

# SIMPLE ABAQUS STUFF

by André Duarte B. L. Ferreira

Last Edition: 16/06/2016

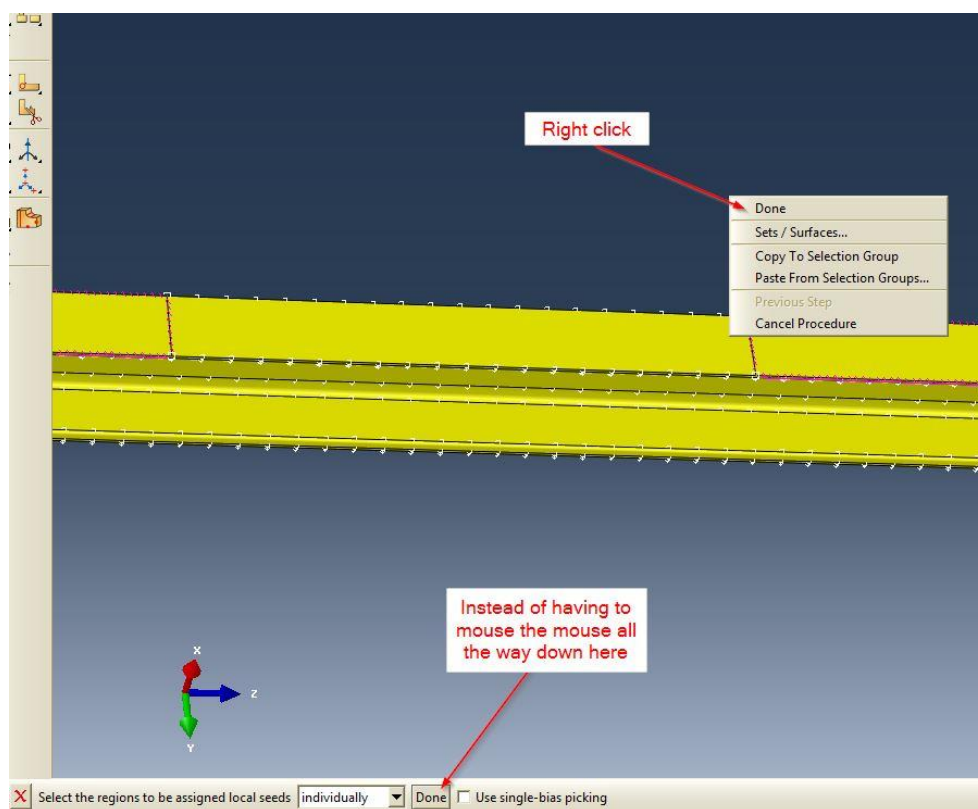
## ABAQUS Online Manual

<https://www.sharcnet.ca/Software/Abaqus/6.11.2/books/usi/default.htm>

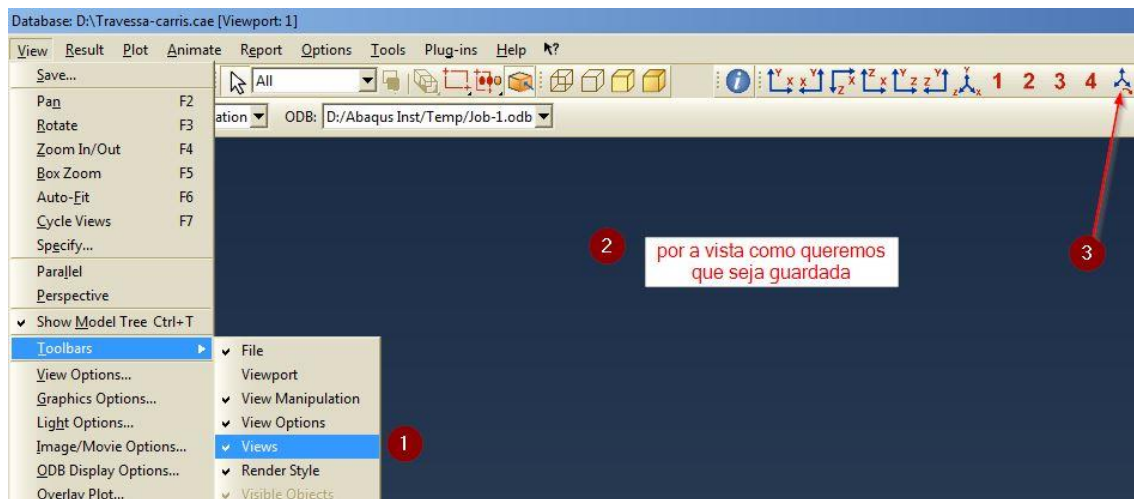
### TIPS

#### Faster clicking

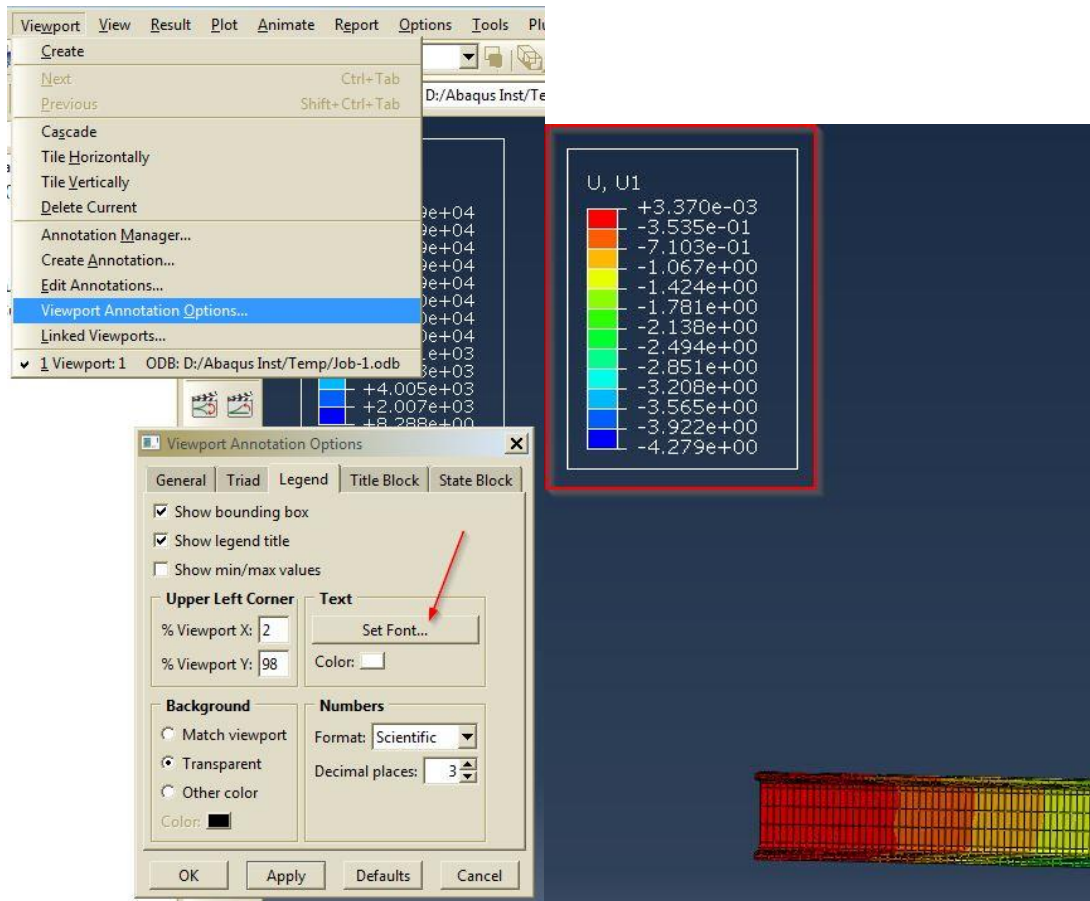
Instead of moving the mouse to click "done" below, right click and then click



#### Saving a custom view

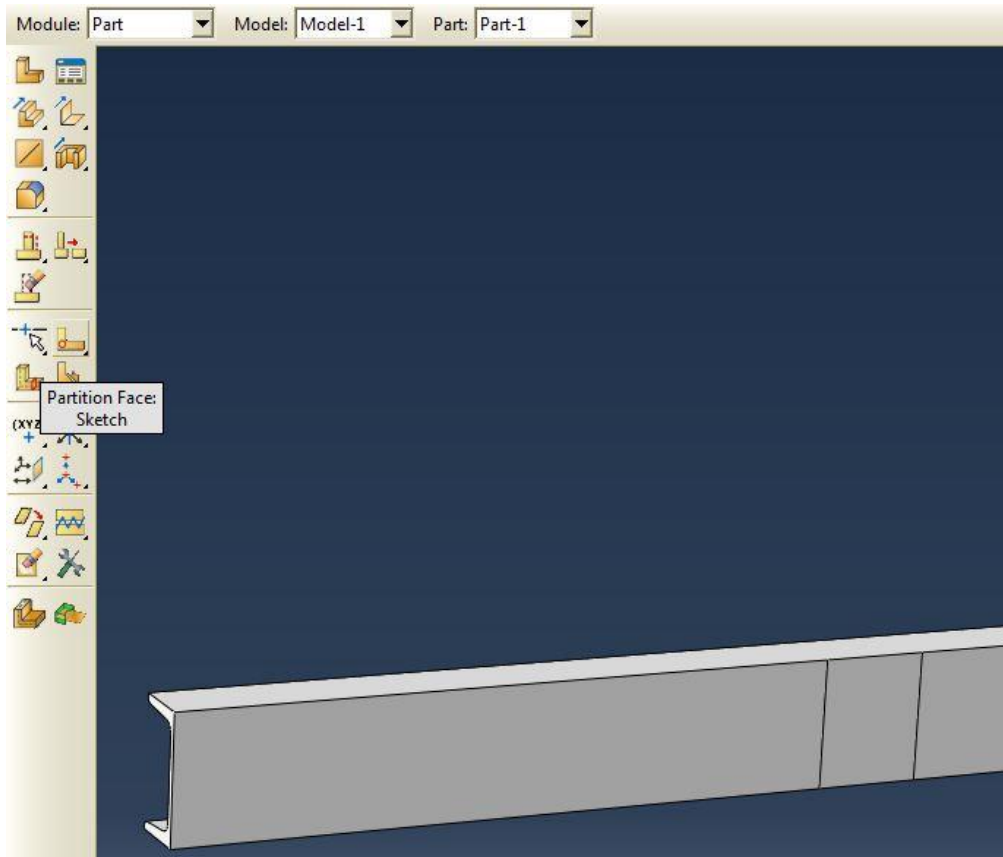


## INCREASE LEGEND SIZE

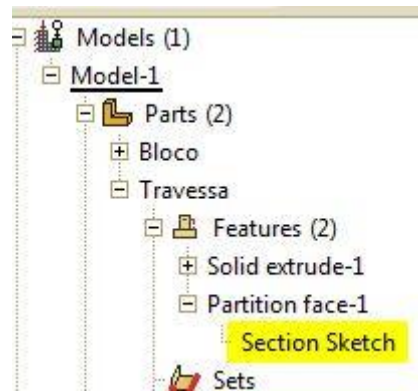


## CREATE NEW PARTITION AND EDIT EXISTING PARTITION

Create



## Edit



## DISCRETE RIGID VS ANALYTICAL RIGID

An analytical rigid does not need to be meshed, and instead can be described by an analytical function. A primitive such as a sphere or a plane would be an example of something that can be an analytical rigid.

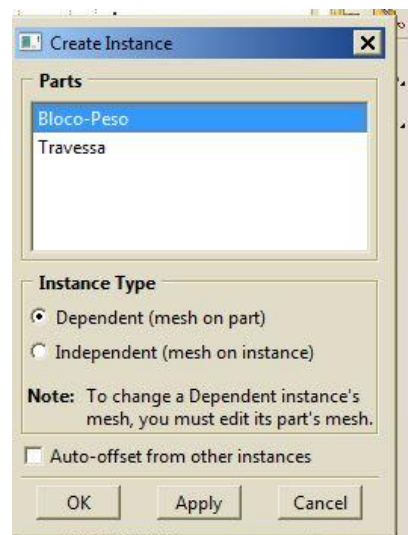
Analytical rigid surfaces represent simple rigid bodies (e.g., extrusions and bodies of revolution). They exactly represent the geometry of the rigid body, but are limited to simpler geometries.

Discrete rigid elements are used to model rigid bodies that have complex geometries (i.e., those that require a finite element mesh to represent the geometry).

## INSTANCE TYPE

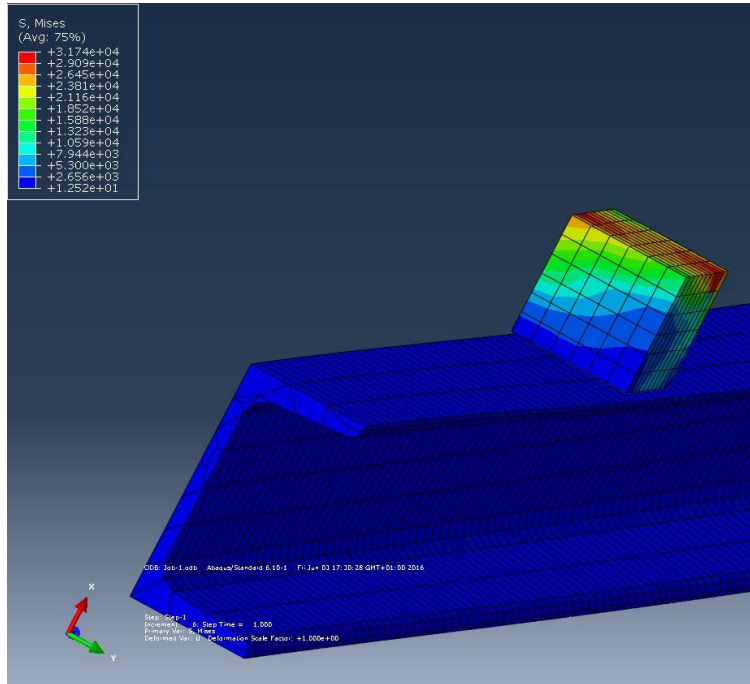
Abaqus uses parts to represent the geometry of some object, and then use instances for the calculations.

Most of the time I use the mesh-on-part option, the mesh-on-instance option can be useful if you want to use different meshes on different instances of the same part.

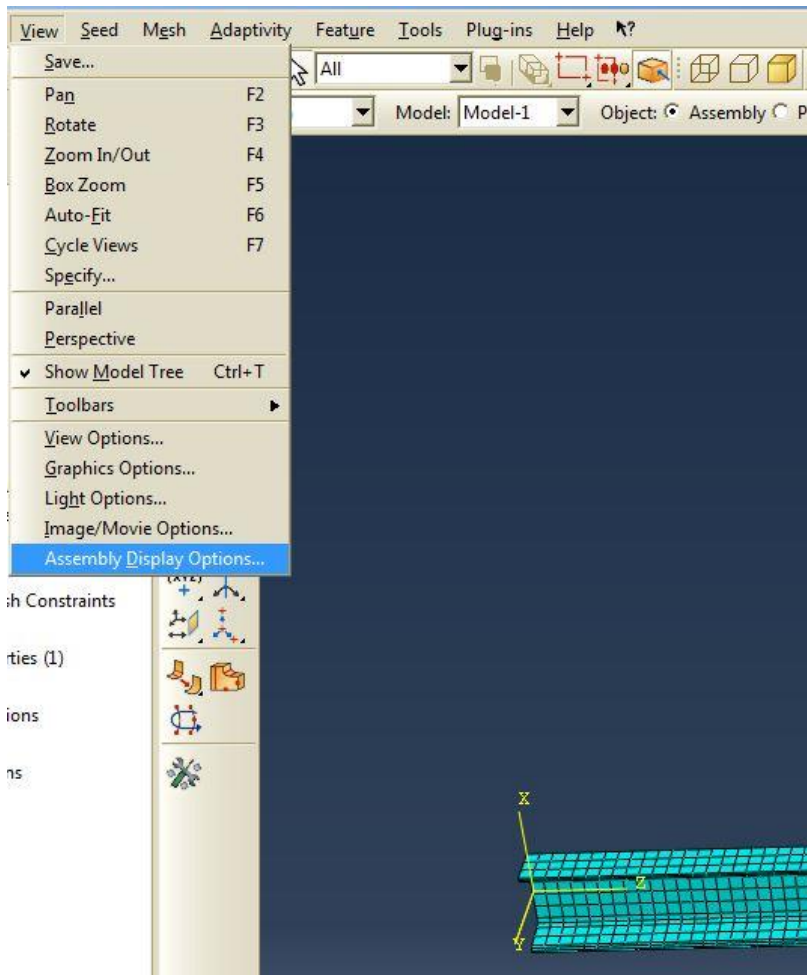


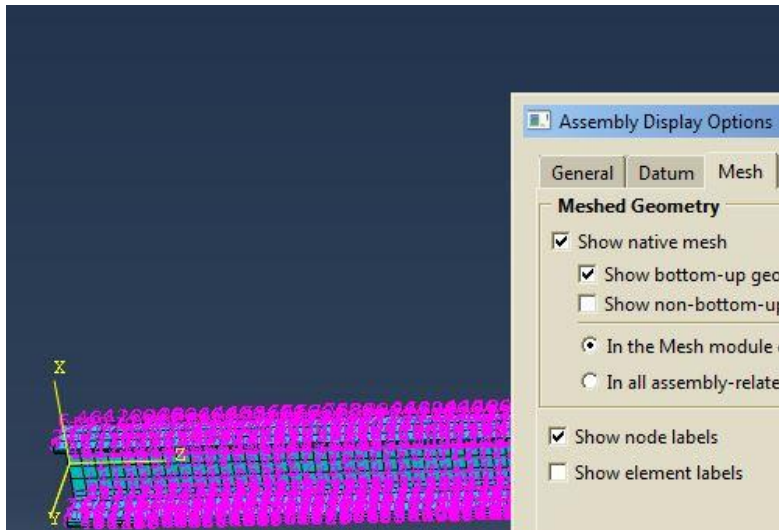
## FINDING DATA FOR A SPECIFIC NODE

In this specific case I want to know the stress in specific nodes of the "travessa" part and not the square block.

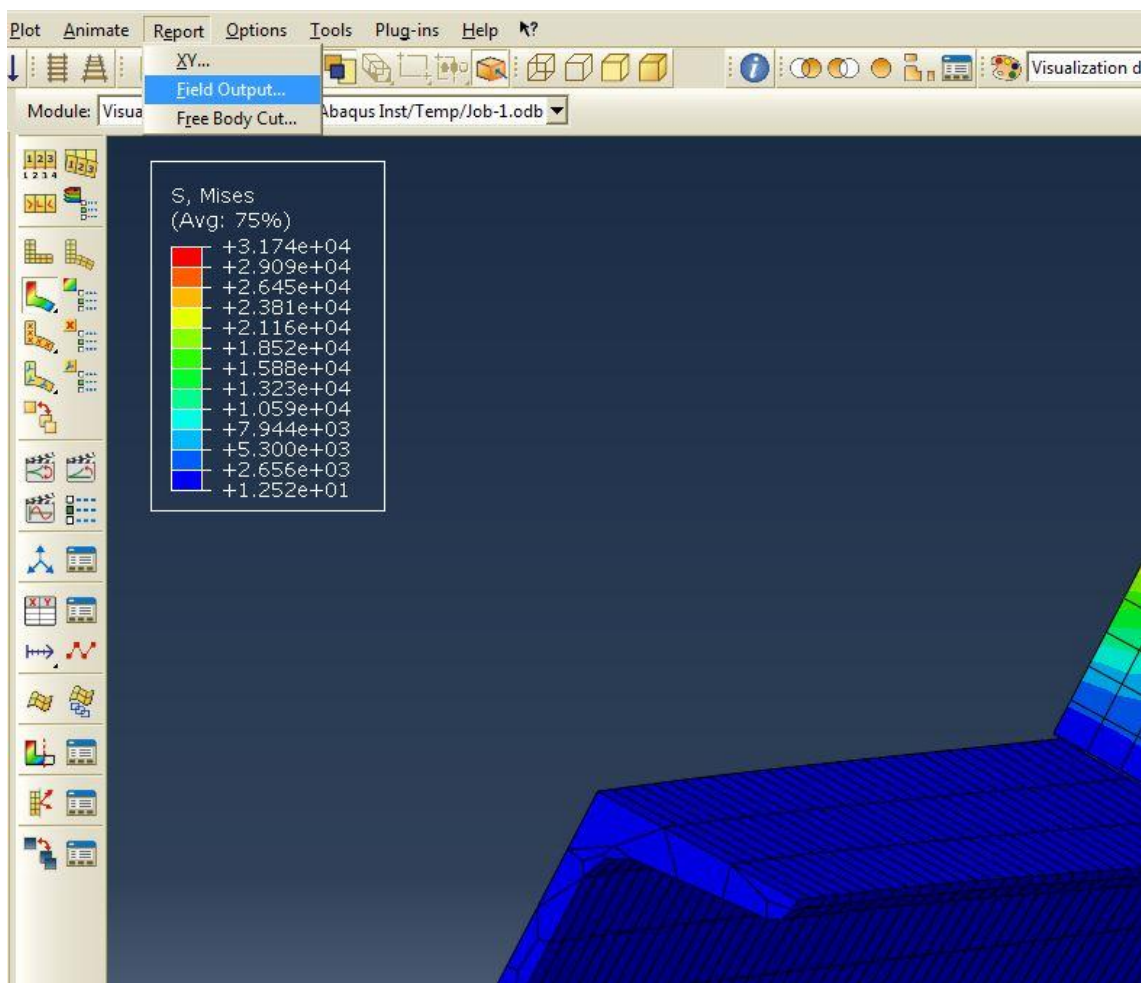


For that I need to make ABAQUS show me the node numbers.



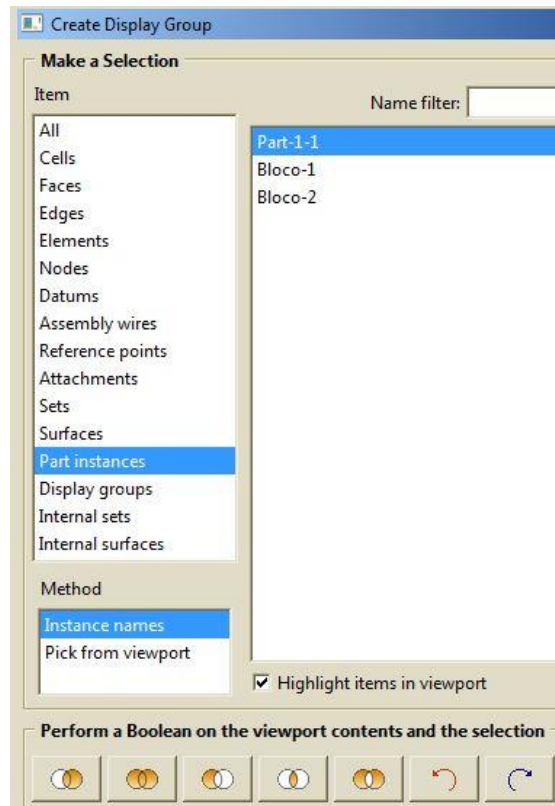
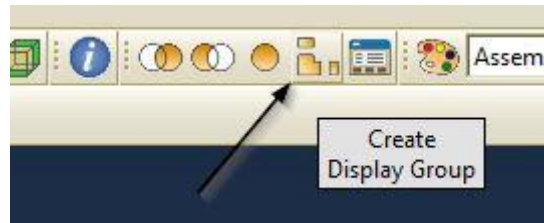


Having the numbers now we can identify the nodes in the report that we're getting next.

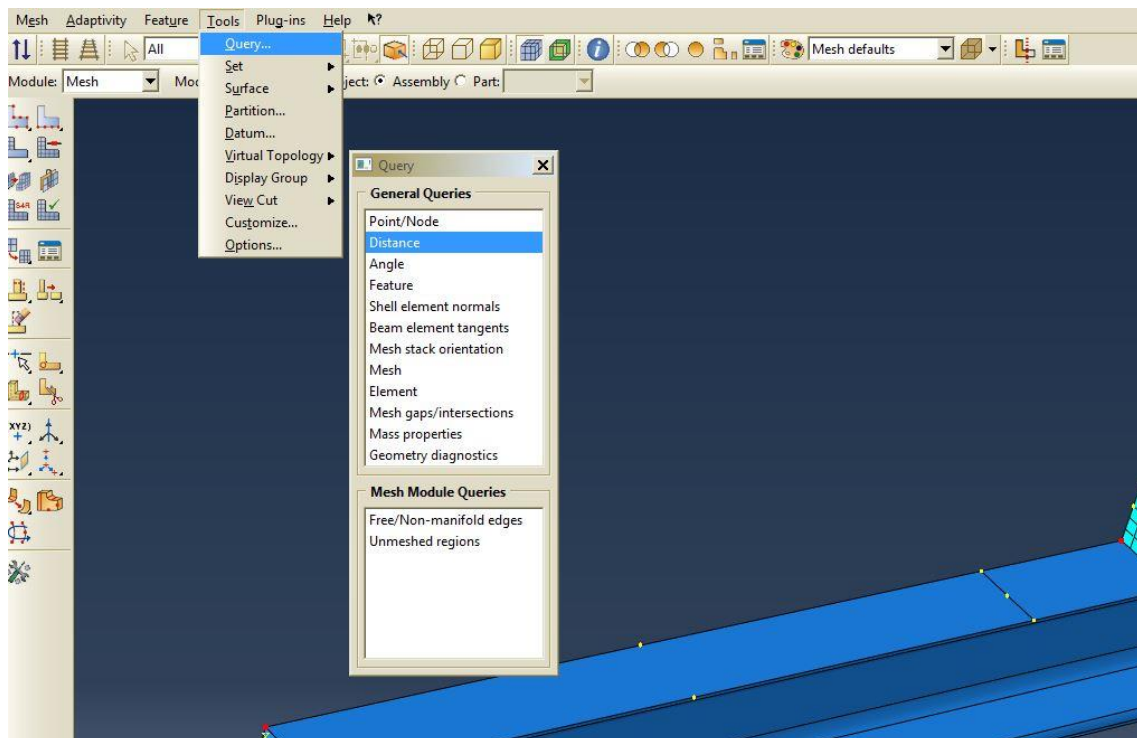


After that chose the data that we want and ABAQUS will send it to the ABAQUS temp folder. Open it with a simple text editor.

## HIDE PARTS



## MEASURE DISTANCE



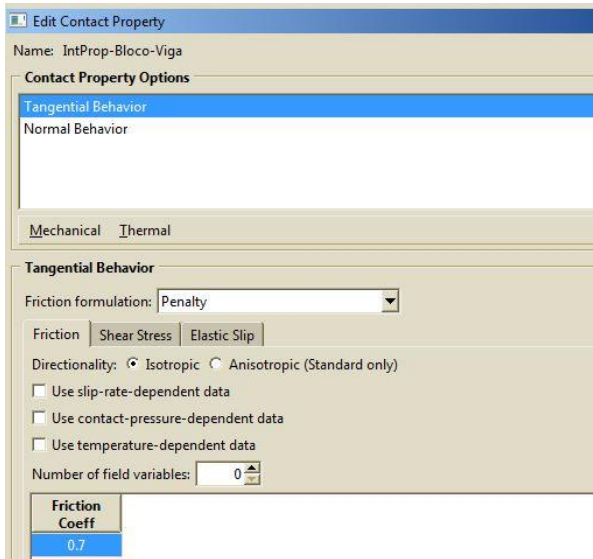
After that click in "distance". Select the 1<sup>st</sup> point, then the 2<sup>nd</sup> point to measure the distance. The value will appear below.

## LOCAL REFINEMENT OF MESH

Create partition, then seed edge.

## TANGENTIAL BEHAVIOR, FRICTION FORMULATION

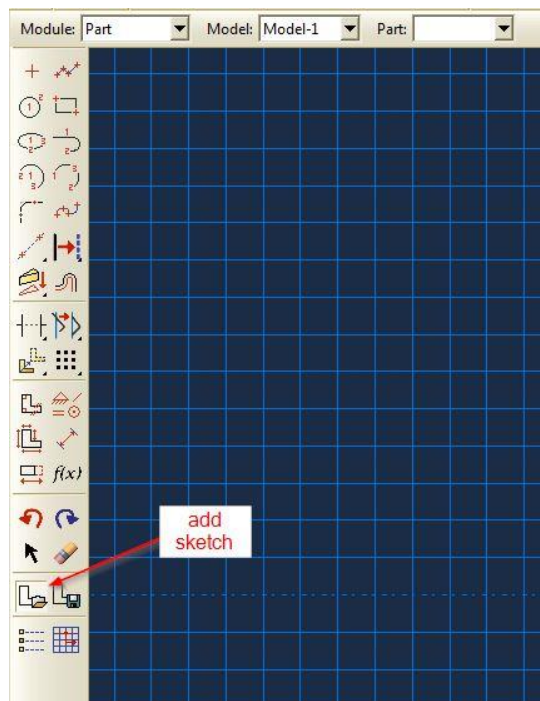
### Penalty



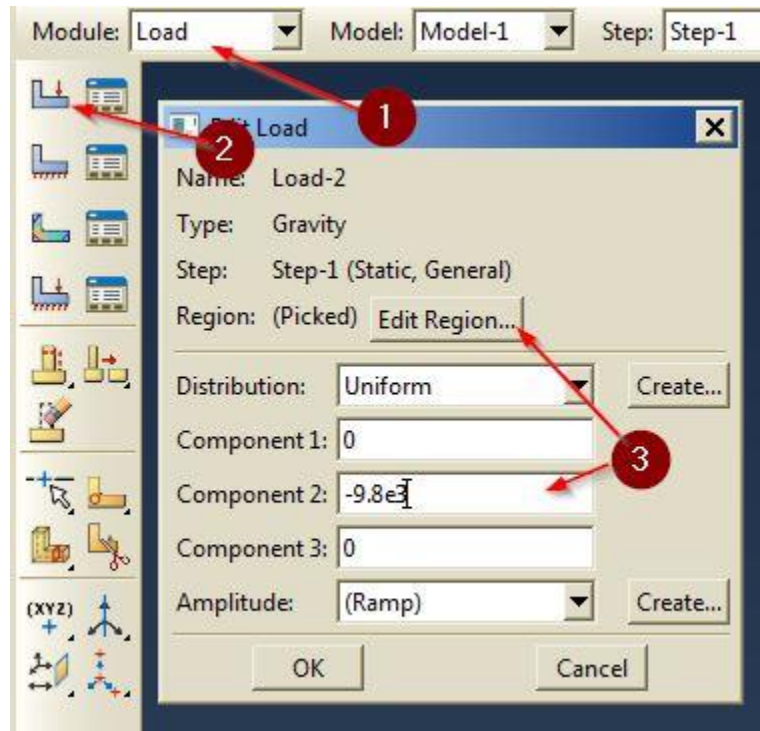
Materials	$\mu_s$ static	$\mu_k$ kinematic
Steel on steel	0.74	0.57
Aluminum on steel	0.61	0.47
Copper on steel	0.53	0.36
Rubber on concrete (dry)	1.0	0.8
Rubber on concrete (wet)	0.3	0.25
Wood on wood	0.25-0.5	0.2
Glass on glass	0.94	0.4
Teflon on Teflon	0.04	0.04
Teflon on steel	0.04	0.04
Waxed wood on wet snow	0.14	0.1
Waxed wood on dry snow	0.10	0.04
Metal on metal (lubricated)	0.15	0.06
Ice on ice	0.1	0.03
Synovial joints in humans	0.01	0.003
Very rough surfaces		1.5

## IMPORT SKETCH FROM SOLIDWORKS

1. Put sketch on frontal view;
2. Save as .dxf
3. Go to ABAQUS and do Import, and import the file;
4. Create part and then



## ADDING GRAVITY TO A PART



## STEP MODULE

A step is like a phase. The initial step is where you establish the starting conditions. Then on the next step you can e.g. apply a force. After that, on the next step, step 2, you can stop that force and apply another force.

## BOUNDARY CONDITIONS OF FACES/PLANES

