# Running ABAQUS 6.9

Arul M Britto

May 7, 2010

## Contents

# 1 Introduction

ABAQUS is a general purpose finite element program distributed by ABAQUS Ltd. It has a large element library and is capable of analysis of a variety of problems (linear, nonlinear, static, dynamic, structural and thermal). ABAQUS/CAE is an interactive graphical for running ABAQUS. Post processing is provided by a separate module known as the Viewer.

The main aim of this manual is provide an overview. It does not provide any instructions on how to run the CAE module. This has been covered extensively elsewhere. However essential use of ABAQUS/Viewer is covered but by no means comprehensive.

The remainder of this section deals with the manuals, the commands to run the various modules and how to go about running an ABAQUS analysis in the Cambridge University Engineering Department's Teaching System (T.S.) Computers.

Section 2 of this manual deals with the input to the ABAQUS Analysis module. Section 3 summarises the system dependent commands necessary to run ABAQUS in the Teaching System. Section 4 lists the various files created during an ABAQUS run. Section 5 summarises the printed output. Section 6 deals with the post processing facilities available with ABAQUS. Section 7 deals with post-processing using the PATRAN program. Section 8 deals with how to troubleshoot when you encounter problems. Section 9 covers further information on additional manuals, examples and ABAQUS FAQ's.

This handout is mainly aimed at ABAQUS/Standard users. There is no equivalent handout for the ABAQUS/Explicit program. ABAQUS/Explicit users should consult the relevant user manuals.

## 1.1 Manuals

A set of the manuals is available for reference in the Computer Operator's Office. Not all the manuals are available in paper form. Some manuals are only available on-line (Example : Theory, Verification ...). See also the **Using ABAQUS On-line documentation**.

The on-line version of the documnetation is quite comprehensive and up-to-date.

### 1.1.1 ABAQUS/Standard and Explicit

1. ABAQUS/CAE User's Manual
2. Getting Started with ABAQUS Interactive Edition
3. Getting Started with ABAQUS Keyword Edition
4. ABAQUS Analysis User's Manual (Volumes I, II, III, IV and V)
5. Theory
6. Example Problems Manual
7. ABAQUS Benchmarks Problems
8. ABAQUS User Subroutine Reference Manual
9. ABAQUS Glossary Manual
10. ABAQUS Keywords Reference Manual (Volumes I and II)
11. Verification Manual
12. ABAQUS Scripting User's Manual
13. ABAQUS Scripting Reference Manual
14. ABAQUS Release Notes
15. ABAQUS Installation and Licensing Guide





## 1.2 Programs

ABAQUS/Standard and ABAQUS/Explicit are both available in the CUED teaching system computers. ABAQUS/CAE is the pre-processor and ABAQUS/Viewer in the post processor. PATRAN is an established pre- and post-processor for ABAQUS.

ABAQUS/Design is suitable for parametric study and optimisation.

## 1.3 Where to Start?

Figure 1 shows the inter-relationship between the ABAQUS Analysis module and the pre and post-processor modules (CAE and Viewer).

- If you want to make a start using ABAQUS with the minimum amount of reading to get familiarised with running simple problems then start with the **ABAQUS/CAE User's manual** Appendix B tutorial of **A loaded cantilever beam**. This takes one through pre-processing, analysis and post processing from within the graphical interface ABAQUS/CAE. Appendix D covers output post processing aspects. In Chapter 4 there is a further tutorial dealing with advanced techniques ?
- If you want to familiarise yourself with the structure of the ABAQUS input data then work through the example given in this handout and try running the example (sections 2 and 3) and then complete the post processing (section 6). Then if you have to make changes to the input data you will have some idea of the structure/layout of an input file. Then you can work through the tutorials mentioned above. Then follow the suggested course of action, in carrying out your own analysis.

## 1.4 Commands

To find out about all the options that are available with ABAQUS and this will list all the options that can be used when running ABAQUS.

**abaqus help**

See also sections 3.1.1, 3.2.1 to 3.2.27 of the ABAQUS User's manual (Volume 1) for more information on execution procedures. Section 3.3.1 has examples of using the **abaqus** command.

To access the on-line information on the keywords (more details in section 9.1)

**abaqus doc**

To find out which examples make use of a particular key word, illustrated here with the keyword **dynamic**,

**abaqus findkeyword job=dynamic**
***dynamic**
*** <CR>**

The output will be written to the file referred to by **job** and with the extension **dat** ie dynamic.dat. The analyses data files listed in this file can be obtained using the **abaqus fetch** command. To extract a copy of the input files for the **airspring** example

**abaqus job=airspring* fetch**

To run the pre-processing program ABAQUS/CAE

**abaqus cae**

It is also possible to submit the job and do the post processing from ABAQUS/CAE without exiting from it.

To run an ABAQUS analysis (both ABAQUS/Standard and Explicit)

**abaqus job=abcd1**

To run the post processing program ABAQUS/Viewer

**abaqus viewer**

To create an executable module from a user written Fortran program prog1.f (this would create prog1.x)

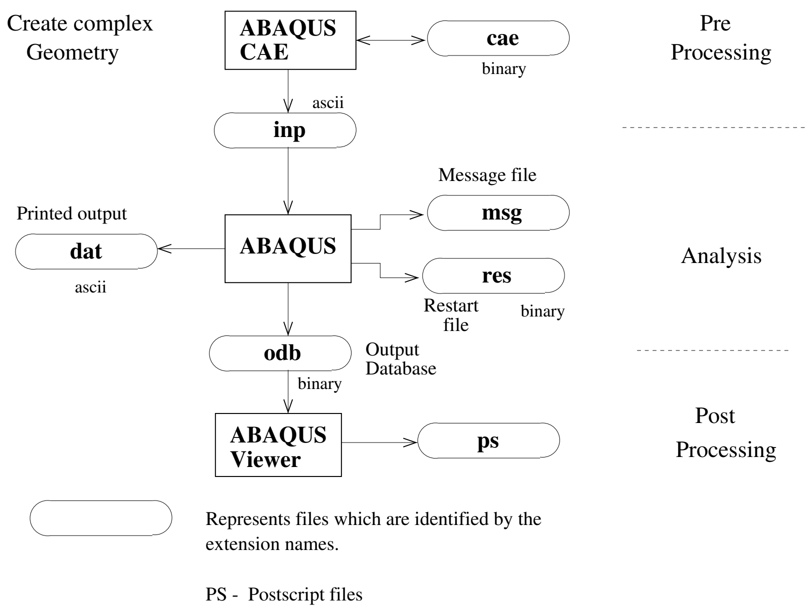



Figure 1: ABAQUS/CAE, ABAQUS/Viewer and the Analysis Module





**abaqus job=abcd1 make user=prog1**
To run the executable module created using the make command :
**abaqus prog1.x**
To run MSC/PATRAN
**patran**
For other ABAQUS commands see section 3.1.1 of the User's manual Vol. 1. Section 3.3.1 has examples of its use.

## 1.5 Suggested Course of Action

- New users are recommended to read through sections 1.1.1 & 1.2.1 of the Analysis User's manual volume I. Section 1.2.2 deals with conventions and units.

  The **Getting Started with ABAQUS/Standard** manual is the best starting place for new users. It has examples on linear, non-linear, dynamic, plasticity and contact problems. It also has a good introduction to the use of continuum elements, shell and beam elements.

  For users who are launching themselves into their first analysis the relevant information for any given analysis is spread over the many ABAQUS manuals.

  You need to read all the relevant sections, make notes as you go along and also follow up on all cross-references you come across.

  You need to find all the information pertaining to the following topics for your analysis

  – Analysis Type (Procedures)
  – Element Types
  – Material Behaviour
  – Loading
  – Boundary Conditions
  – Output

  Only after reading all the relevant information are you in a position to run your own analysis. It is a good idea to find an example which is closest to your analysis requirement.

- Chapter 6 of the Users manual deals with the different **analyses** and **procedures**. This is a good place to start.

- Chapter 17 gives an introduction to the **material behaviour**. Chapters 17 to 22 cover all the material models available in ABAQUS.

  ABAQUS provides a broad range of **material models**. Section 17.1.3 has material model combination tables which is helpful in showing which material properties could be combined together and which cannot be.

- Chapter 23 introduces the **element types**. Look through chapter 23 to 25 of the above manual to identify the element types to be used in the analysis. This section also has information on the type of loading. Chapters 26 to 28 contains information on more non-general specialist element types.

  Once the element type has been identified in the same section a list of active dof, type of loading that can be applied to the element type and the output parameters available for these elements can be found.

- Look through the **Element Index** in volume II for the element types. Look through the Keywords Manual to check the various keywords. Follow the references to the Analysis User's manual. Refer to the sections which are relevant to your analysis.

- Chapter 29 (volume II) has information on **loading** and **boundary conditions**. Chapter 30 has information on **MPC** (Multi Point Constraints). Information on **contact** problem can be found in chapters 31 to 35.
- Section 4.1.1 (Volume I) deals with the **output** from ABAQUS.
- Search through the contents of the Example Problems Manual to find the examples in the field of your interest.
- Look through the contents of the Verification and Benchmarks manuals to find the examples in the field of your interest.
- Another useful source is the course notes of the various courses run by SIMULIA Ltd.
- Run an example problem and familiarise yourself with the various files and in the use of the various program modules (ABAQUS, ABAQUS/CAE, ABAQUS/Viewer).
- Output Variable index can be found in OI.1 nad OI.2 for Standard and Explicit respectively. EI.1 and EI.2 has the Element index for Standard and Explicit versions respectively.
- Start with a simplified situation of your analysis with very few elements, steps and increments. Then the turnaround time while you debug the data and root out the errors will be shorter. Once you familiarise yourself with ABAQUS you can refine the mesh and add complexity to the analysis.
- In order to familiarise oneself with the 6.9 style format of the input file section 2.9.1 **Defining an Assembly** in the Analysis Users's Manual Volume 1 is recommended reading. This is essential reading only if you need to make changes to the input file created by CAE using a text editor. This is again if you are using features/keywords currently not supported by CAE.

## 2 Data Preparation

Figure 2 shows the various ABAQUS modules and the inter-relationship between PATRAN, Pro/ENGINEER programs and ABAQUS. As shown in this figure the ABAQUS input data file *.inp can be created by PATRAN interactively. It is also possible to use the mesh generation commands available in ABAQUS (section 2.4). If you are working with complex and/or 3-dimensional geometries use of either PATRAN or FEMGEN is recommended.

Complex 3 dimensional geometries can be created in Pro/ENGINEER and imported into PATRAN and then meshed before a ABAQUS analysis. It is also possible to use the new ABAQUS/CAE module which provides the capability of both a solid modeller (Pro/ENGINEER) and a pre-processor (PATRAN). However because CAE is relatively new it does not as yet cater for the full capabilities offered by the analysis modules (ABAQUS/Standard & Explicit).

ABAQUS/CAE has solid modelling capabilities which may not be as comprehensive as that of a solid modeler such as Pro/ENGINEER but never the less may be adequate for most users requirement. It also has an option where any keywords which is not yet catered for using the menu based system can be directly typed in using a text editor from within ABAQUS/CAE. The advantage of doing this within the ABAQUS frame work instead of outside is that the ABAQUS database does not get out of step with the input data. ??

Alternatively the user can refer to the ABAQUS Analysis User's manuals and type in the input data file (*.inp) in a text editor consisiting of ABAQUS keyword and data lines..

Section 2.4 gives some simple commands for mesh generation provided within the ABAQUS program itself.

The PATRAN and FEMGEN programs can be used to generate the data files for structural and thermal problems. However if you require to generate the data file for an analysis in any other specialised field it may not be possible to create the complete input data file ready for analysis. You may have to





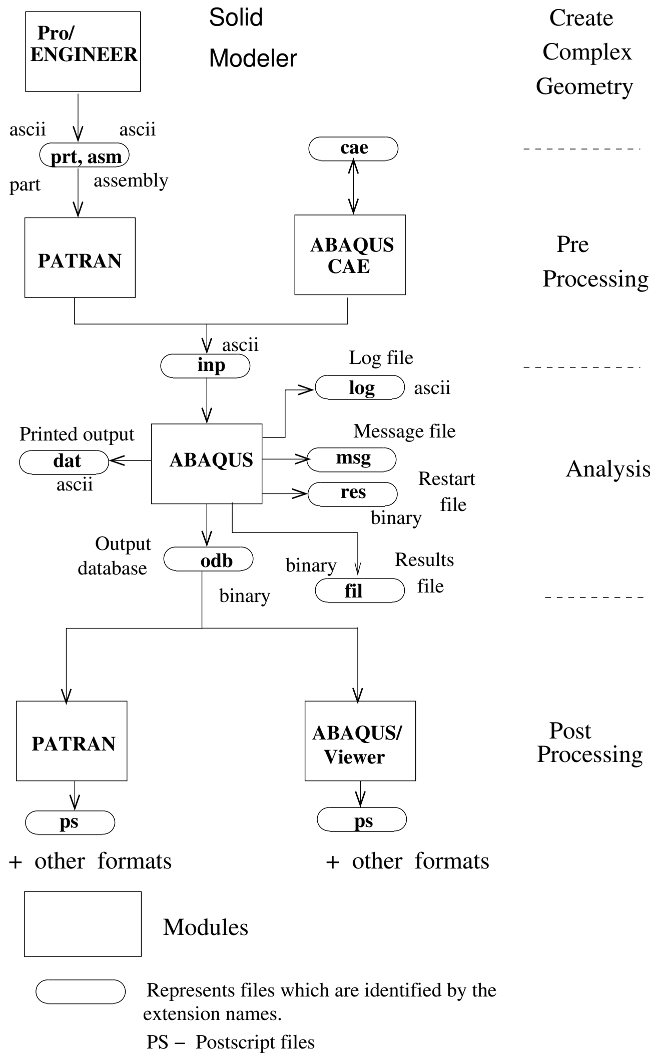



Figure 2: ABAQUS, PATRAN and Pro/ENGINEER Programs

create the datafile for a standard structural or thermal problem and then write out the *.inp data file. Then edit this file using a text editor and make the necessary changes to the element types, material properties by referring to the ABAQUS manuals and then submitting an ABAQUS job.

Section 2.2 explains what constitutes an ABAQUS input data file for a simple static analysis of a structural problem. The input data file is illustrated for a simple example.

Type *patran* to run the PATRAN program. See the "Beginner's Guide to PATRAN" on how to use the PATRAN program. Type *femrun* to run the FEMGV program. See the FEMGV User Manual on how to use the femgen program. A training guide is also available.

See also the CUED web pages under **help** / **Engineering Packages** / **finite elements** / **Pre and Post Processing** for more information on either of these programs.

## 2.1 Units

A question often asked is what units does ABAQUS use? The answer is it does not use any particular units. Another question asked : How does one set the units in ABAQUS? The answer is you cannot set the units. Input parameters are simply numbers as far as ABAQUS is concerned.

There is no units associated with ABAQUS. The user has to choose an appropriate set of units. For structural problem the user should decide on units for Length and Force. For time dependent problems the unit for Time also has to be chosen.

Example :

Length (Co-ordinates) - $m$ (metres)
Force - $N$ (Newtons)
Time - $Sec$ (seconds)

Then all dependent parameters must be specified in a consistent set of units. For example stiffness should be specified in $N/m^2$ (Pascals) and density in $kg/m^3$.

If the chosen Length dimension is metres then a thickness of 4 mm will be input as **0.004**. Instead if the chosen units was mm then input **4.0** .

It is not a good idea to select an arbitrary set of units at the beginning and then hope to sort out the results later (Example : Co-ordinates - mm, Young's Modulus - GPa).

| Quantity | SI | SI(mm) | SI | US Unit(ft) | US Unit(inch) |
|---|---|---|---|---|---|
| Length | $m$ | $mm$ | $m$ | $ft$ | $in$ |
| Force | $N$ | $N$ | $kN$ | $lbf$ | $lbf$ |
| Mass | $kg$ | $tonne$ ($10^3kg$) | $tonne$ | $slug$ | $lbf\ s^2/in$ |
| Time | s | s | s | s | s |
| Stress | $Pa$ (N/$m^2$) | $MPa$ (N/$mm^2$) | $kPa$ | $lbf/ft^2$ | $psi$ ($lbf/in^2$) |
| Energy | $J$ | $mJ$ ($10^{-3}$J) | $KJ$ | $ftlbf$ | $inlbf$ |
| Density | $kg/m^3$ | $tonne/mm^3$ | $tonne/m^3$ | $slug/ft^3$ | lbf $s^2/in^4$ |

## 2.2 ABAQUS Input

Read through section 1.1.2 (ABAQUS USER's manual Volume I) for an introduction to the ABAQUS input data file. This is essential reading for new users of ABAQUS.

The input to the ABAQUS program (<job-id>.inp or *.inp) contains **model data** and **history data**. In the rest of the manual the second option of representing the job-id with an asterisk (*) will be adopted. Model data defines the finite element model : the elements, nodes, element properties, material definition - any data that specifies the model itself. History data defines what happens to the model - the sequence of events or loadings for which the model's response is sought.

In ABAQUS the user divides the history into a sequence of **steps**. Each step is a period of response of a particular type - a static loading, a dynamic response, a soils consolidation transient etc. The definition of a step includes the *procedure* type (static stress analysis, transient heat transfer analysis





etc), *control parameters* for time integration or the nonlinear solution procedures, the *loading* and *output requests*.

The procedure option defines the type of analysis that will be carried out during a **step**. A procedure option is required for each step, and must follow immediately after the *STEP option. Any mixture of procedures may be used from step to step; but only *one* procedure is allowed in any single step.

* STATIC is one of the procedure options used for the example here. It means it is a static analysis. Here follows the list of procedures available in ABAQUS. (see section 6.1 of the Analysis User's manual for a procedures overview).

```
*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
*DYNAMIC, EXPLICIT
```

The ABAQUS/Explicit module is run when the procedure specified is *dynamic, explicit*. For all other procedures the ABAQUS/Standard analysis is run.

All data definitions in ABAQUS are accomplished with **option blocks** - set of data describing a part of the problem definition. The user chooses those options that are relevant for a particular analysis. Each option is introduced by a **keyword line**. Keyword lines begin with a * in column 1, followed by the keyword. Example : *MATERIAL.

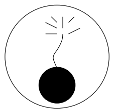
If the option requires data lines, they follow the keyword line. The syntax of keyword and data lines is defined in section 1.2.1 (ABAQUS Analysis User's manual I). Keywords may have parameters. Keyword lines use the format in which blanks are ignored and the commas (,) separate the parameters. Section 2.1.1 deals with node definition and 2.2.1 deals with element definition.

Parameters are sometimes required (for example : on the *ELEMENT option which introduces a list of element definition, the parameter TYPE is required to tell what type of element - solid, shell, beam, etc - is being defined). Many parameters are optional. Parameters may have values.

Example : *MATERIAL, NAME=STEEL

This associates the name STEEL with the material definition.

Any line which has ** in the first two columns are comment lines. Any number of comment lines can be placed anywhere in the data file. Use comment lines with explanations liberally wherever necessary. When you look at these data files after a while it will be easier to comprehend the data.


Warning

Warning : Do not use blank lines as a substitute for comment lines to make the input data file more readable. Avoid blank lines which could lead to errors.

When experimenting with some alternatives some of the keywords and data lines can be taken out of execution by inserting ** in the first 2 columns. These can then be re-activated later by removing the **. This is easier than first deleting and then having to re-enter these lines again.

Figure 3 shows the simple example problem being analysed. Figure 4 shows the finite element mesh used for the analysis. Each element is a 4-noded quadrilateral.

The plate is 1 metre square. The units for length has been chosen to be metres. The force is chosen to be in Newtons. The Young's modulus is then specified in $N/m^2$ for consistency. The density is specified in $Kg/m^3$.

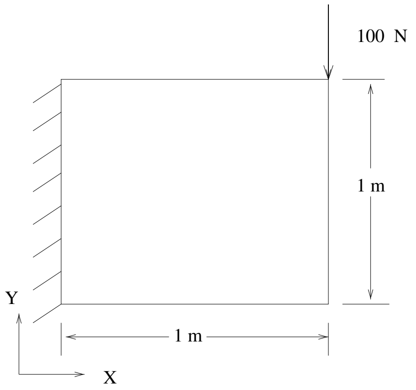



Figure 3: A cantilever example

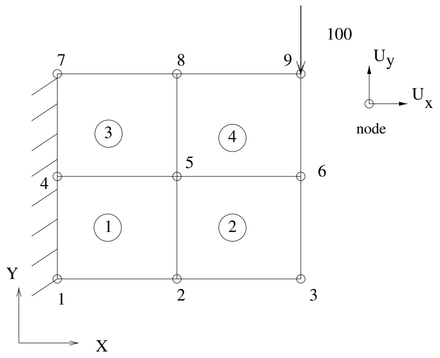



Figure 4: Finite Element Mesh

The first option is usually the header for the analysis : this is given on the *HEADING option. The next line will be taken as the heading and this will appear on each page of the printed output (*.dat file) when ABAQUS has run. It is a good practice to include a line explaining the units being used for length and force as part of the heading.

```
*HEADING
 SQUARE PLATE SUBJECTED TO POINT LOAD
** Job name: cantilever Model name: Model-1
** Units :  Length - m,  Force - N,  Stress - N/m2,  Time - Sec.
```

The next parameter *PREPRINT is optional.





```
*Preprint, echo=YES, model=YES, history=YES
```

By default this is omitted which is same as including it with all 3 options set to NO. This is useful for debugging purposes. More on this later.

In ABAQUS the finite element model is defined as an **assembly** of part **instances**. Part instances are the components of the assembly. Where **parts** are defined as the master components. The instances are copies of these.

This permits re-use of the parts. If an assembly consists of some components which are identical then these components have to be only defined once as a single part. Then there should be equal number of instances of this component representing each of these item.

For the example under consideration the assembly is made up of a single part with a single instance of that part.

```
*Part, name=plate
*End Part
```

This represents the part definition. It simply has the name **plate** defined. It is empty otherwise.

Next comes the assembly definition.

```
*Assembly, name=Assembly
**
*Instance, name=plate-1, part=plate
```

The assembly is given the name **Assembly** and this is followed by the instance definition. Here the instance is given a unique name **plate-1** and the part option identifies the part. It is a good practice to name the instance derived from the part name. This avoids any confusion. However if using CAE this will be done automatically as is the case with this example.

The *NODE option begins the specifying of the nodes and its co-ordinates.

```
*NODE
**  Node No.    X-coord      Y-coord
      1,         0.0,           0.
      2,         0.5,           0.
```

First number in each line is the node number and this is followed by the x and y co-ordinates. Gaps are allowed in the node numbering.

The *ELEMENT option deals with the element-nodal connectivity list. The element type is specified using the TYPE parameter.

The choice of element type is as important as any other aspect of a finite element analysis. The quadrilateral elements that can be used in a plane stress analysis are CPS4, CPS4I, CPS4R, CPS8 and CPS8R. Here C represent Continuum and PS represents Plane Stress. This is then followed by the number of nodes in the element. Optional R which follows represents Reduced integration.

For illustrative purposes the lower order 4-noded elements are chosen here. Because bending is involved (see the loading and boundary conditions) and because of the use of lower order element it is better to use the **incompatible mode** element. In general the higher the order of the element the more accurate will be the results for the same level of descretisation.

CPS4I is a 4-noded bilinear incompatible modes **Plane Stress** element.

```
*ELEMENT, TYPE=CPS4I
**  Element  Node-1   Node-2   Node-3   Node-4
      1,       1,       2,       5,       4
      2,       2,       3,       6,       5
```

First number in each line is the element number and this is followed by the node numbers listed in the anti-clockwise sequence. Gaps are permitted in the element numbering.

```
*Elset, elset=_I1, internal, generate
  1,  4,  1
```

This creates an element set labelled **_I1** and the parameter *generate* causes the elements 1 to 4 to be created with increments in element number being 1.

In the following input the elements in the element set **_I1** are specified to have SOLID SECTION with a thickness of 50 mms (in the direction normal to the paper) and they are also assigned the **STEEL** material properties which is defined below :

Plane stress and plane strain analysis deal with problems where the structure is subjected to *in plane* loading. In the current example if the loading is transverse ie normal to the plane of the plate then this ceases to be a plane stress problem. It becomes a three dimensional problem. Then you need to use **shell** (example : **S4R5, S8R5**) elements.

```
*Solid Section, elset=_I1, material=steel
**
** thickness
0.05,
```

The next statement completes the instance definition. In a more general problem other instance specifications follows this until all the instance specification is complete.

```
*End Instance
```

This is then followed by node sets and element sets that may be required for the analysis.

```
*Nset, nset=_G4, internal, instance=plate-1, generate
 1,  7,  3
*Elset, elset=_G4, internal, instance=plate-1
 1, 3
*Nset, nset=_G5, internal, instance=plate-1
 9,
```

The first of these creates a node set called **_G4** containing the nodes 1, 4 and 7. The generate parameter means the data line is interpreted as

*starting-node number, ending-node number, increment in node number*

The second groups the elements 1 and 3 into an element set called **_G4**. Notice that the same name can be assigned to node sets and element sets. There is no ambiguity here because the entities are different.

The last set consists of the node 9 and it forms the node set **_G5**.

The assembly specification is completed by the following statement :

```
*End Assembly
```

The material definition for this example is

```
*MATERIAL, NAME=STEEL
**
*DENSITY
7680.,
**
*ELASTIC
** Young's Mod   Poissons ratio
2.1E+11,         0.3
```





Here the material is identified by the name **STEEL**. It is isotropic (TYPE=ISO) which is the default option. 2.1E+11 is the Young's modulus in $N/m^2$ and 0.3 is the Poissons ratio. The density is 7680 $kg/m^3$. All keywords that follow the *MATERIAL option and introduce material behaviour (like *ELASTIC) define the properties of this material, until a keyword appears that does not define material properties (such as a *STEP option as in this example).

ABAQUS only uses the material properties that are relevant for the current step and ignores the rest. The material behaviour definition can therefore be as complex as needed :

```
*EXPANSION - thermal expansion

*PLASTIC

*DENSITY
```

This completes the **model** part of the data input for this example. Rest of the data comprises the **history** part.

The analysis consists of 2 steps. In the first step loading is applied (pre-load) and in the second step the eigenmodes and frequencies are calculated.

```
*Step, name=pre-load
pre-loading
*Static
**
**  Initial Time, Step Time, Min time and Max time.
1., 1., 1e-05, 1.
....
*End Step
```

The *STEP option begins the step; the *END STEP option ends it. A more complex analysis would have several steps, and might use the *RESTART option to divide up the analysis.

The line which follows the Step statement will be displayed in the CAE/Viewer for this step and acts as a label. This is specified when the STEP was created in CAE. Which shows that meaningful headers are preferable than the default labels/headers.

Next the boundary conditions are listed. The idea is to fix the left hand vertical boundary i.e. the nodes 1, 4 and 7 are fixed in both X and Y directions i.e. PINNED. These nodes form the node set _G4.

```
*Boundary
_G4, PINNED
**
```

The general form of the data input consists of the Node number or node set label, First d.o.f. constrained, Last d.o.f. constrained, the magnitude.

The following input represents this general form.:

```
*Boundary
**Node set, start DOF, fin DOF, value
_G4, 1, 2, 0.
**
```

Here PINNED is a short form. However both are equivalent. Some of the other useful acronyms are XSYMM, YSYMM ENCASTRE.

Next the loading is listed. Here CLOAD stands for Concentrated loads. In the line which follows, _G5 is the nodeset with the node 9 to which the load is applied, 2 the direction or degree of freedom and -100 the magnitude in Newtons. The minus sign indicates that the load is applied in the negative y direction.

```
** Name: pt-load   Type: Concentrated force
*Cload
** Node set,  dir  Force Magnitude
_G5,  2, -100.
**
```

More increments will be needed in an analysis with material non-linearity (eg. elasto-plastic behaviour) and/or geometric non-linearity. Then add the INC=*N* option, with *N* replaced by a suitable number of increments, to the *STEP statement.

Because the eigenmodes under the pre-loading state is required this has to be treated as a non-linear analysis with large deformation. For that include the NLGEOM parameter to the *STEP line. Then the geometric changes will be accounted for during this and subsequent steps. The INC parameter defines the number of increments to be 100 for this example. Increments is used to sub-divide the total loading in a step. This results in the load being applied in small increments (fractions).

For the following setting (time step of 0.01 for first increment) the first 1% of the total load is applied during the first increment in the step. Depending on the rate of convergence the load fraction is increased in subsequent increments until the total loading has been applied.

In the absence of the NLGEOM parameter the problem will be treated as linear elastic and the geometry is not updated (ie the nodal coordinates are not updated by the displacements that take place during the analysis) and in the second step the eigenmodes will be calculated based on the original geometry hence the loading applied will not have any effect on the mode or the frequency.

```
*STEP, AMPLITUDE=RAMP, NLGEOM, INC=100
*STATIC
   0.01, 1.0, 1.E-5, 1.0
```

The data consists of initial time increment (0.01), total time step (1.0), minimum (1.E-5) and maximum (1.0) time increment allowed.

```
*RESTART,WRITE
```

This creates the restart file (*.res) which is also used for re-starting the analysis. This record can be included anywhere but here it is included before the first *STEP record.

For a large analysis with many elements and steps the restart file can be very large (several MBytes). To reduce the size of the restart file add the frequency option.

```
*RESTART,WRITE,FREQUENCY=N
```

Where N can be a suitable number appropriate for the no. of increments used in that step. For example if you are using 100 increments in a step then N can be set to 10. The results will only be written to the restart file at the end of every 10th increment.

Alternatively if you are only interested in restarting the analysis at the end of the current analysis then you can use the following command :

```
*RESTART,WRITE,OVERLAY
```

This will write the results at the end of every increment after rewinding the restart file. At the end of the analysis the *.res file will contain only the results of the last increment. If the analysis is interrupted for some reason then it will contain the results of the increment which was completed just prior to the interruption. If necessary the analysis can be continued from the increment written to the restart file.

ABAQUS includes default output requests : these provide the necessary information of element variables (stress, strain, etc) and nodal variables (displacements, reaction forces) which are used for post-processing using the Viewer module.

In general users may request output of variables at locations (nodes, element integration points) that are of interest to them. The complete list of variable names can be found in section 4.2.1 of the USER's manual (Volume I).





A list of the PRESELECT variables for each type of procedure is listed in the table 4.1.3-1 in the USER's manual (Volume I).

Here part of the table for static and frequency procedures have been reproduced below:

| **Procedure Type** | **Preselected element variable (field)** | **Preselected nodal and surface variables (field)** | **Preselected energy variable written? (history)** |
|---|---|---|---|
| *STATIC | S, E, PE, PEEQ, PEMAG | U, RF, CF, CSTRESS, CDISP | yes |
| *FREQUENCY | | U | no |

This represents the information written the output database (*.odb file).

As can be seen there is no need to list the pre-selected variables. Additional output parameters can be added if necessary. For example CF. If you want to plot the force vs displacement at a given node.

```
*Restart, write, frequency=1
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
```

By default no results are written to the results file (*.res). This keeps the *.dat file to be compact. This is favouring visualisation of the results in terms of graphical plots (contours, deformed shape, graphs etc) and less of printing of several tables of stresses, strains and displacements which are hard to fathom at the best of time.

However there are times when debugging or carrying spot check of calculation for a benchmark problem when printed output is needed. For this case the following statements allows one to print out the results for checking purposes.

```
*NODE PRINT, FREQ=1, NSET=_G4
U
RF
```

This writes out the nodal displacements (U) and the reaction forces (RF) to the *.dat (printed output) file. The setting of the parameter NSET restricts the output to the nodes in that set.

```
*NODE PRINT, FREQ=1
U, RF
```

To print the displacements and the reactions in a single table use the above commands.

```
*NODE FILE, FREQ=1
U
RF
```

This writes out the nodal displacements (U) and the reaction forces (RF) to the *.fil file for post processing using PATRAN or FEMGV. Latest version of PATRAN (2003 R2) can also make use of the *.odb file (ABAQUS output database) for post-processing.

```
**
*EL PRINT, POSITION=INTEGRATION POINT, FREQ=1, ELSET=_G4
S
E
COORD
```

This writes out the stresses (S), strains (E) and co-ordinates (COORD) at integration points to the *.dat (printed output) file. It is also possible to write out the stresses and strains averaged at node by specifying POSITION=AVERAGED AT NODES. If using this option then omit COORD from the parameters to be written. Similarly interpolated values at element centroids can be printed using POSITION=CENTROIDAL.

Here again the output is restricted to the elements in the set _G4.

```
*EL FILE, POSITION=INTEGRATION POINT, FREQ=1
S
E
COORD
```

This writes out the stresses (S), strains (E) and co-ordinates (COORD) at integration points to the *.fil file for use with user written post processing programs. The number set by the FREQ parameter (1, in this example) is the output frequency in terms of **increments**.

Use the FREQ=0 option to suppress information being written to the *.fil file in the above example. Then the data lines are omitted.

The following statement completes the first step :

```
*End Step
```

Since we are interested in the eigenmodes of this example then a second step as shown can be included :

```
*Step, name=eigenmodes, perturbation
eigenmodes
*Frequency, eigensolver=Lanczos
5, , , , ,
 .....
*END STEP
```

The *FREQUENCY tells the program that this step is a frequency analysis (this should not be confused with the FREQUENCY parameter used in the output options where it refers to increment numbers) and the number 5 which follows means that the user is only interested in the first 5 eigenmodes.

The complete input data is listed below :

```
*Heading
 deformation-eigen
** Job name: cantiever Model name: Model-1
 Units :  Length - m,  Force - N,  Stress - N/m2,  Time - Sec.
*Preprint, echo=YES, model=YES, history=YES
**
** PARTS
**
*Part, name=plate
*End Part
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=plate-1, part=plate
*Node
      1,           0.,           0.
      2,          0.5,           0.
      3,           1.,           0.
```





```
      4,           0.,          0.5
      5,          0.5,          0.5
      6,           1.,          0.5
      7,           0.,           1.
      8,          0.5,           1.
      9,           1.,           1.
*Element, type=CPS4I
1, 1, 2, 5, 4
2, 2, 3, 6, 5
3, 4, 5, 8, 7
4, 5, 6, 9, 8
** Region: (Section-1:Picked)
*Elset, elset=_I1, internal, generate
 1,  4,  1
** Section: Section-1
*Solid Section, elset=_I1, material=steel
0.05,
*End Instance
*Nset, nset=_G4, internal, instance=plate-1, generate
 1,  7,  3
*Elset, elset=_G4, internal, instance=plate-1
 1, 3
*Nset, nset=_G5, internal, instance=plate-1
 9,
*End Assembly
**
** MATERIALS
**
*Material, name=steel
*Density
7680.,
*Elastic
 2.1e+11, 0.3
** ----------------------------------------------------------------
**
** STEP: pre-load
**
*Step, name=pre-load, nlgeom
pre-loading
*Static
1., 1., 1e-05, 1.
**
** BOUNDARY CONDITIONS
**
** Name: lhs-fixed Type: Symmetry/Antisymmetry/Encastre
*Boundary
_G4, PINNED
**
** LOADS
**
** Name: pt-load   Type: Concentrated force
*Cload
_G5, 2, -100.
**
```

```
** OUTPUT REQUESTS
**
*Restart, write, frequency=1
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
**
*NODE PRINT, FREQ=1
U
RF
*NODE FILE, FREQ=1
U
RF
**
*EL PRINT, POSITION=INTEGRATION POINT, FREQ=1
S
E
COORD
*EL FILE, POSITION=INTEGRATION POINT, FREQ=1
S
E
COORD
*End Step
** ----------------------------------------------------------------
**
** STEP: eigenmodes
**
*Step, name=eigenmodes, perturbation
eigenmodes
*Frequency, eigensolver=Lanczos
5, , , , ,
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=1
**
** FIELD OUTPUT: F-Output-2
**
*Output, field
*Node Output
U,
*Element Output
S,
**
*NODE PRINT, FREQ=1
U
*NODE FILE, FREQ=1
U
**
```





```
*EL PRINT, POSITION=INTEGRATION POINT, FREQ=1
S
E
*EL FILE, POSITION=INTEGRATION POINT, FREQ=1
S
E
*End Step
```

Data File : cantilever.inp

Figure 5 shows the organisation of a model defined in terms of an assembly of part instance. This organisation of the model matches the way models are created in ABAQUS/CAE.

See the section 2.9.1 of the ABAQUS User's manual (Volume 1) for a complete definition of an assembly.

## 2.3 Exercises

### 2.3.1 Exercise 1

What are the changes you need to make to the above input file if the loading is transverse ie normal to the plate in the z direction?

### 2.3.2 Exercise 2

Rerun the above example as a non-linear analysis with a higher load and see the effect it has on the frequencies in the second step.

Make the following changes.

```
*STEP, AMPLITUDE=RAMP, NLGEOM, INC=100
*STATIC
   0.01, 1.0, 1.E-5, 1.0

**
*CLOAD, OP=NEW
 _G5, 2,    -200.E6

*NODE PRINT, FREQ=10
U
RF
*NODE FILE, FREQ=10
U
RF
**
*EL PRINT, POSITION=INTEGRATION POINT, FREQ=10
S
E

*STEP, NLGEOM
Frequency analysis
**
*FREQUENCY
```

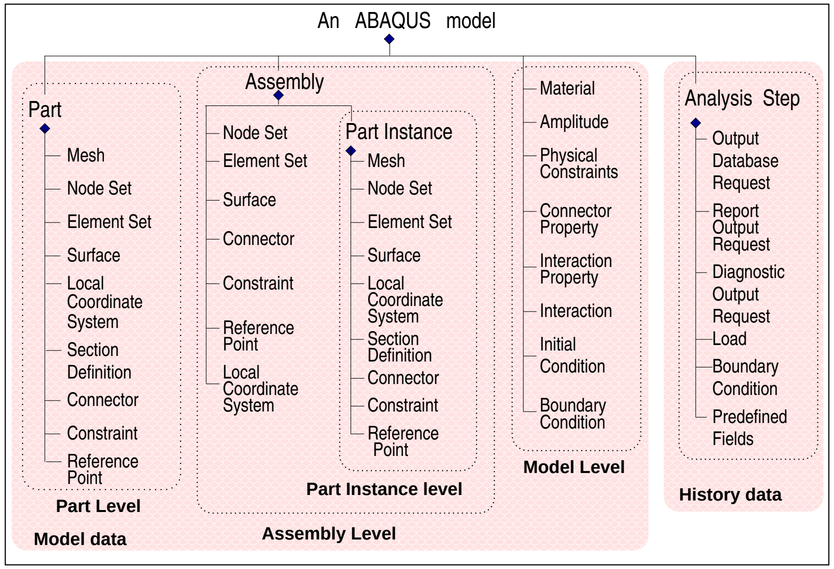



Figure 5: Model defined in terms of an assembly of part instance





## 2.4 Simple Mesh Generation Commands

ABAQUS has some simple mesh generation commands which can be used to generate meshes which are of a regular pattern i.e generated by repeating a basic unit.

Consider a beam which is modelled using five elements and six nodes. The end nodes are specified as follows :

```
*NODE, NSET=ENDS
   1,    0.,   0.
   6, 300.,   0.
*NGEN
   1, 6, 1
```

The *NODE option starts to read the node numbers and co-ordinates. The parameter **NSET** has been included with the name **ENDS**. This will place the nodes 1 and 6 in that group.

The *NGEN option generates the nodes 2 to 5 and calculates the co-ordinates assuming equal spacing between the nodes.

Similarly the elements are defined as follows :

```
*ELEMENT,TYPE=B21
  1, 1, 2              (element number and nodes at either end)
*ELGEN, ELSET=BEAMS
  1, 5, 1, 1
```

First number is the master element and the second number is the number of elements in the row. The third is the increment in the first node number from element to element and the fourth number is the increment in the element number.

B21 is a linear interpolation element for two-dimensional problems. *ELGEN will generate the element-nodal connectivity for all the remaining elements based on the information given with the *ELEMENT option. All 5 elements are identified by the same set name (BEAMS) which can then be used to specify the material properties.

See sections 2.1.1 and 2.2.1 of the ABAQUS User's manual for more examples on the use of NGEN and ELGEN commands.

*NFILL command can be used to generate the nodes within a region using the nodes along 2 lines demarcating that region. ??

*NMAP command allows one to map nodes from one co-ordinate system (say cylindrical) to another (say Cartesian). ??

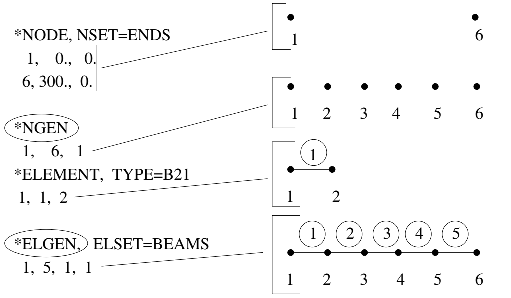



Figure 6: Mesh Generation Commands

# 3 Running ABAQUS

Before running ABAQUS, you will need to set up an input data file called *analysis*.inp (*analysis* is job-id and represents the name of the analysis, but it must end in .inp) contains the data for the program. Detailed instructions for setting up this data file is given in the ABAQUS USER's Manual. The input data file can be prepared interactively by running the CAE. See the ABAQUS CAE Users's Manual.

ABAQUS is licenced to run on the Teaching System Linux servers (DPO and EIETL). It is also available in the twn's (locally in the Windows OS). ABAQUS cannot be run on the gateway servers. After logging into the T.S. (Teaching System) Computers (HP-UX) first of all run a window manager (example : **twm&** ). This is optional.

To run a ABAQUS/Standard or Explicit job with the **job** parameter is set to the *analysis-id*. If you have the input data file called *cantilever.inp* then you can type the following to run the analysis:

**abaqus job=cantilever interactive**

This is recommended for small jobs and during the early stages of a new analysis. It will echo the different stages of the analysis and in case of errors will point to which file to look at. However it will tie-up the terminal until the ABAQUS job finishes. Also you cannot log out from that session until the job finishes. It writes out the information normaly written the the *.log file (cantilever.log file for this example) to the screen, No *.log file is created.

The alternative is :

**abaqus job=cantilever**

This will submit the job to the queue without any prompts. ABAQUS will then start running. You could monitor the progress of the job by typing **tail -10f cantilever.log** in a separate x-term. This will tie-up the x-term in which the command was typed. When the run has completed use CTRL/C to free that x-term.

ABAQUS will produce a number of output files during its run, these are listed in section 4.

Whenever you run an ABAQUS analysis look at the *.log file first (If you use the **interactive** option then the contents of that file will be directly printed to the screen and the *.log file not created). In case of errors it will tell you which file to look at.

Look at the *.msg file for possible errors and warning messages. In the absence of the *.msg look at the *.dat for any error messages and the end of the *.dat file for the summary. Note that *.dat can be large. Use the **tail** command to look at the last few lines as shown below :

**tail -50 cantilever.dat**

This will list the last 50 lines in that file.

If the name of the data file (*.inp) is known after consulting either the examples manual or the verification manual then the data file can be obtained using the **fetch** command. For example to get the input data for Detroit Edison pipe whip experiment input type

**abaqus fetch job=detroitedison**

This will fetch the detroitedison.inp file.

It is also possible to get a copy of the input file (using cut and paste) from the on-line copy of the ABAQUS Input files and the Verification Manual (using abaqus doc).

## 3.1 Running Batch Jobs

During term time and when running large ABAQUS jobs the users are recommended to use the **gridengine**. This will execute the ABAQUS jobs on one of the Teaching System Computers at a low priority at a later time when the demand is not high. For the above example type :

**batch -mbao -ttimelimit 3600 -QD "abaqus job=cantilever interactive "**

The timelimit is set in seconds (which is equivakent to 1 hour for this example). The progress of the submitted batch job can be monitored using the **batchq** command. Use the **batchrm** to delete the jobs (before these are run) if you subsequently change your mind. See the man pages on batchq, batchrm for further details. Example : type "man batchq".

The files created by a ABAQUS job will always be placed in the directory from which the abaqus job was submitted. However the batch command cannot be used if the input files are in **/tmp**.

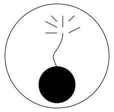




## 3.2 Running Large Jobs

Care is needed when running medium to large size ABAQUS jobs in the teaching system. Large ABAQUS jobs should only be run using the gridengine command. ABAQUS jobs can be cpu intensive and resources hungry. Give due consideration to other users in running ABAQUS jobs. The Computer Operators have the authority to terminate any ABAQUS jobs inconveniencing other users IMMEDIATELY. If in doubt check with them before starting a large ABAQUS job. For large ABAQUS jobs gridengine is the recommended choice. Report any problems to the Computer Operators (operators@eng).

In addition to the files created in the directory from which ABAQUS is run (*.dat, *.fil, *.res) solution files (files used in solving the FE equations) are created in the **scratch** directory. In the teaching system /tmp is the scratch directory. In general /tmp has about 9 GBytes available.



Warning

/tmp is a temporary workspace shared by many different programs while these execute and care is needed when using this region in running ABAQUS jobs. DO NOT use this as an extension to your home directory or to store large files over a long period which is likely to seriously inconvenience other users. If in doubt ask the computer operators.

Before starting a large analysis you need to find out the system requirements (memory and size of solution files). In order to obtain this information run the job with the datacheck option.

**abaqus job=cantilever datacheck**

In the *.dat file look for the heading **File sizes** under **Problem Size**. An estimate of the size of the total solution files will be listed. If these exceed the free (unused) space available in /tmp then the analysis could not be run.

To continue a job run using the datacheck option type

**abaqus job=cantilever continue**

If you are using a research system and if you want to override the default setting of /tmp then in the abaqus_v6.env file include the following statement and set the scratch parameter to a directory which has sufficient disk space.

**scratch="/aba_tmp"**

Here the directory name /aba_tmp is used as an example. Using this in the teaching system will cause the ABAQUS program to fail because such a directory does not exist.

If you are running very large analyses then the disk space in the **scratch** directory may not be sufficient. If this happens ABAQUS may issue messages similar to the following :

1. ***** Error : Sequential I/O error on unit 21.**

   **Out of disk space or disk quota.**

2. **scratch directory /: write failed, file system is full**.

   **direct access I/O Error on unit 2. Out of disk space or disk quota exceeded**.

Type **df /tmp** to find out how much unused disk space is available. If this is only a few Megabytes then /tmp on that particular cluster cannot be used. Report this to the Computer Operators giving the details of the cluster. /tmp is local to each cluster. Try slogging into another cluster where there is plenty of unused disk space in /tmp.

## 3.3 ABAQUS Environment File : abaqus_v6.env

The default setting of memory etc can be changed by setting alternative value in the abaqus_v6.env file. The settings for a particular job which has been submitted can be found in the *.com file.

For example to increase the memory from the default 2800 MB to 4000 MB include the following line in the *abaqus_v6.env* file.

If this file does not exist then create this file in the current directory using a text editor.

memory="4000 MB"

See section 3.4.1 of the User's manual (Volume I) for a complete list of the ABAQUS environment parameters. The current setting of these parameters can be found by typing

**abaqus info=environment**

In case of problems with the Graphics on a Windows XP System include the following line in the abaqus.env file.

**abaqus_no_hardware_acceleration=ON**

### 3.4 Checking for Errors

For small jobs or jobs which fail quickly use the **interactive** option on the command line. This will pinpoint which file to look at in case of errors. If you have not used that option then look at the *.log file. This is the quickest way of finding out whether any errors have been detected in the ABAQUS run. The *.log file will contain the same information which is written to the screen when the **interactive** parameter is used.

The absence of the *.odb file when you are expecting it to be created is also an indication that there are errors in the ABAQUS run.

If the *.msg file is missing then the errors must have been detected at the datacheck stage ie while running **pre.x**.

The *.dat file always has a summary at the end and will list a count of all the errors encountered. If the *.dat file had terminated abruptly without this summary and if there are no errors listed in the *.dat file then it could be that your disk quota has been exceeded while creating the various ABAQUS run time files. Check your quota by typing **quota -v**.

Delete all the old *.dat, *.fil and *.res files and any other unwanted files. Contact the computer operators if your disk quota needs to be increased.

If you are not going to use the restart option to continue an analysis then delete the *.mdl and *.stt files as well.

### 3.5 Killing an ABAQUS job

Care is needed in killing ABAQUS analysis jobs. There are two commands available. To suspend a submitted job after it has started executing, use

**abaqus suspend job=analysis**

here **analysis** should be the same for the previously submitted job ( whch is being suspended).

To kill the analysis, use **abaqus terminate job=analysis**. To resume a previously suspended job **abaqus resume job=analysis**. Do not routinely use the command **kill -9** <**process-id**> to kill currently running ABAQUS jobs. It should only be used as a last resort if other options to kill off any abaqus jobs have failed.

Here <process-id> is the process number of the ABAQUS job. The list of process ids (which would include the ABAQUS job) can be found using the command

**ps -eaf** | **grep** <**your-user-id**>.

### 3.6 Checking the License Server

To check whether the ABAQUS License Server is running use the **abalicence** command. This is not a part ABAQUS but is a local command which makes use of the lmstat command and consists of the line

**abaqus licensing lmstat -S ABAQUSLM**

### 3.7 Restarting an ABAQUS Run

The restart option is quite useful to break up a long abaqus run into a series of smaller jobs. Each such job will continue from the end of the previous job. This option is also useful if you want the change some parameter or rate of load application depending on the results obtained so far.

The following command illustrates the restart option available in ABAQUS.





The first part is run as follows :

**abaqus job=dynam**

Then create a data file called dynamr.inp and type the following:

**abaqus job=dynamr oldjob=dynam**

## 4 File extensions used by ABAQUS

The ABAQUS program generates several files. Some of these files exist only while ABAQUS is executing and are deleted as a run completes. Other files contain analysis, post-processing and translation results and are retained for use by other analysis options, restarting or post-processing. A summary of the essential files are listed below (the complete list can be found in section 3.5.1 of the User's manual) :

| | |
|---|---|
| cae | The database. |
| com | Analysis command file has the current job parameter settings. |
| dat | Printed output file. It is written by the **analysis, datacheck** and **continue** options. |
| f | User subroutine FORTRAN file. |
| fil | Results file. It is written by the **analysis** and **continue** options. (default binary) |
| inp | Analysis input file. It is read when the **analysis** and **datacheck** options are selected. |
| log | Log file. A log of all the abaqus analysis jobs. |
| mdl | Restart file ctrated by ABAQUS/Standard, for continuing an analysis. |
| msg | Message file. It is written by the **analysis** and **continue** options in ABAQUS/Standard. |
| odb | The results output database used by the **Viewer** module. |
| ps | ABAQUS/Viewer Postscript output file. |
| res | Restart file. Used for restarting analysis and post processing. (default binary) |
| sta | Status file. ABAQUS writes increment summaries to this file in the **analysis** and **continue** options. |
| stt | Restart file. |
| 023 | Communications file, used by ABAQUS/Standard only. This file is written to the same directory as the other user files (*.dat). Deleted when ABAQUS terminates. |

## 5 Printed Output

The *.dat file contains the printed output. If the

```
*PREPRINT, ECHO=YES, MODEL=YES, HISTORY=YES
```

option is used in the input file then the printed output consists of a listing of the data in the *.inp file.

It is good practice to establish a routine where the first thing you do after the analysis has been completed is to look at the *.dat file to see whether ABAQUS has interpreted the input parameters correctly. Experience has shown that some of the common mistakes can be attributed to either wrong input data or wrong interpretation of the input data by ABAQUS. this is only applicable if you have prepared the input file by hand. This is not so relevant if the input file was generated by CAE or other pre-processing program (eg. PATRAN).

If applicable a list of warning and error messages are printed. This is followed by several tables for each category of input recognised in the input file starting with the “Element definition”. This is followed by information relevant to the solution. This is terminated by the line

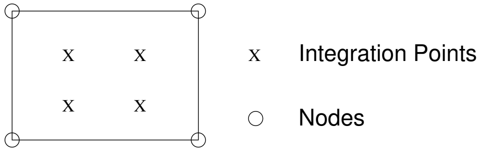



Figure 7: 2 x 2 integration scheme for CPS4I Element

END OF USER INPUT PROCESSING
This indicates the end of the pre-stage of the analysis.
Next comes the results from the analysis proper; the results output for information requested using the *NODE PRINT or *EL PRINT keywords in the *.inp file.
For the example problem under consideration this consists of stresses at element integration points (Fig. 7). The stresses at the 4 integration points at each element are printed. S11, S22 and S12 represent $\sigma_{xx}$, $\sigma_{yy}$ and $\tau_{xy}$ respectively for this element type (CPS4I).

| ELEMENT | PT | FOOT-NOTE | S11 | S22 | S12 |
|---|---|---|---|---|---|
| 1 | 1 | | -6615. | -1083. | -2450. |
| 1 | 2 | | -6615. | -285.2 | -2450. |
| 1 | 3 | | -2554. | -1083. | -2450. |
| 1 | 4 | | -2554. | -285.2 | -2450. |

This is followed by the strains ($\epsilon_{xx}$, $\epsilon_{yy}$ and $\epsilon_{xy}$) at the element integration points.

| ELEMENT | PT | FOOT-NOTE | E11 | E22 | E12 |
|---|---|---|---|---|---|
| 1 | 1 | | -2.9952E-08 | 4.2918E-09 | -3.0338E-08 |
| 1 | 2 | | -3.1092E-08 | 8.0920E-09 | -3.0338E-08 |
| 1 | 3 | | -1.0614E-08 | -1.5098E-09 | -3.0338E-08 |
| 1 | 4 | | -1.1754E-08 | 2.2904E-09 | -3.0338E-08 |

The integration point co-ordinates (x, y) are printed next.

| ELEMENT | PT | FOOT-NOTE | COORD1 | COORD2 |
|---|---|---|---|---|
| 1 | 1 | | .1057 | .1057 |
| 1 | 2 | | .3943 | .1057 |
| 1 | 3 | | .1057 | .3943 |
| 1 | 4 | | .3943 | .3943 |

In the next table the displacements are printed at the nodes ($u_x$, $u_y$) which are free ie not restrained.

| NODE | FOOT-NOTE | U1 | U2 |
|---|---|---|---|
| 2 | | -1.8800E-08 | -2.5188E-08 |
| 3 | | -2.3296E-08 | -4.9401E-08 |
| 5 | | -2.0527E-09 | -2.1897E-08 |
| 6 | | -2.7304E-09 | -5.5466E-08 |
| 8 | | 2.1645E-08 | -2.0987E-08 |
| 9 | | 3.4694E-08 | -7.7596E-08 |





This is followed by the reaction forces at nodes which are restrained

| NODE | FOOT-NOTE | RF1 | RF2 |
|---|---|---|---|
| 1 | | 102.6 | 42.06 |
| 4 | | -5.178 | 19.78 |
| 7 | | -97.41 | 38.16 |

For the second frequency step the eigenvalues and frequencies are listed for each mode that had converged.

E I G E N V A L U E   O U T P U T

| MODE NO | EIGENVALUE | FREQUENCY (RAD/TIME) | FREQUENCY (CYCLES/TIME) | GENERALIZED MASS | COMPOSITE MODAL DAMPING |
|---|---|---|---|---|---|
| 1 | 1.10800E+07 | 3328.7 | 529.77 | 156.71 | .00000E+00 |
| 2 | 6.44741E+07 | 8029.6 | 1277.9 | 165.92 | .00000E+00 |
| 3 | 6.46343E+07 | 8039.5 | 1279.5 | 147.12 | .00000E+00 |
| 4 | 8.48167E+07 | 9209.6 | 1465.8 | 137.42 | .00000E+00 |
| 5 | 1.44871E+08 | 12036. | 1915.6 | 200.78 | .00000E+00 |

This is followed by the stresses, strains and nodal displacements for each eigen mode.

# 6 Running ABAQUS/Viewer

ABAQUS/Viewer is an interactive post-processor for ABAQUS. It reads the output database file (*.odb) generated by a ABAQUS run, and allows the user to plot the results as views of the deformed mesh, contour plots, variable-against-variable plots, XY plots. The plots generated using ABAQUS/Viewer may be saved to a file in a number of formats or directly printed.

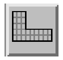
## 6.1 General

Type **abaqus viewer** to run ABAQUS/Viewer.
Type **abaqus viewer database=plate1** to run ABAQUS/Viewer with plate1.odb.
To quit from ABAQUS/Viewer, from the main menu bar select **File** → **Quit**.

## 6.2 Mesh Plots

From the main menu bar , select **Plot** → **Undeformed Shape**, or use the the tool in the toolbox (shown below).
To display the mesh with node and element numbers

- choose **Options** → **Undeformed Shape**. The **Undeformed Shape Plot Options** dialog box should appear.



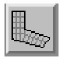
Undeformed Mesh Plot Icon

- Click the **Labels** tab. Toggle on **Show node labels**. Toggle on **Show element labels**. Click on **Apply**.

  This should display the mesh with node and element numbers.

To plot the deformed mesh

- From the main menu bar, select **Plot** → **Deformed Shape**, or use the icon shown below.



Deeformed Mesh Plot Icon

To plot the deformed shape superimposed over the undeformed mesh

- In the **Deformed Shape Plot Options** dialog box, toggle the **Superimpose undeformed plot**. Then click on **OK**.

By default, the undeformed mesh is coloured green and and the deformed mesh is coloured white. To change the the displacement scale factor :

- choose **Options** → **Deformed Shape**.
- In the **Deformed Shape Plot Options** dialog box click on the **Basic** tab.
- From the **deformation scale factor** region, toggle on **Uniform** and enter 1.0 in the **Value** field. Click on **Apply**.

  This should display the actual displacement.
- Now change the value to 1.E6 and then click on **Apply**.

To get a displacement vector plot :

- From the main menu bar select **Result** → **Field Output**. In the **Field Output** dialog box Select **U** from the list **Output Variable** and then click on **Magnitude** in the **Invariant** table.
- This will ensure that the displacement vector (both components) is selected. Click on **OK**.

  If a dialogue box called **Select plot Mode** appears then choose the toggle option ◇ **As is** and click on the **OK** button.

  This should draw the displacement vectors in Red.

## 6.3 Displacement Plots and Getting Hard Copies

A hardcopy of any generated plot can be obtained by directly printing to a Postscript printer. Alternatively the plot can be saved in a Postscript (PS), Encapsulated postscript (EPS), Tag Image File Format (TIFF), or Portable Network Graphics (PNG) file.

To get a hard copy of the plot from the main menu bar select

- **File** → **Print**

In the print command line type : lp -oraw -d<printer-name>
Example : lp -oraw -dljmr1





## 6.4 Contour Plots

To plot the Contours of von Mises stress

- From the main menu bar select **Result** → **Field Output**.
- In the **Field Output** dialog box Select **S** from the list **Output Variable** and then click on **Mises** in the **Invariant** table. Alternately click on the **Plot Contours** icon. Click on **Apply**.

  Sometimes it might be necessary to produce a black and white output where instead of banded plot only lines with labels or different linetypes are required.
- In order to obtain such a plot click on the **Contour Options** which can be seen just below the bottom right hand corner of the Viewport. In the **Contour Plot Options** dialog box click on the **Basic** tab. In the region marked **Colour type** toggle on the **Line** option.
- Then click on the **Color & Style** tab and select the **Line** tab. This option is shown in Figure 8.

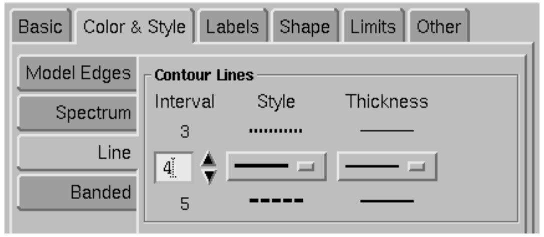



Figure 8: Option to change contour type

- Under the heading **Contour Lines** the sub-headers are **Interval** / **Style** and **Thickness**. The middle row has arrows pointing upwards and downwards. Set the interval to be 1 (by clicking on the appropriate arrow) then set the style and thickness.
- Then increment it by 1 and set the style and/or thickness to be different. Once the required number of intervals have been set click on **Apply**.

This should display the required contour plot with lines of different types.

## 6.5 PATH Plots

Consider taking a section along the left hand vertical edge and then plotting the variation $\sigma_{yy}$ along this line. This is called a PATH plot where the line section represents the path.

- From the main menu bar select **Tools** → **Path** → **Create**. In the **Create Path** dialog box Type **vertical** for the Name and then toggle on **Point list** and then click on **Continue...**.

  Here there is a choice of inputting the node numbers to define the line position or defining points (by specifying the co-ordinates). Here the latter option is chosen.
- In the **Edit point List Path** enter the co-ordinates 0., 0. on the first line and press the RETURN Key. Then enter 0., 1. for the 2nd point and press the RETURN Key. Then click on **OK**.

  The path has now been defined. Next step is to select the variable to plot. There are 2 ways of selecting the variable for the plot. The first way is same as selecting a variable to draw contours :

- From the main menu bar select **Result** → **Field Ouput**. In the **Field Ouput** dialog box Select **S** from the list **Output Variable** and then click on **S11** in the **Invariant** table. This selects the variable $\sigma_{xx}$.

  The next step is to generate the path plot. The second method of selecting the variable is included in this.

- From the main menu bar select **Tools** → **XY Data** → **Create**. In the **Create XY data** dialog box toggle on ◇ **Path** and then click on **Continue...**.
- In the **XY Data from Path** dialog box make sure **Path** is set to **vertical**, the previously created path. In the region marked **Y Values** Click on **Field Output...** and select **S** as the output variable and **S11** as the Component.
- This is where a different variable can be selected, subsequently. Click on **OK**.
- Then in the original dialog box click on **Save As...** and give it a name (example : s11 on vertical). Then click on **Plot**. This should display the Path plot in the main viewport.
- If the **Select Plot Mode** dialogue box appears then toggle on ◇ **As is** and then click on OK.

Once a path has been defined it is possible to plot the variation of a different variable (example : Mises stress) along this path.

## 6.6 X-Y Plots

In the current analysis the loading was applied in a single increment. In order to illustrate the X-Y Plots the analysis will have to re-run as a non-linear analysis with several increments.

The following changes are made to the second step. Changes to the output is also made to illustrate some of the features available.

```
**
** STEP: pre-load
**
*Step, name=pre-load, nlgeom
pre-loading
*Static
0.01, 1., 1e-05, 0.1

.......

**
** LOADS
**
** Name: pt-load   Type: Concentrated force
*Cload
_G5, 2, -200.E6
**

.......

**
*Output, history, frequency=1
*node output, nset=_G5
  CF2,U2
*element output, elset=_G5
E22, S22
```





```
**
*monitor, node=_G5, dof=2
```

The initial time step is changed to 0.01 from 1.0 and the **nlgeom** parameter has been added to the Step line.

The loading has been increased to 200.E6 to show that the displacements will be sufficient to illustrate the non-linear effect.

Further history information is added to the output in order to generate a stress-strain plot.

The analysis is run again with the modified data. Once the analysis is completed access the odb database as before.

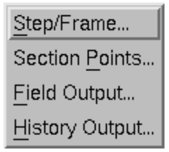



Figure 9: Option to change contour type

- From the main menu bar select **Result** → **History Ouput**.
- In the **History Ouput** dialog box Select **Spatial displacement : U2 at node 9** from the list **Output Variable**.
- Click on **Save As...** button. Enter **yDisp-9** in the name box in the **Save XYData As** form. Then click on **OK**

  Notice the note : XYData are only saved for the current ABAQUS session.

- Click on **Plot** in the original dialog box.
- Repeat these steps for the **Output Variable** with the description **Point loads: CF2 at Node 9**. Save this as **yForce-9**.

## 6.7 Creating a Force Displacement Plot

Combining the plots created in the above section the Force-Displacement plot can be created as follows :

From the main menu bar

- select **Tools** → **XY Data** → **Manager**.
- In the **XY Data Manager** dialog box click on **Create...**.
- In the **Create XY Data** dialog box toggle on **Operate on XY data** and then click on **Continue...**.

  The **Operate on XY Data** dialog box appears. This contains the following lines :

  The XY data field on the left has a box for entering an expression. Below that a list of existing X-Y data objects (this should have the 2 plots created in the above section).

  The operators's field on the right contains a list of all the possible operations you can perform on the data object.

- From the operators field click on **combine(x,x)**.

  **combine ( )** appears in the expression box.

- Double click on the XYData **yDisp-9** and this will appear within the brackets.

- Type a comma "," and then double click on the XYData **yForce-9**.

  The expression box should look like :

  **combine ( " yDisp-9" , " yForce-9" )**
- Then click on **Save As...** button and name the plot as **Force-Disp-Nde9**. Click on **OK**.
- Then Click on **Plot Expression** button and this should display the Force-Displacement plot.

### 6.8 Writing Out Results from X-Y Curves to an External File

This section deals with how to write out the results from an X-Y Curve. Example the force displacement plot results created in the previous section.
From the main menu bar

- select **Report** → **XY ...** .
- In the **Report XY Data** dialog box toggle on ◇ **XY Plot in the Current Viewport**. That plot will be highlighted.
- Click on the **Setup** tab and the name of the file to which the results will be written to is displayed as **abaqus.rpt**. This is the default fiel name. If the current results are to be appended to the existing contents make sure the toggle **Append to File** is ON.
- It is also possible to control the format of th results from the settings in this dialogue box.
- Once you have altered the settings to suit your requirement click on OK.

  Use a text editor to check the contents of the file. It should be straightforward to import this into a spreadsheet and produce plots comparing either with available experimental or theoretical results.

## 7 Post Processing using Third Party Programs

### 7.1 General

Once the *.odb file is created one can use ABAQUS/Viewer to post-process the results. PATRAN may not recognise some of the specialist element types (piezoelectric, semi-infinite).

For more general problems the *.odb can be used for post processing using PATRAN.

See the next section on how this is done.

### 7.2 Post Processing using PATRAN

See the PATRAN beginner's guide on how to use the PATRAN program to post-process the abaqus results stored in the *.fil file. Type **patran** to start running PATRAN.

If you had used the PATRAN program to generate the *.inp file in the first place then section 10 of the above guide explains how to generate the plots on the screen and how to produce the hard copies of the generated postscript files.

However if the *.inp file was not created using PATRAN then you need to follow the instructions in section 2 "How to start a PATRAN session" of the above guide. Create a new database of the same name as the analysis you have just run. Change the template to **abaqus.db** and then follow the instructions given in section 9 of that guide. But choose 'Read both' instead of 'Read Results' for 'action'. This means that instead of reading only the 'results' you need to read both the 'results' and 'model data'. This is because PATRAN needs the information about the model (mesh) as well as the mesh was not generated using PATRAN. Then follow the instructions given in that guide.





## 8 Troubleshooting

### 8.1 No Message File

ABAQUS run has finished and the *.msg file is missing. Error must have occurred during the pre stage of the ABAQUS run. This may be due to errors in data detected by ABAQUS.

The *.dat file does not have a summary at the end. Probably your disk quota has been exceeded. Type **quota -v** and check your quota.

See also section 3.3.

### 8.2 Blank pages instead of Plots for hardcopy

ABAQUS CAE/Viewer is producing blank pages when plots are sent to the DPO printers ljmr1, ljmr2. Saving the plots as postscript files and then printing also gave blank pages.

Add the ' -oraw' option to the print command.

Example : lp -oraw -ljmr1 abaplot.ps

For more information see the ABAQUS web page under Computing / Help.

### 8.3 Bugs

Up-to-date information on any bugs in the ABAQUS program can be found in the status reports. See the ABAQUS Ltd Home page. Search for the occurrence of the keyword you are interested in.

These error numbers will be preceded by **v69_** to indicate in which version the bug was found. If the bug had been fixed you will find in which version the bug had been fixed. The versions installed on the teaching system is version 6.9-1 and 6.9EF. The version available in the teaching system is updated once a year.

## 9 Frequently Asked Question (FAQ) and Other Information

In the CUED web page look under **Help / Engineering Packages / Finite Elements / ABAQUS** for a list of Frequently Asked Questions (FAQ) and answers. Work is currently under progress to set this up.

Consult the SIMULIA home page (accessible from the above page) regarding ABAQUS newsletters and product development information.

### 9.1 On-line Documentation

On-line documentation available with ABAQUS can be accessed by typing

**abaqus doc**

Ignore any warning messages that appear on the window from which the above command was invoked.

Then click on the "Using ABAQUS Online Documentation" under the Book Titles section. Then click on **File / Open Book** and read any relevant sections. Users are recommended to read this part before proceeding further. Select the section from the Left Hand Side (LHS) window.

See section 3.2.4 on usage of the on-line document.

On-line documentation can also be accessed from ABAQUS/CAE. Start up CAE and then look at the **Help** menu options.

#### 9.1.1 Keywords

Click on "ABAQUS/Standard Keywords" and then click on **File / Open Book**.

There are two ways of using the Keyword part of the on-line documentation. First is to use the keyword search option. Type in the keyword on which you want information in the box marked "keyword". Then click on the box marked "Find". This is the most useful if the keyword you are searching for is likely to appear in a number of different locations. If using more than one word separate these by '-' (hypen) instead of blanks. Also watch out for American spelling of words.

The second choice is if all the information is likely to be grouped together under a single heading. For example if you want to look at the information for the Drucker-Prager Model then select the character **D** from the left hand side window. All the keywords are grouped under the letters of the alphabet. Clicking on D displays the start of the sections under the heading D on the RHS.

Click on the ( ) symbol next to D and this will open up and list all the keywords which start with the letter D. Click on the right pane (to bring it into focus) and then use the Page down/up keys to scroll. Alternatively use the vertical slide bar next to the left hand side window to scroll down until you come across Drucker-Prager. Clicking on that heading will display the relevant section on the right hand side. The various sections under this heading can be displayed by clicking on the ( ) sign next to Drucker-Prager on the LHS window.

### 9.1.2 User Manual

Click on "ABAQUS/Standard User's Manual" and then click on **File / Open Book**.

In this case the heading of the 11 parts (see below) will be listed on the left pane. In the right pane you will find hypertext links ie words which are underlined. When you click on these words it will jump to the appropriate section in the manual. Because of the numerous links one could easily get lost or go around in circles. Therefore it is important to be aware of the relative position within the manual. For this reason the chapter headings are listed below. The ABAQUS Analysis User's manual consists of 5 volumes. The whole manual is divided into 9 parts. Each part in turn is divided into chapters.

When clicking on any links be patient. If equations and figures are present in the page to which you are jumping to it might take a while for it to display it. By clicking on the ( ) one can open up the subheadings.

Subheadings and subsections can be closed by clicking on the appropriate ( ) sign.

To quit from the on-line documentation choose **File /Quit** from the first window.

**—————- Volume 1 ——————**

**Part 1 : Introduction, Spatial Modeling and Execution**
1. Introduction
2. Spatial Modeling
3. Execution Procedures

**Part 2 : Output**
4. Output
5. File Output Format

**—————- Volume 2 ——————**

**Part 3 : Analysis Procedures, Solution and Control**
6. Analysis Procedures
7. Analysis Solution and Control

**Part 4 : Analysis Techniques**
8. Analysis Techniques Introduction
9. Analysis Continuation Techniques
10. Modelling Absractions
11. Special-Purpose Techniques
12. Adaptivity Techniques
13. Eulerian Analysis
14. Extending ABAQUS Analysis Functionality
15. Design Sensitivity Analysis
16. Parametric Studies

**—————- Volume 3 ——————**





**Part 5 : Materials**
17. Materials : Introduction
18. Elastic Mechanical Properties
19. Inelastic Mechanical Properties
20. Progressive Damage and Failure
21. Hydrodynamics properties
22. Other Material Properties

**—————- Volume 4 ——————**

**Part 6 : Elements**
23. Elements : Introduction
24. Continuum Elements
25. Structural Elements
26. Inertial Rigid and Capacitance Elements
27. Connector Elements
28. Special Purpose Elements

**—————- Volume 5 ——————**

**Part 7 : Prescribed Conditions**
29. Prescribed Conditions

**Part 8 : Constraints**
30. Constraints

**Part 9 : Interactions**
31. Defining Contact Interaction
32. Contact Property Models
33. Contact Formulation and Numerical Methods
34. Contact Difficulties and Diagnostics
35. Contact Elements in ABAQUS/Standard
36. Defining Cavity Radiation