< This is followed by 2 cursor clicks to indicate 2 points to define plane >  
EYE NORMAL (to get a view normal to cut section)  
RESULTS NODAL STRESS XX (select an output parameter)  
< Then issue commands to draw the plot >  
PRESENT CONTOURS LEVEL 10
```

Q23.7 : Is it possible to produce a contour plot of the un-smoothed stresses?

If the stresses at the integration points are written to the results (*.fil) file along with the integration point coordinates then it is possible to plot contours of the **un-smoothed** stresses.

Use the following commands for this.

```
RESULTS GAUSSIAN STRESS component  
Example : RESULTS GAUSSIAN STRESS XX  
PRESENT CONTOUR LEVEL 10 or P C L 10 (in short)
```

Similar plots can also be obtained for strain components.

Q23.8 : Is it possible to produce a contour plot of the smoothed stresses if only the stresses at the

integration points were written to the results (*.fil) file?

Yes it is possible. Use the following commands.

```
RESULTS GAUSSIAN STRESS component
Example : RESULTS GAUSSIAN STRESS XX (select the first stress component)
RESULTS CALCULATE AVERAGE (this calculates averaged nodal values)
RESULTS NODAL STRESS component (select nodal stress component)
PRESENT CONTOUR LEVEL 10 or P C L 10 (in short)
```

Similar plots can also be obtained for strain components.

The alternate option is to write the stresses (and strains) averaged at the nodes in the results (*.fil) file. In the ABAQUS input file use the following data.

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

24. ABAQUS - Errors

Q24.1 : In a 3-dimensional analysis using mixture of elements (CDR8, S4R5 and M3D4) degrees of freedom of M3D4 elements appear to get fixed where these are suppose to be free. Why is that?

The membrane elements do not have any stiffness in the normal direction. Unless stiffness contributions are made in this direction by the other element types this results in zero pivots. ABAQUS tries to fix these in order to able to find a solution.

amb2@eng.cam.ac.uk

Last updated on 27 Auguts 1998