

# Abaqus Protocol for Finite Element Analysis

(For Abaqus/CAE 6.14)

Dr. Alana C. Sharp, Evolutionary Morphology and Biomechanics lab, Institute of Life Course and Medical Sciences, University of Liverpool, UK

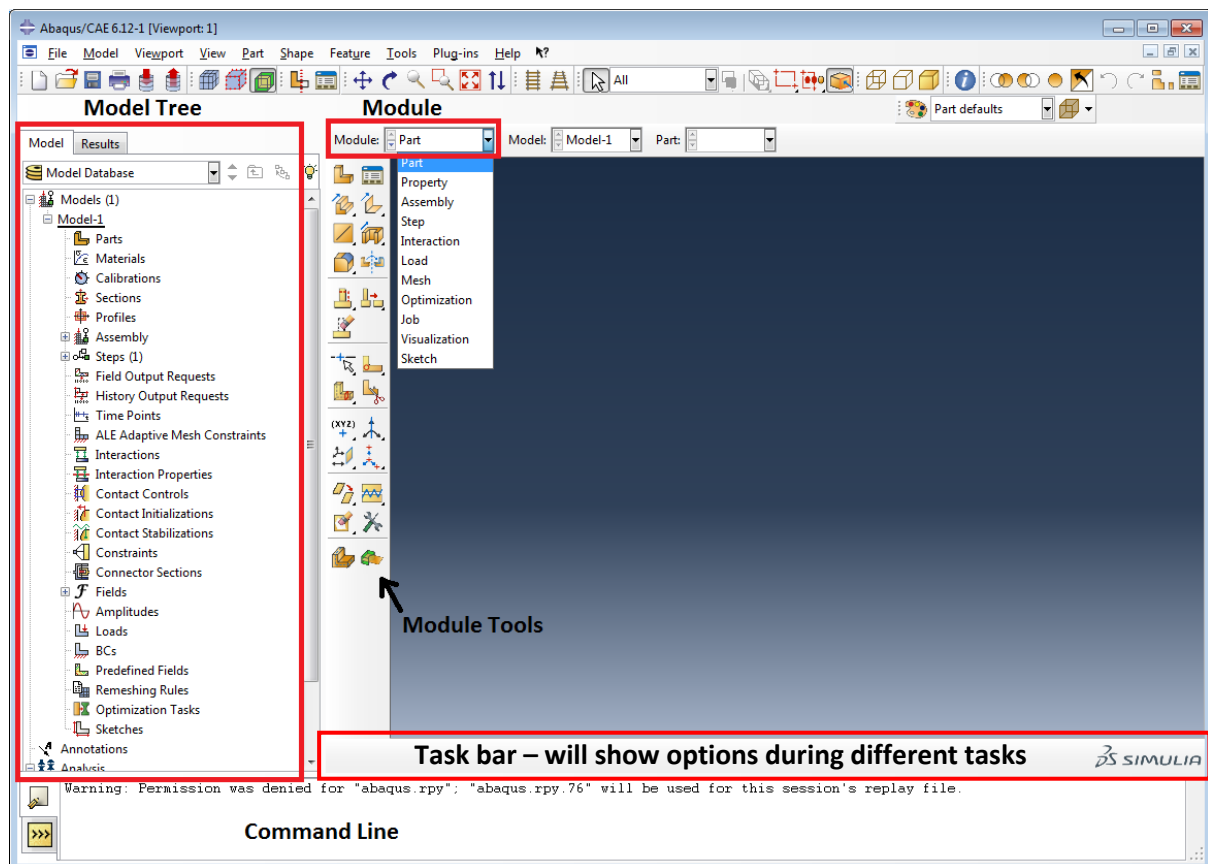
Alana.sharp@liverpool.ac.uk

March, 2021

## Introduction

This is a step-by-step protocol to set up a simple, static, example loading scenario including how to assign material properties, boundary conditions (constraints and loads) and run jobs. There are various ways to set constraints (e.g. single nodes, coupling constraints etc.) and loads, and this protocol provides a couple of examples but by no means is exhaustive and I'm sure there are many other ways to do it! But hopefully this will help you get started. There are also different types of loading scenarios, such as thermal expansion, which are not described here.

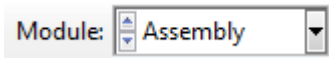

## The User Interface



The model tree is located on the left hand side of the window. This is the main place to find all the tools to create boundary conditions, loads, sets, materials etc.

The Command Line Interface is located at the bottom of the window and shows recent activity that you have performed.

The menus available in the top toolbar will change depending on the Module you have selected. The Module can be changed by selecting from the dropdown list

. The module will change automatically when you select different tasks from the Model tree. For example, when you want to apply a load by double clicking  Loads in the model tree, it will automatically switch to the Load Module.

The Module tools, located to the right of the model tree will also change depending on the Module selected.

## The basics

Pan: Ctrl + Alt + Middle mouse button (MMB)


Rotate: Ctrl + Alt + Left mouse button (LMB)

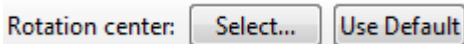
Zoom: Mouse wheel

Select: LMB

Add to selection: Shift + LMB



Deselect: Ctrl + LMB

Change rotation centre: Click  in the toolbar > then click Select in task bar



Change the Perspective:  

Change the background colour : View > Graphics Options... In Graphics Options you can also change the Drag mode to wireframe for faster movement.

Take a screen capture: File > Print, in settings - select File, Browse files  , select format and choose the image size by clicking .

**Set working directory:** You may want to change the working directory (the default is C:\Temp), File > Set work directory...

To change the element type (linear to quadratic): In the Mesh Module, select the menu Mesh > Element Type... then select the region to be assigned. In the dialog box that appears, change the Geometric Order (Linear or Quadratic). Linear are 4-node tetrahedral elements (C3D4) and quadratic are 10-node tetrahedral elements (C3D10). Quadratic elements are more accurate but take a lot longer to solve. 4-node tetrahedral elements are standard from Avizo and do the job if you have enough of them.

## Step-by-step Instructions

### 1. Import a model:

Before importing a model into Abaqus it is important to make sure your mesh is of good quality. The better quality your surface mesh is in Avizo (or whatever meshing software you use – Hypermesh, Mimics, 3Matic, Blender etc.), the better quality the tetrahedral mesh will be (for producing a high-quality mesh in Avizo, see my Avizo protocol). If the mesh has errors, such as high aspect ratio triangles, it may not run in Abaqus, which means you will have to start all over again and produce a better mesh. Get it right the first time and never look back!

Abaqus can import many file types; I have only used an Abaqus Input file (\*.inp) and NASTRAN, exported from Avizo. Importing a \*.inp model from Avizo or Hypermesh will set up a Part and an Instance for the Assembly. If you have multiple materials in your mesh that you want to apply different material properties to in Abaqus, it is better to save it as a NASTRAN file, because this will set up different sets, sections and materials when you import it to Abaqus.

File > Import > Model, select file type Abaqus Input File (\*.inp) or NASTRAN Bulk file (\*.bdf, \*.dat, \*.nastran etc.) and import the model.

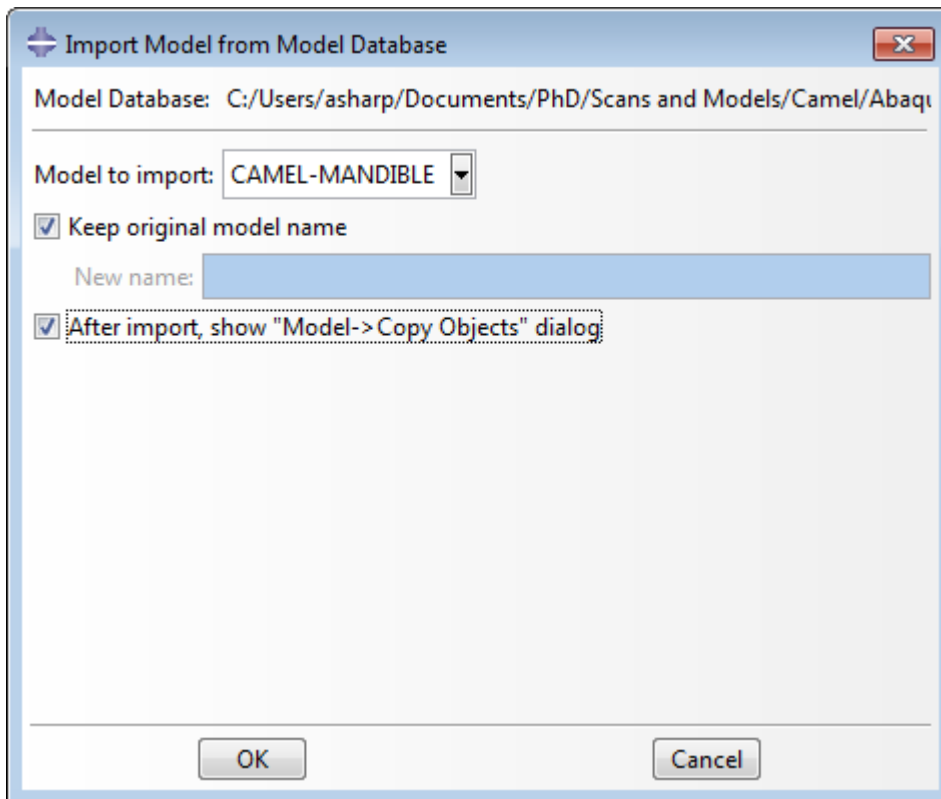
When the model imports, it will set up one or more Sets found under Part and/or Assembly in the model tree. A couple of important ones are: EALL, a set of all the elements; NALL, a set of all the nodes, because these tell (remind) you how many elements and nodes are in your model. These may be different in different version of the software, or depending on the software from which you export your model. When you hover your mouse over a Set it will tell you the type (element or node) and size.

### 2. Importing a second model (Part):

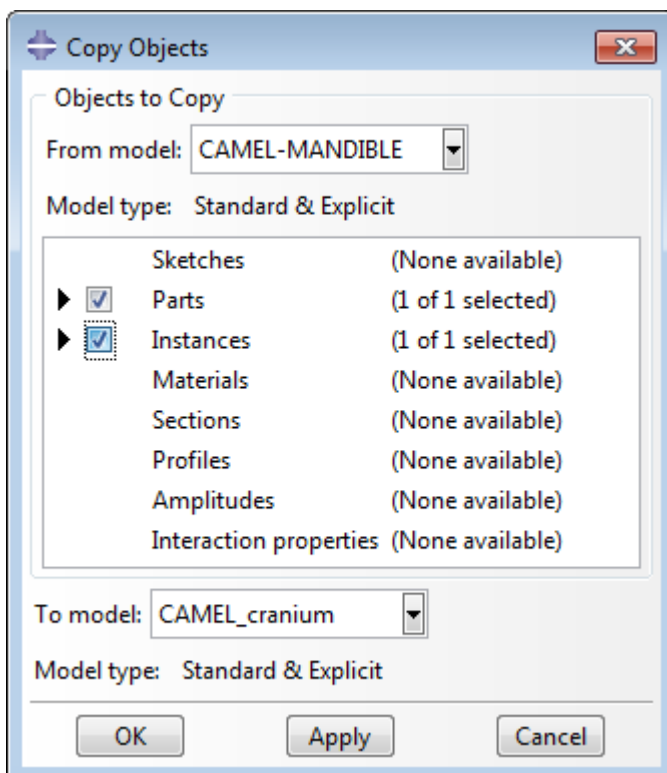
Importing a second Part and Instance, such as a jaw, can be done separately so that the jaw can be excluded from the simulation. The jaw can then be used to correctly align the loads. Import the second model as above in a separate Abaqus Database and save the database in the Abaqus/CAE Database format (\*.cae). Then, in the Abaqus database that you want to work in (e.g. set up simulations and run FEA on the cranium) import the jaw model as follows:

File > Import > Model, select file type Abaqus/CAE Database (\*.cae)

In the dialog box that appears, select *After import, show "Model->Copy Objects" dialog*

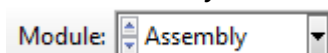




Another dialog box will appear. Select the Part and Instance you want to copy



When the new model has imported, it might need to be realigned.

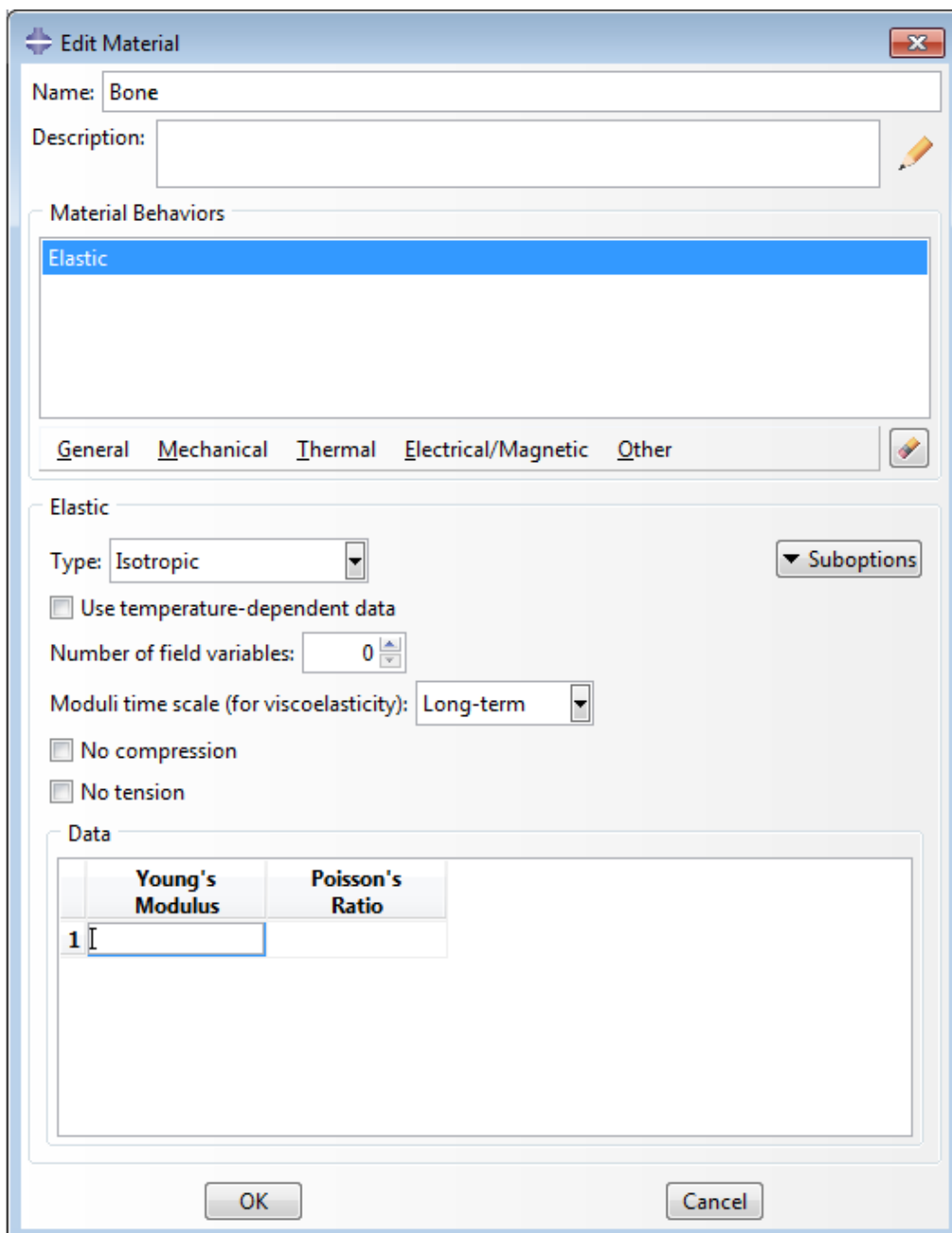
Select *Assembly* in the drop down menu next to Module in the toolbar



To translate an Instance with respect to another, select the Translate Instance tool  then select the Instance to be moved. To rotate the Instance, select the Rotate Instance tool .

### 3. Create Materials:

Under the Model tab on the LHS of the screen expand your selected model (you can also delete *Model-1* and any other models you don't need). In the model tree, double click on Materials. Name your material and select the type of material it is e.g. Elastic, Isotropic. Enter the Young's Modulus (e.g. 20E9 for 20 GPa) and Poisson's Ratio (e.g. 0.3) in the Data section window (see below). If a material is already created with your imported model (\*.inp), you can double click on the material to edit it and add its properties.



**Edit Material**

Name:

Description:

**Material Behaviors**

Elastic

General Mechanical Thermal Electrical/Magnetic Other

**Elastic**

Type:  Suboptions

☐ Use temperature-dependent data

Number of field variables:

Moduli time scale (for viscoelasticity):

☐ No compression

☐ No tension

**Data**

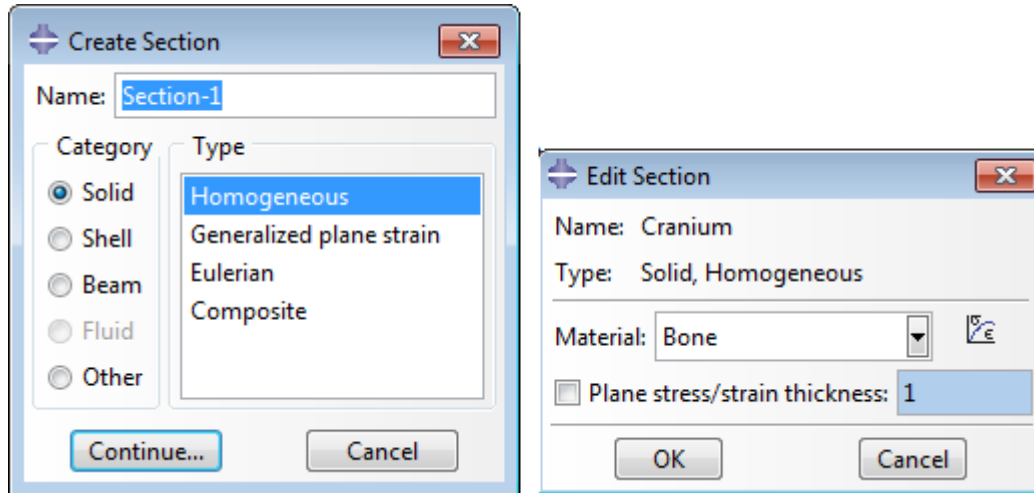
	Young's Modulus	Poisson's Ratio
1	<input type="text"/>	<input type="text"/>

OK Cancel

#### 4. Create a Section that you want to assign the material:

If a Section has already been set up when you import your model, you don't need to do this step. But, it's important that you make sure the Section is "Solid, Homogeneous", and assigned to the correct material.

Double click on the Section option in the model tree, name your section and select the Category (e.g. Solid) and Type (e.g. Homogeneous), click Continue...



A second dialog box (above right) will appear to select the material for the section.




#### 5. Assign the Section to your model:

If a Section Assignment has already been set up when you import your model, you don't need to do this step, but it is best you check that the section you want it assigned to the correct region/part and material.

Expand the Part you want to assign the section to and double click the Section Assignments option.

Uncheck the Create Set option in the task bar. You can select the region individually (each element, or select the entire model) using the mouse, or you can select the region using a set you have already created (or a set created for you if you have multiple materials) by clicking the "Sets..." button on the right. Then click Done.

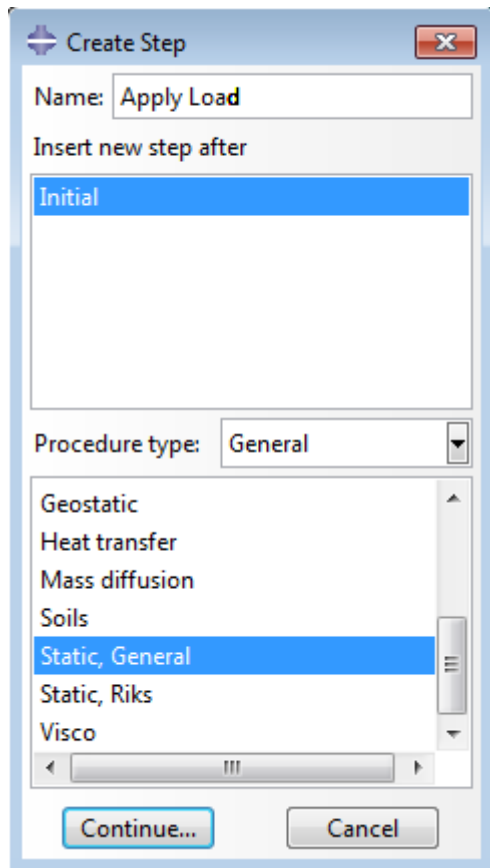


Note: When making a selection of the model, it is import to have the right options chosen. In the top toolbar there is an option to select ALL elements/nodes , just the EXTERNAL elements/nodes  or just the INTERNAL elements/nodes . When assigning the section individually, you need to have ALL elements selected. Later when you want to make Sets of surface nodes (e.g. for muscle attachments), you will need to change this to just the EXTERNAL nodes.

## 6. Create an analysis Step:

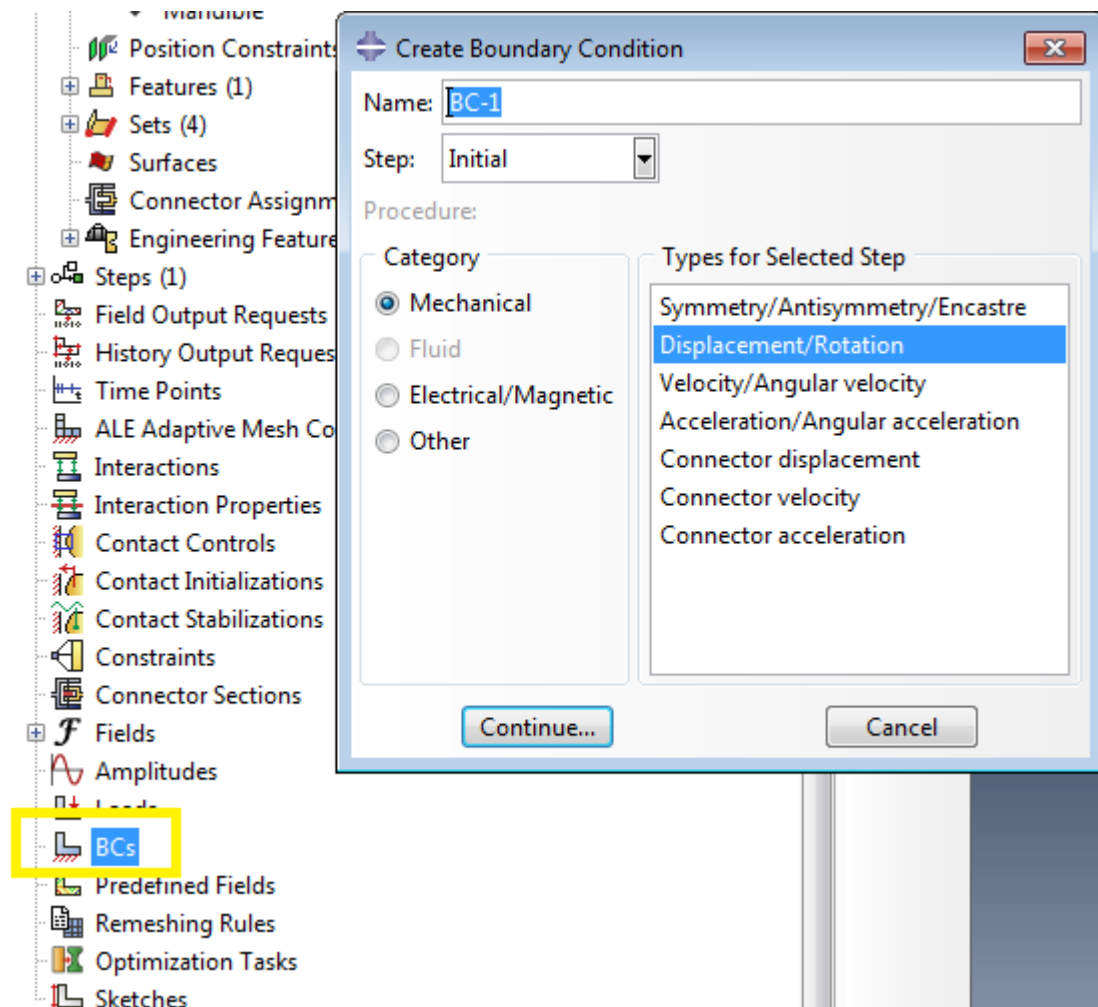
You will need to create an analysis step at which point the load is applied. An "Initial Step" is already created when you import the model. You can apply constraints to this step and they will carry through to the analysis step.

Double click the Step option in the model tree, name the Step and select the type of Step is it (e.g. Static, General). Another dialog box will appear where you can write a description and change some settings. However, the default settings are usually used.



## 7. Apply Boundary Conditions (BC):

This is where you apply your constraints to prevent rigid body motion. In the dialog box that appears, name your BC and select the Step (e.g. Initial), Category (e.g. Mechanical) and Type (e.g. Displacement/Rotation).



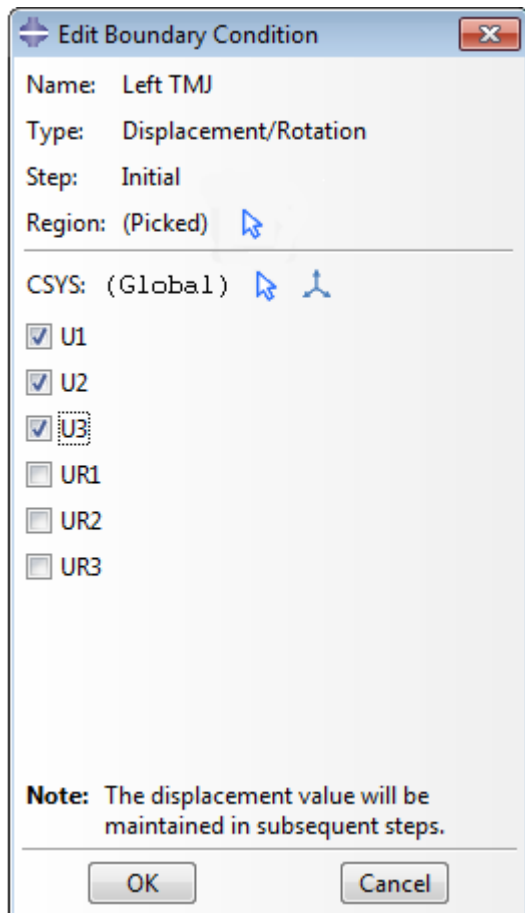
Uncheck the Create Set option in the task bar...






Select the nodes for your constraint and then select the axis you want to fix (no displacement) in the dialog box that appears. To select multiple nodes, hold the Shift key. To deselect nodes, hold the Ctrl key.

For a completely fixed constraint, select the x-, y- and z-axis (U1, U2, U3). For a semi-fixed constraint, only select the axis you want to restrict movement in (see figure below).



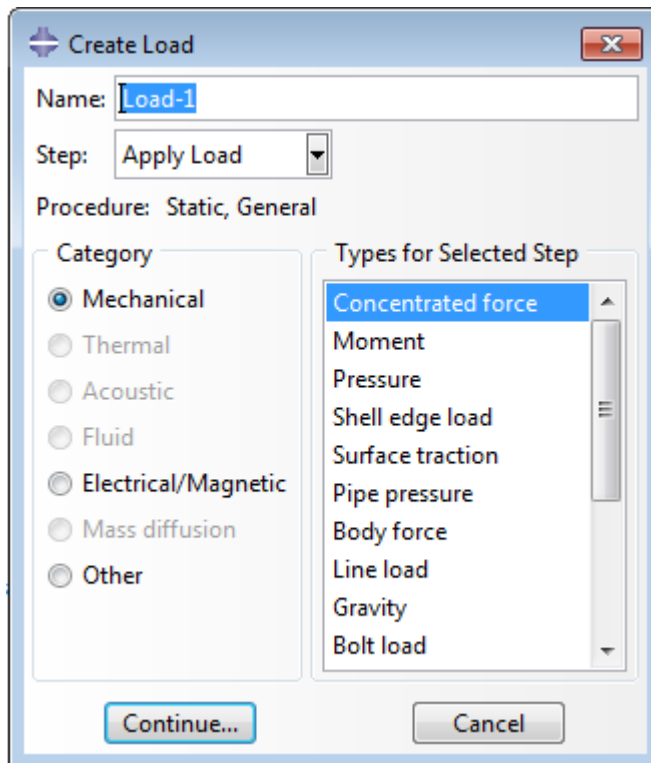


Create all you BC, naming each separately.

If you want to edit a BC, double click the BC to edit and the dialog box will reappear so you can change the region picked , change the coordinate system (CSYS)  or set up a new coordinate system  (See step 11 below), or change the axis that are fixed.

## 8. Apply Loads to nodes:

To create a load, double click the Load option in the model tree. In the dialog box that appears you can name the load and select the Step, Category (e.g. Mechanical) and Type (e.g. Concentrated force).



Then select the nodes on the model that you want the load to be applied to. If you want to add the load to a set of nodes, it is easier to first create a set (see next step).

Apply the load in the CF1 section (x direction). Note that the force will be applied per node, so you need to know how many nodes are in your set if you are using a set!

When the load is applied, yellow arrows will appear on your model. Double check the direction of the arrows to make sure they are in the correct direction.

When you have set up all the loads, make sure only one arrow originates from each node. If a Set of nodes overlaps, loads will be applied in two or more separate coordinate systems from any overlapping nodes. This will appear as an error if you try to run the simulation:


*"Following attributes are applied to overlapping, intersecting, or adjoining regions in different coordinate systems."*

Go back into the Sets and edit them to remove overlapping nodes.

## 9. Creating a Set:

You sometimes need to create a Set of nodes so that you can apply a load over the entire set.


To do so, expand the Assembly option in the model tree and double click Sets.

When selecting the nodes for a set, you only want to select the external nodes, so make sure you select this option in the toolbar . To select an area, LMB + drag the mouse over



the area. You can also change the drag shape. This will still select all the nodes on the other side of the surface, so rotate the model and deselect all the unwanted nodes. This can sometimes be very tricky, especially if the area is very curved or thin, so be patient. Alternatively, if you don't have too many nodes to select, do it one by one.




## 10. Applying loads to a set:



Double click the Load option in the model tree. In the dialog box that appears you can name the load and select the Step, Category (e.g. Mechanical) and Type (e.g. Concentrated force). Then select the Set for the load, select  on the right hand side of the task bar, and select the set in the list the appears.

Apply the load in the CF1 section (x direction). Note that the force will be applied per node, so you need to know how many nodes are in your set if you are using a set!

Also note that when you set up a load this way, the force will be parallel, directed along the x-axis of the CSYS from each node. If you want to set up a load from a Set with each node having a different direction, like the fan shape of the temporalis muscle, you can set up a coupling constraint (see step 12).

## 11. Setting up a new/local CSYS:

You can create a new CSYS by clicking  or select an existing CSYS with . Or, in the Assembly Module drop-down list, you can select the icon in the bottom right hand corner .



- To create a new CSYS click  or  and select *Rectangular*. Select a point to be the origin (the centre of the set, or origin of the muscle, is the best option) and then select a point to be the direction of the x-axis. The direction of the x-axis should be the direction of the load (e.g. from muscles origin to insertion). Click *Create Datum* and you're done. The *Create Datum* CSYS dialog box will appear again if you want to create more CSYS, or click cancel if you don't. Coordinate systems will appear under *Features* under the *Assembly* in the Model Tree.
- If you need to create a datum point and reference point (RP) offset from your model that represents the insertion of a muscle on the jaw (if you don't have a jaw model to load), you will need to follow the instructions below to create a datum and RP before you create your new CSYS.

## 12. Setting up a coupling constraint:

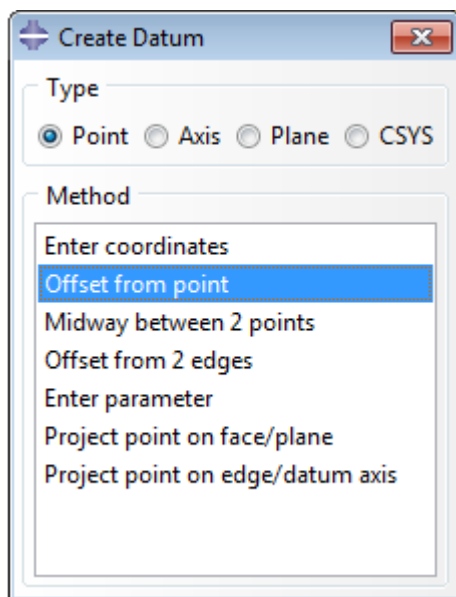
A coupling constraint is useful if you want to set up a load from an area (e.g. muscle origin) and direct it to a single point so that the force is applied to all the nodes in a different direction, like a fan (this is good for the temporalis muscle!)

Make a Set for the muscle origin as above, but instead of applying the load to the Set, you need to make a Datum point and connect a coupling constraint from the datum to the set. This makes the nodes in the Set constrained to the rigid body motion of a single node. Make sense? Here's how to do it...

### 13. Setting up a “floating” Datum and reference point:


In the top menu bar go to Tools > Datum and the *Create Datum* dialog box will appear. Or, you can click one of these icons in the Assembly Module drop-down list:  to offset from a single point, or  to select a midpoint between 2 points (hold the icon to see the different menu options).

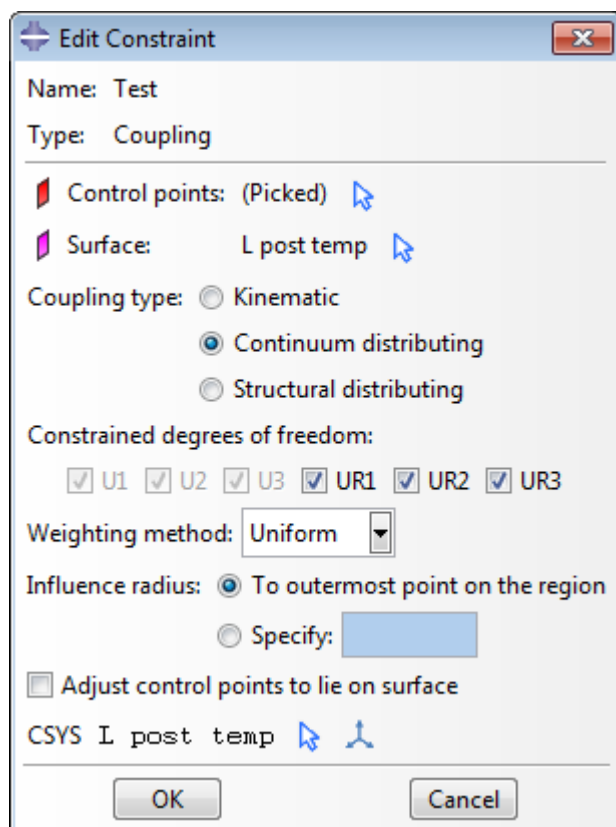
You have different options for how to place your datum: (1) you can Enter the exact coordinates, (2) you can offset the datum from a point on your model by selecting the point and typing in an offset value in the x- y- or z-axis (this is usually the best option), or (3) you can select two points and have the datum midway between them. These will be the most common selection you will need, but there are others.



Now you can make the datum a *Reference Point* by selecting the  icon under the *Interaction Module*.

### 14. Setting up a coupling constraint, continued:

Add the coupling constraint by double clicking the  **Constraints** icon in the model tree. Name your constraint and select "Coupling" in the dialog box that appears. You will then be asked to select the *Control Point*, so select *Geometry* and then select the RP. Click Done. Now select the *node region* you want to constrain (e.g. your Set) and the next dialog box will appear.



Make the coupling type distributed, not kinematic (read about the different in the Abaqus Documentation Manual), constrain all the degrees of freedom and specify the CSYS you want the constraint to be in (select the CSYS set up for that set with the x-axis in the direction of the force).

Now you can set up a Load and apply the force to the RP. This will distribute the Load through the coupling constraint to the nodes at the muscle origin.

## 15. Setting up a job:

Now that all the materials, BCs and Loads have been assigned, you can set up each Job.

Double click the *Jobs* tab under *Analysis* in the model tree. Name your job, select your model and click Continue...

In the next dialog box you can write a description of the job and change some general setting. Leave all the options as the default, but under the *Memory* tab you may want to change the percentage of physical memory the process will take.

Now change any of the BC or loads for your particular job. You can *suppress* or *resume* BCs to change the bite point (right click the BC for the menu), and you can change the loads in the same way. When you have all the loads and BC set up for your job, right click on the Job and select *Write Input*. This will write an input file to your working directory that you can run later in a batch file. Alternatively, if you just want to run one job, you can click *Submit*. You can also *Data Check* first for any errors.

Create more Jobs, change the loads and BCs and write an input file for each different simulation.

## 16. Running your Jobs from a batch file:

If you have a series of Jobs it is faster and easier to run as a batch file from the command line. You do not need the Abaqus user interface open for this.

To write a batch file, open Notepad (or any text editor) and type:

```
call abaqus j=Job-1 int ask_delete=off  
call abaqus j=Job-2 int ask_delete=off  
call abaqus j=Job-3 int ask_delete=off
```

Replace Job-1, Job-2 etc. with the name of the input file you saved your jobs as. Save the file as runAbaqusJobs.bat (or whatever you want, but it's important it's saved as a .bat NOT .txt) in the same directory as the input files.

Open your command line by searching cmd in the start menu.

Change the directory to the working directory you saved the input files, and the batch file to:

```
cd "C:\...\Documents\PhD\Abaqus working directory"
```


Then run the batch file you wrote earlier by typing:

```
runAbaqusJobs.bat
```





This will call each input file in succession so you can go home and relax :)


## 17. Abaqus post processing

To see the results from a job that you have submitted, or ran as a batch file, right click the job and select *Results* (make sure you have the correct working directory set). This will open the *Results* tab and display a new model tree on the LHS of the window. You will find the output file in this model tree.


You can change the appearance of the model by clicking . This will open a dialog box that gives you the options to change the *Render Style*, *Colour* and the type of *Visible Edges*.


You can also display the deformed  model or the undeformed model .

To display the stress or strain contours select the deformed shape , undeformed shape , or both . When displaying contours on both, change the appearance of the undeformed model by clicking .



Change the Contour Plot Options by clicking . In the dialog box you will find the maximum and minimum stress, principle strains etc. and the location, and you can change the appearance of the contours (under the *Basic* tab) to *Line*, *Banded* or *Quilt*, and the contour intervals from *Discrete* to *Continuous*. Play around...this is all entirely up to you.


At the top toolbar you can change the type of results that are being displayed: stress (S), strain (E), displacement (U), reaction force (RF)...these are probably the only ones you are going to need.

To get the RF from the bite points click  to probe the nodes. Select *Nodes* in the dropdown list next to *Probe* and then click on the node for the bite point.

To get the internal strain energy ( $\mathcal{E}$ ) expand the Output database file in the model tree, expand the History Output and then double click *Strain energy: ALLSE for Whole Model*. A temporary XY plot will appear. Click  to probe the plot and click the line near the top.


Export a Field Output Report to get the stress and strain values for every node (and then get mean, median, boxplots etc. in R). Go to the menu *Report > field output*, and select the components you want to export. In the *Setup* tab, name your file and uncheck *Column totals* and *Column min/max*. The field output file can then be opened in R for analysis.

Setting up a *View Cut* is easily done by clicking the *Activate/Deactivate View Cut* icon . By clicking the next icon across, , you can change the axis and location of the cut.

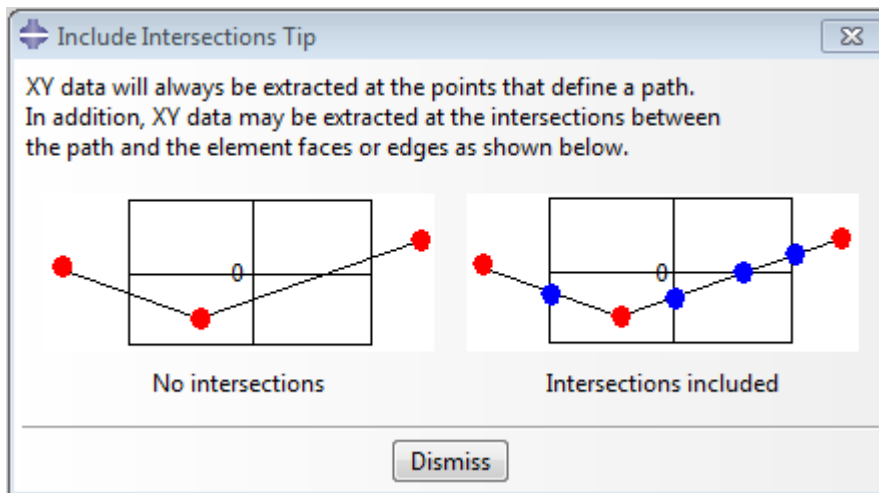
You can also probe  nodes or elements on the model for stress or strain values in critical areas of interest.

### Setting up an XY plot of a path:

Create a path: Tools > Path > Create. In the dialog box that appears, click "*Add Before...*" and select the nodes for the path, click done.

Click the *Create XY Data* icon  and select path in the options, click continue. In the dialog box that appears select the path to be plotted from the dropdown list and change any of the settings.

Under Point Locations, you can choose to include intersection along the path, or not.



You can also choose the X and Y values. X values can be displayed as the True Distance, Normalised Distance, Sequence ID or X, Y, Z Distances.

Y Values can be changed by selecting the *Field output variable* by clicking [Field Output...](#).

When you're done, you can see your results by clicking *Plot*, then *Save As...*

