

TMM4160 Fracture Mechanics

Manual for Computational Exercises

Version 3.0

Zhiliang Zhang

Dept. of Structural Engineering



Contents

1. Problem Definition.....	4
1.1 Specimen geometry	4
1.2 Materials	4
1.3 Loading and boundary conditions	6
2. Preliminary about ABAQUS/CAE.....	6
2.1 Units in ABAQUS	6
2.2 File types in ABAQUS	7
2.3 Starting ABAQUS/CAE	7
2.4 Different Modules in ABAQUS/CAE	8
3. Creating a Finite Element Model with ABAQUS/CAE	8
3.1 Create a Part for the half model	8
3.2 Assign the Material Properties	12
3.3 Define the Assembly	13
3.4 Configure the analysis Step	14
3.5 Apply the load and boundary conditions	17
3.6 Mesh the model	19
3.7 Create an analysis job	21
3.8 Post-processing the results.....	22
3.8.1 Graphical plots	22
3.8.2 Global load-displacement curves	23
3.8.3 Nominal stress-strain curve of a tension specimen	25
3.8.4 Bending stress of a bending specimen	25
4. Elastic Fracture Parameter.....	26
4.1 Calculate the K based on stress distribution along the crack ligament	26
4.2 Define a crack to compute the J-integral	28
4.3 Create History Output for the J-integral	28
4.4 K-J relation for elastic materials.....	28
5. Elastic-plastic Fracture Parameters	29
5.1 Create elastic-plastic materials in ABAQUS.....	29
5.2 J-integral output.....	29
5.3 Define the node for outputting CTOD/ CMOD.....	29
5.4 J-CTOD relations.....	30
6. Tension versus bending	30
7. Mesh size effect	30
8. Computational Fracture Mechanics Report	31
8.1 Steps and time schedule (suggestion)	31
8.2 Graded report	32
9. Computational Fracture Mechanics Networks.....	32

Preface

In TMM4160 fracture mechanics course, you will not only master the fracture mechanics theory. You will also learn to use the finite element program ABAQUS to calculate the fracture mechanics parameters (J, K and CTOD). This documentation is intended to cover the minimum you need to finish the computational exercises. For deepening in finite element method please refer to finite element method courses at Dept. of Structural Engineering. Modern finite element programs have advanced to such a degree that it is very easy to obtain results and color plots. It is therefore very important to take your time to critically judge/understand your results and link the results to fracture mechanics theory.

Zhiliang Zhang
Professor of Mechanics and Materials
Dept. of Structural Engineering
Email: zhiliang.zhang@ntnu.no
<https://www.ntnu.no/ansatte/zhiliang.zhang>
www.ntnu.edu/nml

Problem Definition

Fracture mechanics concerns the interaction of the applied **crack driving force** and materials **fracture resistance**. For simple geometry and linear elastic materials fracture parameters can be easily calculated based on existing analytical equations. For complicated geometry and elastic-plastic materials finite element method is necessary. The main purpose of these computational exercises is to use ABAQUS to calculate the fracture parameters of a 2D plane strain single edge notched specimen.

1.1 Specimen geometry

The dimension of specimen is shown in *Figure 1*. The crack size (a) is 20 mm, specimen depth (w) is 50 mm and length (L) is 300 mm.

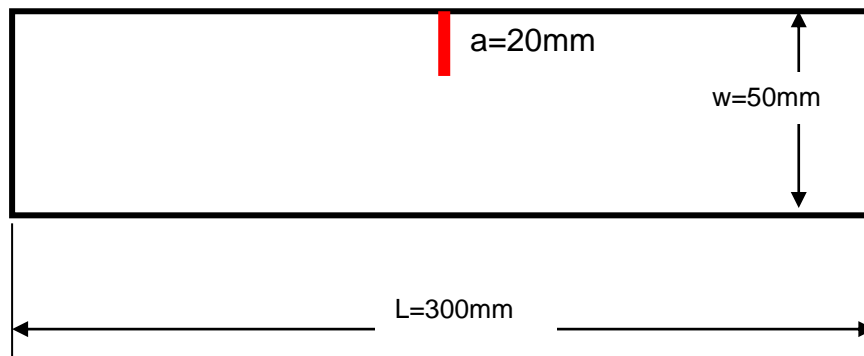


Figure 1. Dimensions of the specimen considered in this course

1.2 Materials

Four materials will be considered in this exercise: one linear elastic and three elastic-plastic.

A **linear elastic material (E1)** will be considered first. Linear elastic material is relevant for the computation of stress intensity factor K . Typical stress-strain curve for a linear elastic material is shown in *Figure 2*. There are two materials constants associated with a linear elastic material: Young's modulus E and Poisson ratio ν .

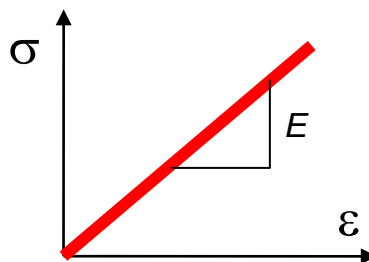


Figure 2. Linear elastic material stress-strain relation.

The next step in your exercise is to consider more realistic engineering materials, such as steel alloys, which are **elastic-plastic materials (EP0, EP1, EP2)**. The behavior of such material consists of elastic, yielding and plastic behavior. Typical elastic-plastic stress-strain relation is shown in *Figure 3a*). It should be noted that in ABAQUS the elastic-plastic behaviors are defined separately. The stress-equivalent plastic strain curve (*Figure 3b*) is used for defining

the plastic behavior in ABAQUS (not the stress-strain curve of **Figure 3a**). Materials stress-strain curve (**Figure 3a**) is usually obtained from uniaxial tensile tests.

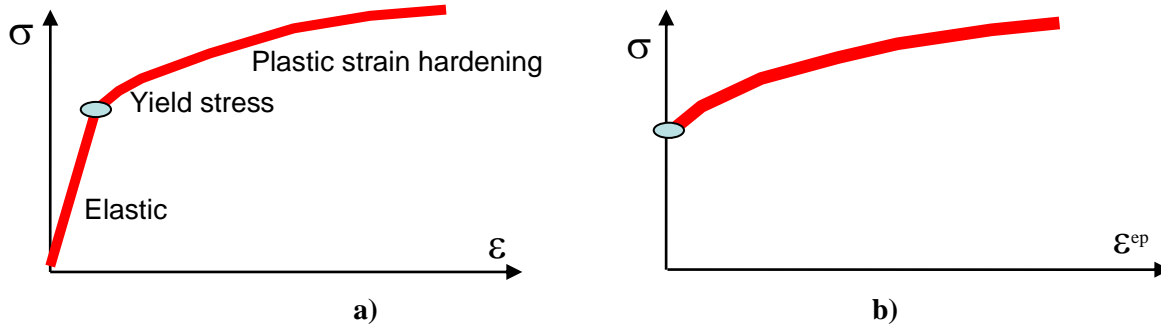


Figure 3. a) Stress-strain curve for an elastic-plastic material and b) stress-equivalent plastic strain curve needed for ABAQUS input.

Materials	E1	EP0 ($n=0$)		EP1 ($n=0.10$)		EP2 ($n=0.20$)	
Young's Modulus [MPa]	200000	200 000		200 000		200 000	
Poisson Ratio	0.3	0.3		0.3		0.3	
Plastic behaviour		ϵ^{ep}	σ_f [MPa]	ϵ^{ep}	σ_f [MPa]	ϵ^{ep}	σ_f [MPa]
		0.0	400	0.0	400	0.0	400
		0.8	400	0.02	508	0.02	646
				0.04	542	0.04	735
				0.06	564	0.06	795
				0.08	580	0.08	841
				0.10	593	0.10	878
				0.15	617	0.15	951
				0.20	635	0.20	1007
				0.30	661	0.30	1091
				0.40	680	0.40	1155
				0.50	695	0.50	1208
				0.60	708	0.60	1252
				0.80	728	0.80	1326

Table 1. Materials used in the exercise

In this exercise “model-materials” will be considered. Following elastic-plastic material model with **three** different plastic strain hardening exponents will be used.

$$\sigma_f = \sigma_0 \left(1 + \frac{\bar{\epsilon}^p}{\epsilon_0} \right)^n \quad (1)$$

where n is the strain hardening exponent, σ_0 , $\epsilon_0 = \sigma_0 / E$ are the yield stress and yield strain, σ_f and $\bar{\epsilon}^p$ are the flow stress and equivalent plastic strain, respectively. The yield stress for all the three elastic-plastic materials is the same $\sigma_0 = 400$ MPa. In order to facilitate your inputs of material behavior in ABAQUS, equation (1) has been tabulated and the materials have been categorized in the following table. The material EP0 where $n=0$ corresponds to a perfectly plastic material i.e. no strain hardening occur.

1.3 Loading and boundary conditions

Same specimen shown in *Figure 1* will be used for the analyses of tension (*Figure 4a*) and bending (*Figure 4b*) problems.

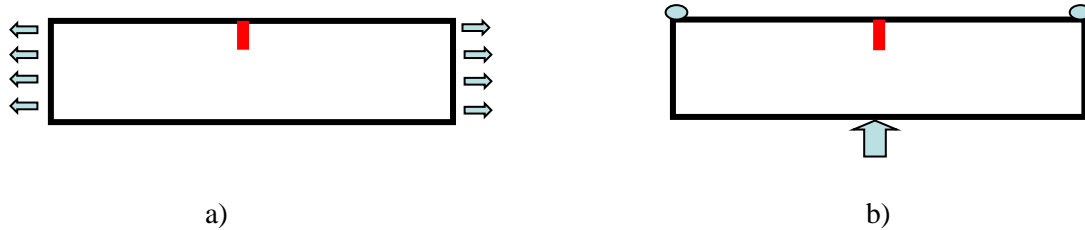


Figure 4. a) single edge notched tension and b) single edge notched bending

Both the boundary conditions and loadings are symmetrical. By using the symmetry half of the specimen should be used. *Figure 5* suggests the half models to be used.

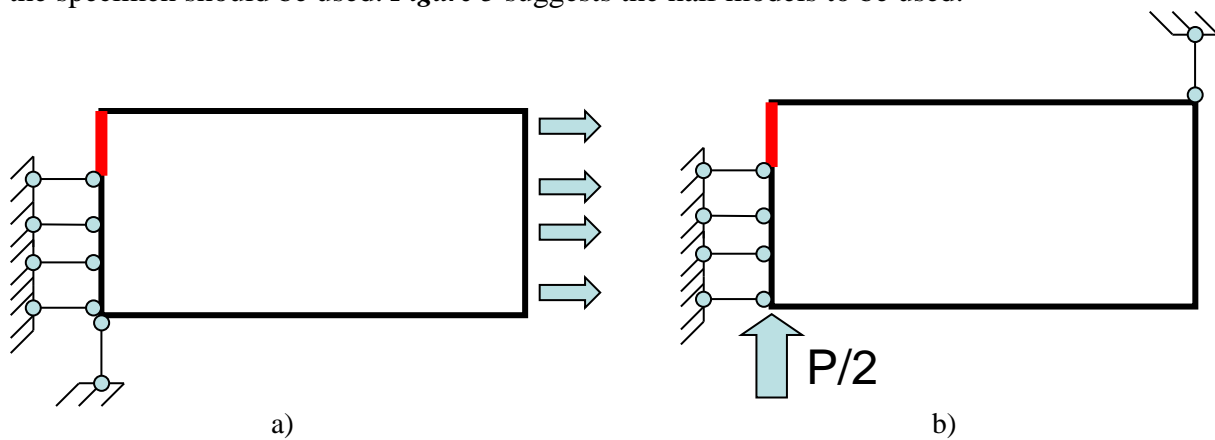


Figure 5. a) half tension model and b) half bending model

2. Preliminary about ABAQUS/CAE

2.1 Units in ABAQUS

Before starting to define any model, you need to decide which system of units you will use. ABAQUS has no built-in system of units. All input data must be specified in consistent units.

Quantity	SI	SI(mm)
Length	m	mm
Force	N	N
Time	s	s
Stress	Pa(N/m ²)	MPa (N/mm ²)
Young's modulus	Pa(N/m ²)	MPa (N/mm ²)

Table 2. Common consistent units

In this exercise it is suggested that SI (mm) units will be used. That means that the units for length, force, and Young's modulus will be mm, N and MPa, respectively. Young's modulus usually dictates the units of other quantities.

2.2 File types in ABAQUS

ABAQUS/CAE is a complete environment that provides an interface for creating, submitting, monitoring, and evaluating results from ABAQUS solvers named ABAQUS/ Implicit and ABAQUS/Explicit. ABAQUS/Explicit is mainly for dynamic and transient problems. In this course we will limit our self to ABAQUS/Implicit.

There are two major file types in ABAQUS/CAE:

- *.cae – file, which contains all the modeling (pre-processing) data and information.
- *.odb – file, which contains the analysis results.

When you run analyses with ABAQUS, all the files (including the **odb** – file) will be stored on the TEMP-directory of the computer you use. You can modify this location in “**File → Set Work directory**” and set the file where you store the **cae** – file for instance. Also please **name** the analyses systematically in a way you can easily identify analysis afterwards. DO NOT put more than one analysis in a **cae** – file. If you are to change only the material, you should make a copy of the **cae** – file and rename/edit the copy. If you have more than one analysis in the same file, you can be sure you will mix up the different analyses quite soon.

It is suggested that you make a directory where you keep all the files for this exercise. By the end of the course you will have quite a lot of files, so it is important to be able to distinguish them from the very beginning.

ABAQUS is a comprehensive program and is well documented. A complete set of online ABAQUS manuals can be found (the exact path could be dependent on the installation):

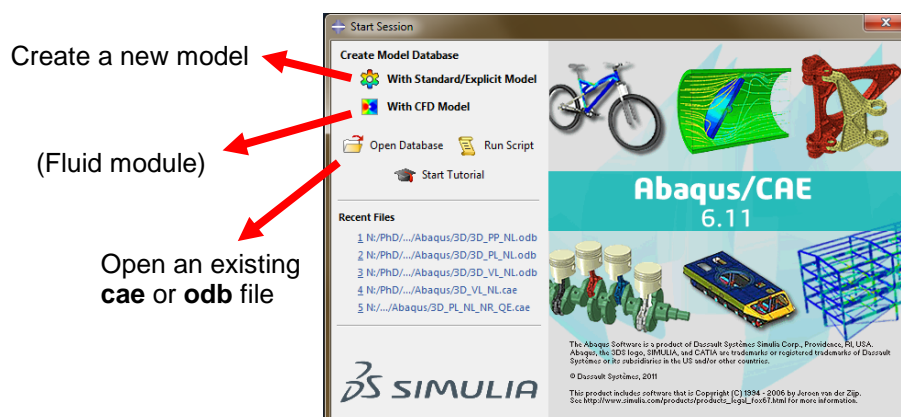
- **Start → ABAQUS 6.12 → ABAQUS Documentation**

2.3 Starting ABAQUS/CAE

Always bring the “Fracture Mechanics” textbook with you. You will regularly need to recover some theory and results should be compared with some of the plots or equations in the book.

- **Start → ABAQUS 6.12 → ABAQUS CAE → Create Model Database**

You will get the following starting panel of the ABAQUS/CAE.



2.4 Different Modules in ABAQUS/CAE

ABAQUS/CAE is divided into modules, where each module defines an aspect of the modeling process; for example, defining the geometry, defining material properties, and generating a mesh. As you move from module to module, you build the model from which ABAQUS/CAE generates an input file that you submit to the ABAQUS/Standard or ABAQUS/Explicit solver for analysis. For example, you use the Property module to define material and section properties and the Step module to choose an analysis procedure. The ABAQUS/CAE postprocessor is called the Visualization.

You can enter a module by selecting it from the Module list under the toolbar, as shown in Figure 6. Also, you will find the equivalent of all modules in the Model tree.

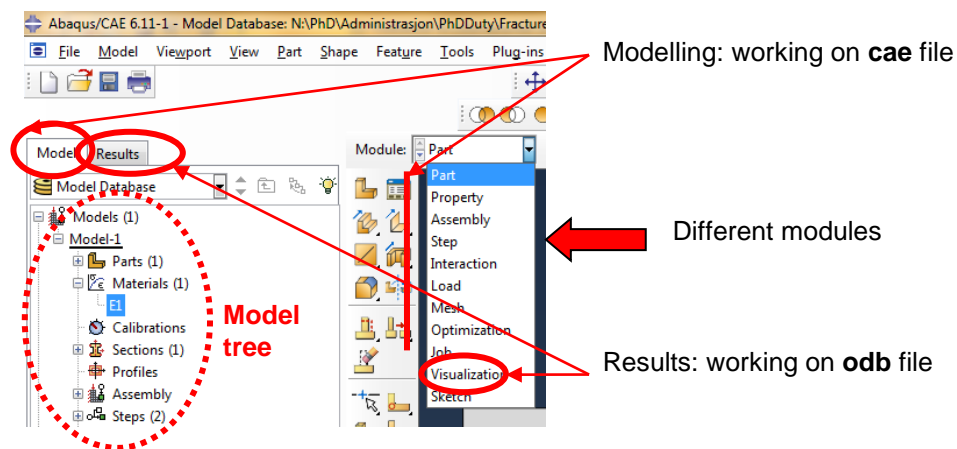


Figure 6. Different modules in ABAQUS/CAE

3. Creating a Finite Element Model with ABAQUS/CAE

3.1 Create a Part for the half model

Parts define the geometry of the individual components of your model and, therefore, are the building blocks of an ABAQUS/CAE model. You can create parts that are native to ABAQUS/CAE, or you can import parts created by other applications either as a geometric representation or as a finite element mesh. You will start the **half tension problem** by creating a **two-dimensional, deformable shell** part. You do this by sketching the geometry of the specimen. ABAQUS/CAE automatically enters the Sketcher when you create a part.

- Enter the ABAQUS/CAE “**Part**” module. The Model Tree appears in the left side of the main window. Between the Model Tree and the canvas is the Part module toolbox. A toolbox contains a set of icons that allow the users to bypass the menus in the main menu bar. For many tools, as you select an item from the main menu bar or the Model Tree, the corresponding tool is highlighted in the module toolbox so you can learn its location. In the following we will use the icons frequently. In this exercise we will use the icons as much as possible.

ABAQUS/CAE often displays a short message in the prompt area indicating what you should do next, as shown in *Figure 7*. Click the Cancel button to cancel the current task. Click the Previous button to cancel the current step in the task and return to the previous step.

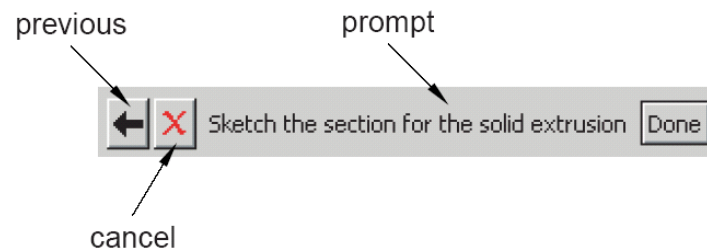
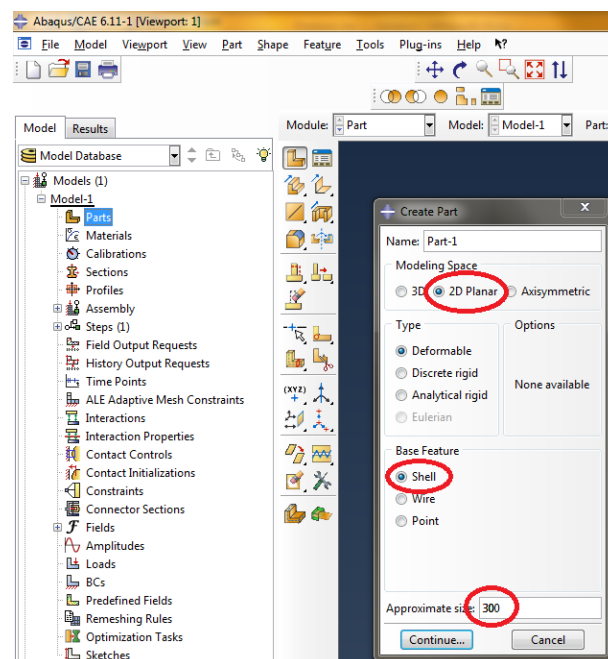








Figure 7. Messages and instructions are displayed in the prompt area.

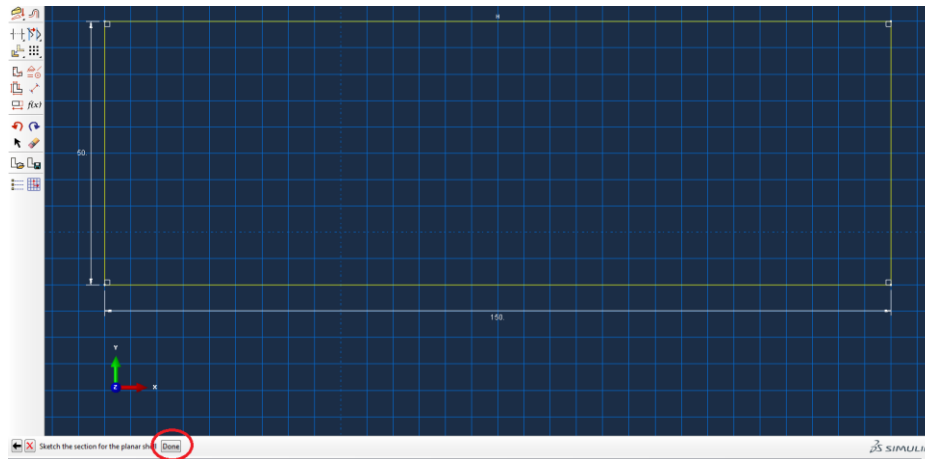
- In the Model Tree, double-click the **Parts** container to create a new part. The Create Part dialog box appears. ABAQUS/CAE also displays text in the prompt area near the bottom of the window to guide you through the procedure. You use the Create Part dialog box to name the part; to choose its modeling space, type, and base feature; and to set the approximate size. You can edit and rename a part after you create it; you can also change its modeling space and type but not its base feature.

Tips: In all tools in ABAQUS/CAE, if you simply position the cursor over a tool in the Sketcher toolbox for a short time, a small window appears that gives a brief description of the tool. When you select a tool, a white background appears on it.

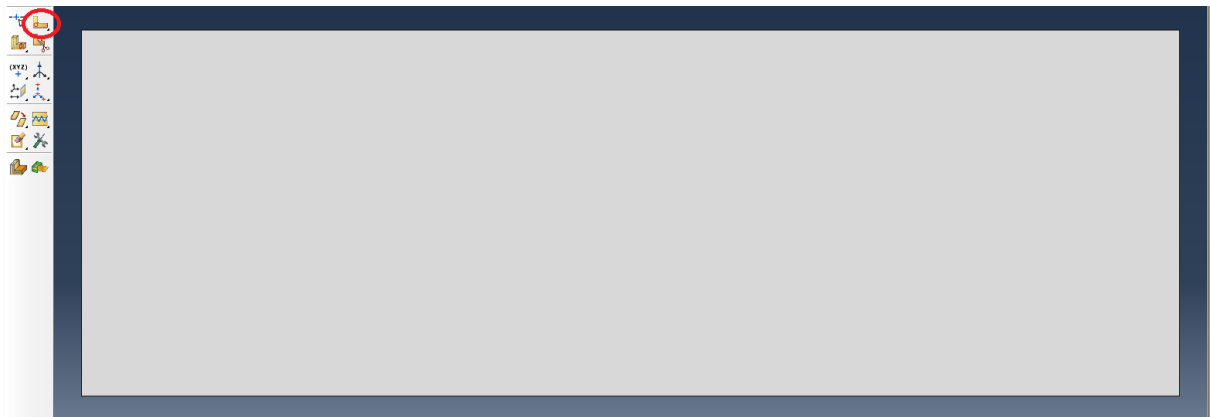



- Choose 2D planar, deformable, shell and approximate size 300. Click Continue to exit the Create Part dialog box and enter the sketcher window.
- Use  or  in the “Part” module toolbox create a rectangular part with dimension 150x50mm. Draw a rectangle and then constrain your sketch by adding appropriate dimensions using . If you want to modify a dimension, you may use

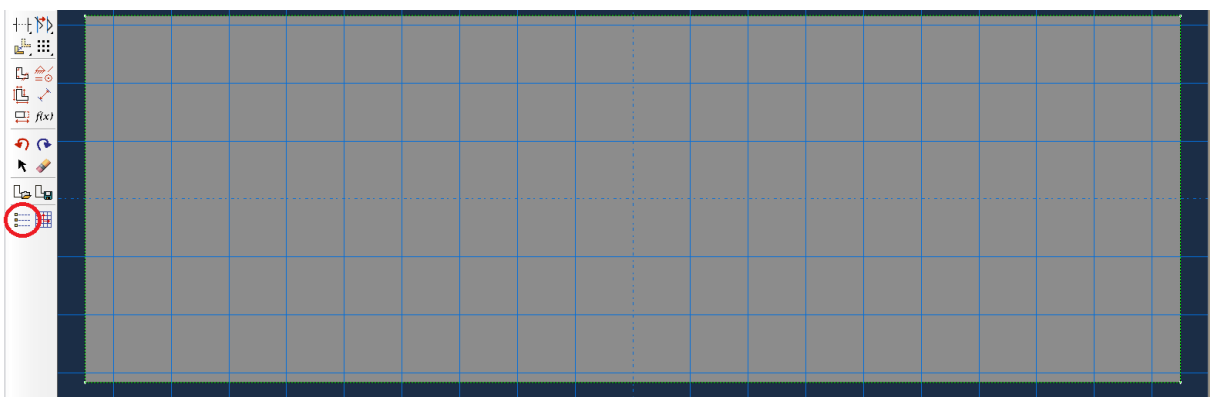
. Use  to auto fit the view. You will get the following figure. If you made some mistakes, you can always use  to delete the components you created. The position of your figure in the sketch is not important for that exercise.




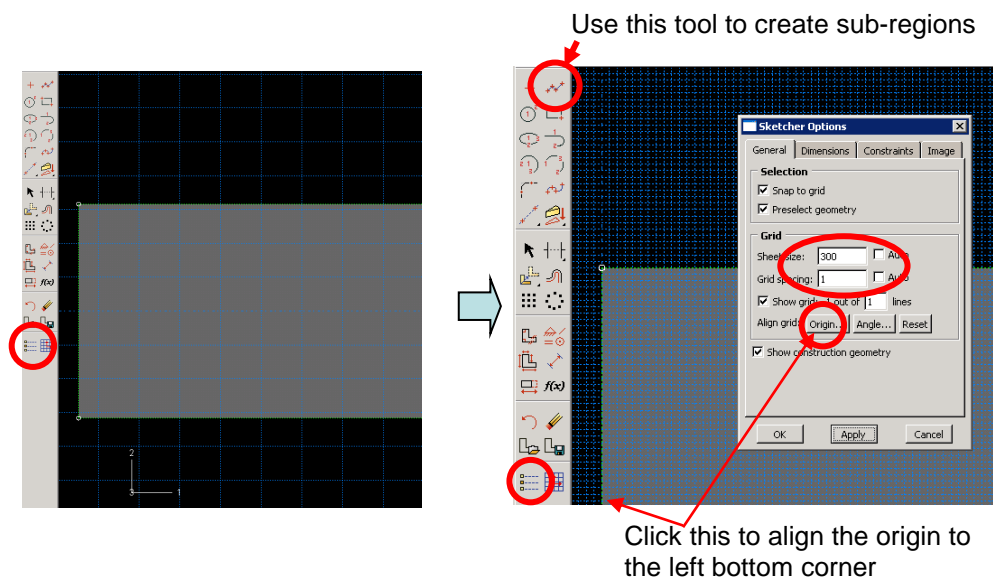
- Click **“Done”** to exit the sketch and return to the part module. ABAQUS/CAE uses the top-down methodology to create the model. You model the global geometry first and model the local features afterwards. Now you have created a region for the model.




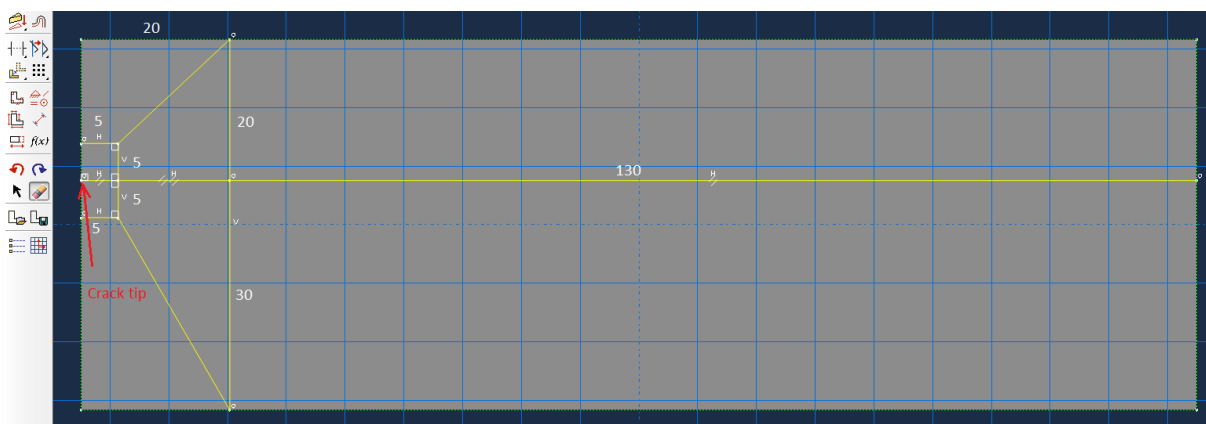
- Click the **“Partition tool”**  to cut the region into different sub-regions for different mesh density.



Before you start to partition, you can use the sketcher option  to adjust the grids and align the origin.





- Use  to create 11 lines (in yellow color). You can always use the right mouse button or push “Esc” to stop the procedure during creating the lines. The position of the points connecting the lines can be easily calculated from the dimensions given below. In fracture mechanics, because of the stress/strain concentration at the crack tip, we need to create fine mesh at the crack. In the region far away from the crack tip, course mesh can be used. In total there are 8 sub-regions in this model and there are two small regions around the crack tip.





- Once you have created these sub-regions, click “**Done**” and return to the part module. Now you have created a part for the half crack model with appropriate partitions.

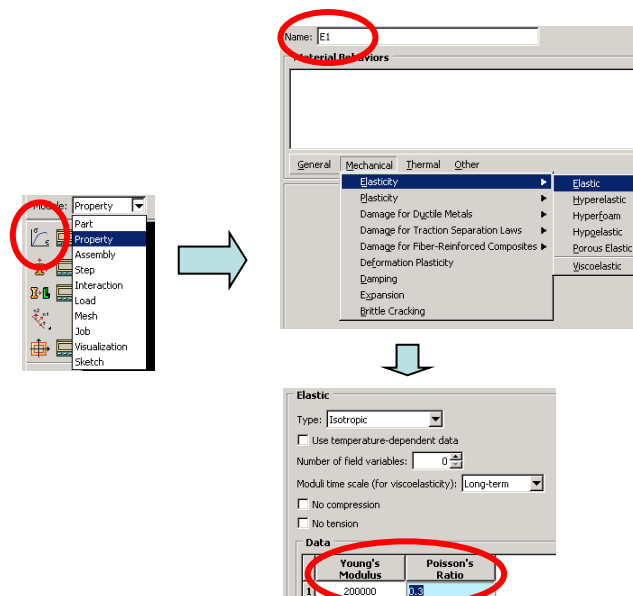



- Click  save the model to a file. Once you move to a new module always click . Before you close ABAQUS, copy the file to your home directory!

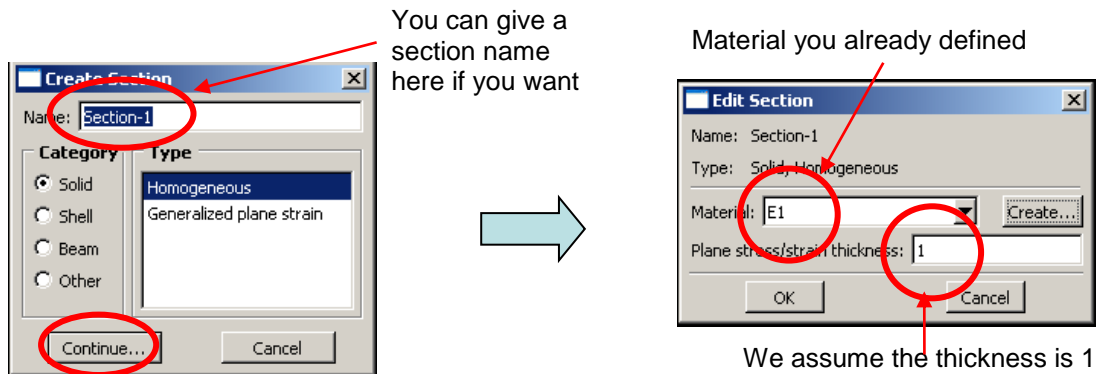
3.2 Assign the Material Properties

Next step is to create materials and the linear elastic material E1 will be analyzed first.


Choose “**Property**” module and click  to create the material, or alternatively, double click “**Material**” in the left tree. As you can see in the following figure, there are a lot of possibilities in ABAQUS, but in these exercises, we will only consider simple isotropic material models. When the material properties are entered, click OK. You can always use  to edit the material, section and section assignment if necessary, or alternatively, right click on the corresponding items in the left tree and choose “**Edit**”.

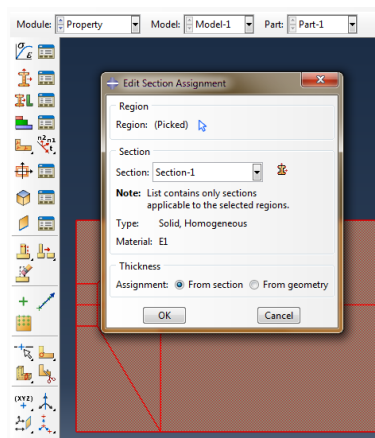


- Create a section. Now the material is defined, but it is not linked to any regions. To do so, we must specify a connection between the material and the regions. This connection is called “**Section**”. Click  to create a section based on the material, or alternatively, double click “**Section**” in the left tree.



The section is the connection between the material properties and the regions, a material must be selected. If the model has only one material, the material will appear in the box “**Material**”. For models with several materials, you will have to select the correct material for the section. The model you are going to analyse is a so-called plane-strain model, so the thickness is set to unity. This is why the thickness value of 1 is preset in the text-box “**Plane stress/strain thickness**”. Of course you can specify a different value if you want. Remember the force applied to the model is proportional to the specified thickness.

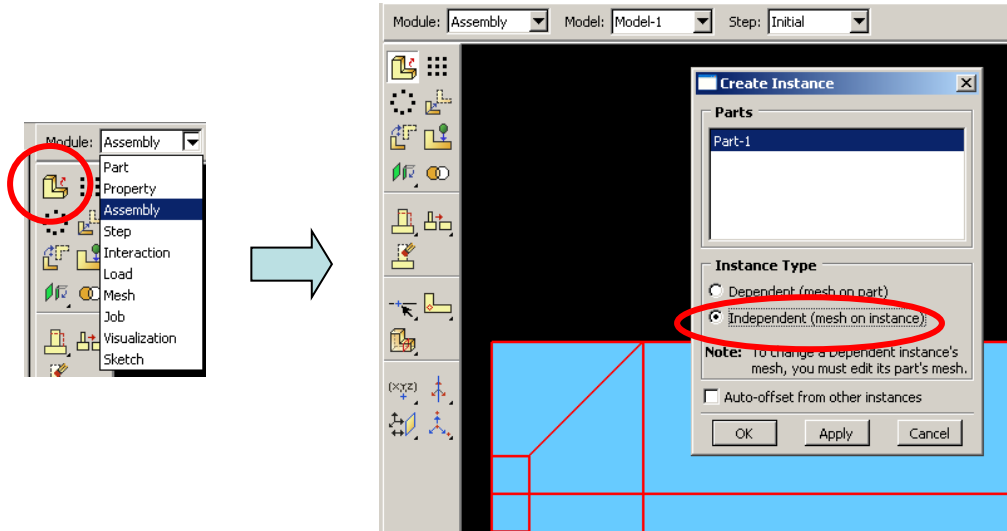
- You need to connect the section to the regions now. To do so, go to , or alternatively in the left tree, develop the “**Part**” module, and double click “**Section Assignment**”. Select all the regions in the part, and click “**Done**” at the bottom of the window, and following window will appear. Now select the correct section (in this model, you only have one section, so this will be pre-selected), and click “**OK**”. Now the material is linked to all the regions in the part. In advanced modeling, same part could contain (sub-) regions with different materials. In this exercise, all the regions will have the same material.



3.3 Define the Assembly

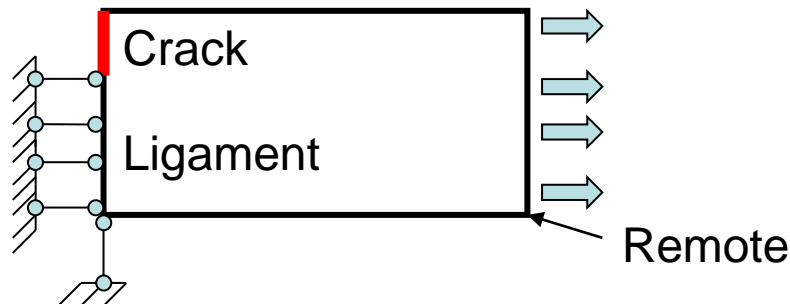
Each “**Part**” that you create is oriented in its own coordinate system and is independent of the other parts in the model. Although a model may contain many “**Parts**”, it contains only one “**Assembly**”. You define the geometry of the assembly by creating “**Instances**” of a part and then positioning the instances relative to each other in a global coordinate system. An instance may be independent or dependent. Independent part instances are meshed individually, while

the mesh of a dependent part instance is associated with the mesh of the original part. For this exercise you will create a single independent instance of your half tension model. ABAQUS/CAE positions the instance so that the origin of the sketch that defined the specimen overlays the origin of the assembly's default co-ordinate system. If using the left tree, develop the “**Assembly**” item, and double click “**Instances**” to create a new one.



3.4 Configure the analysis Step

Now that you have created your “**Assembly**”, you can configure your analysis. In this simulation we are interested in the static response of the half tension model with a half uniform tension 1 mm on the right hand side.



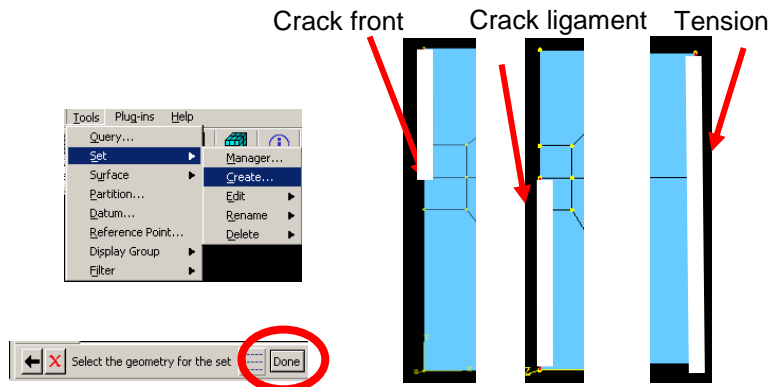
The analysis will consist of two overall steps:


- An **initial step**, in which you apply boundary conditions that constrain the specimen.
- An **analysis step**, in which you apply displacement at the right hand side of the specimen.

ABAQUS/CAE generates the initial step automatically, but you must create the analysis step yourself. You may also request output for any steps in the analysis.

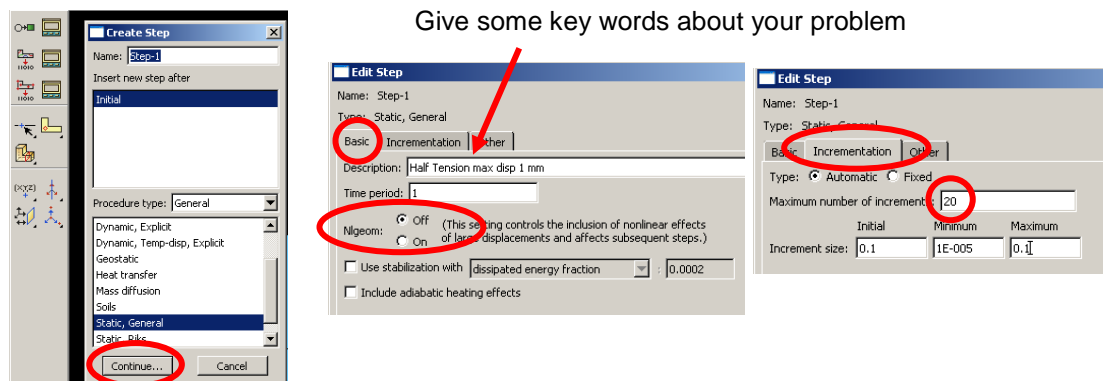
- In order to define the analysis steps afterwards, you need to create “**Sets**” (collection of nodes) for results output (you may not wish to output results for the whole model) and defining boundary conditions. In a model, usually there are parts of the boundary have the same boundary conditions and we can define the parts with the same boundary conditions as “**Sets**” in ABAQUS.

There are three sets in the half tension model to be created: Crack front, crack ligament and Tension. If using the left tree, double click “Sets” under “Assembly”, otherwise choose Tools in the top menu bar, then Set and Create. Note that you should press “Shift” on your keyboard to select elements simultaneously.



- In the module “Step”, choose  to create a step. Alternatively, double click “Steps” in the left tree. You need to assign some general values in “Basics” and “Incrementation” options.

The time period in a static analysis does not have any physical meaning. Time period “1” means the total time to perform the analysis of given displacement/load is “1” – 100% of the applied displacement will be used.




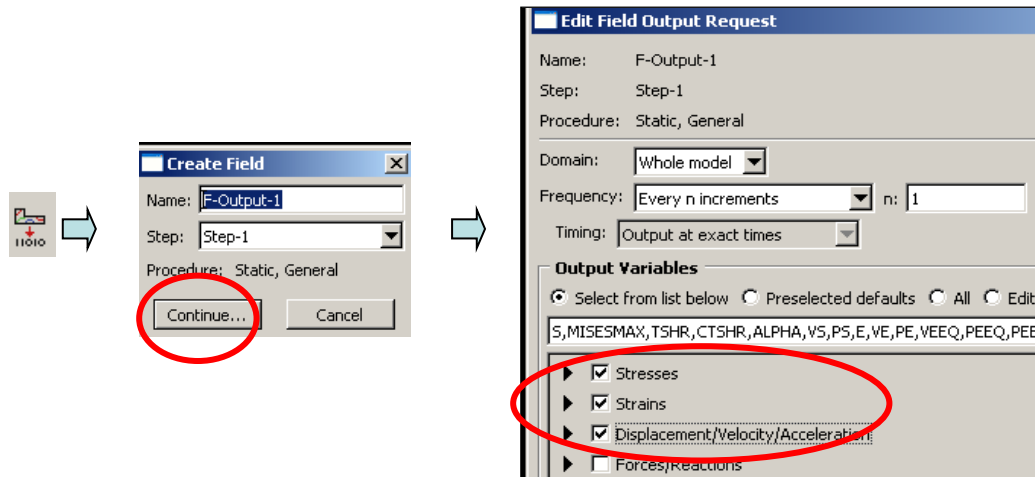
- Turn off the “Nlgeom” option. ABAQUS analyses can include and exclude the nonlinear effects of large deformations. The nonlinear geometrical effects (large deformation) will be neglected in this exercise.


By default, ABAQUS/CAE writes the results of the analysis to the output database (.odb) file. When you create a step, ABAQUS/CAE generates a default output request for the step. A list of the preselected variables written by default to the output database is given in the ABAQUS Analysis User’s Manual. You do not need to do anything to accept these defaults.

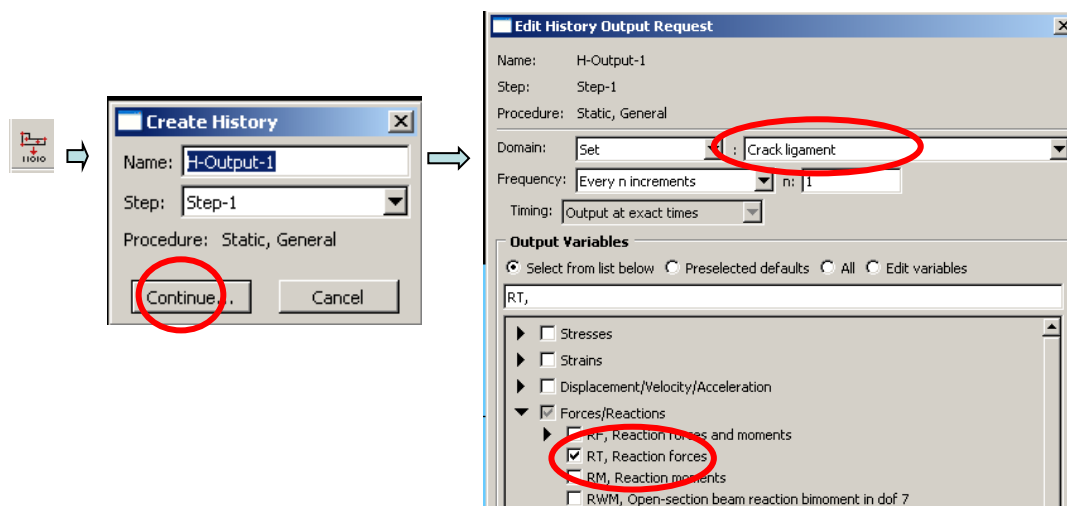
You use the “Field Output Requests Manager” to request output of variables that should be written at relatively low frequencies to the output database from the entire model or from a large portion of the model. You use the “History Output Requests Manager” to request

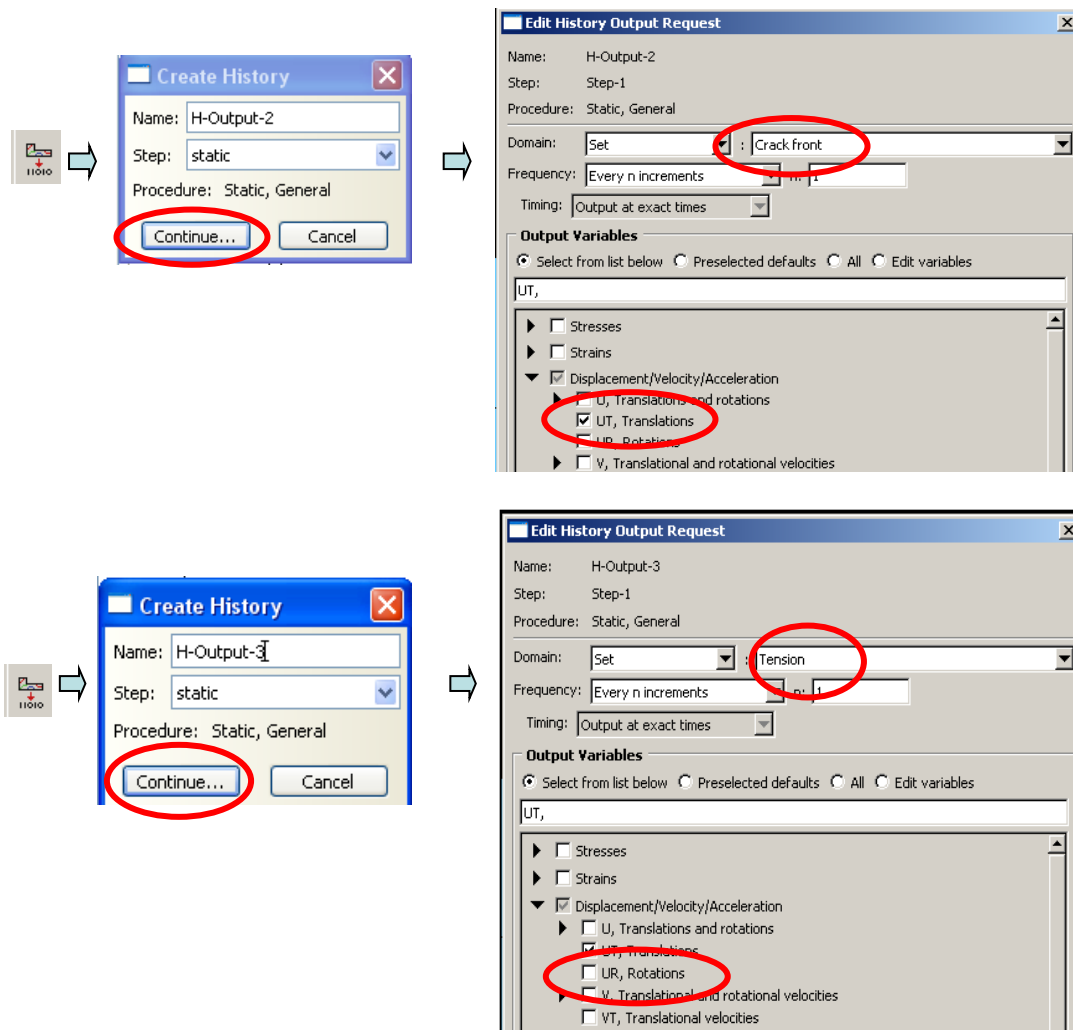
output of variables that should be written to the output database at a high frequency from a small portion of the model; for example, the displacement of a single node.

- Use  to define field output request, or alternatively double click “**Field Output Requests**” in the left tree. The default one is of no interest for you, so you can either delete it and create a new one, or modify it to take into account only Stresses, Strains and Displacement/Velocity/Acceleration.



- Use  to define history output request, or alternatively double click “**History Output Requests**” in the left tree. You can delete the default one (you do not need it) by right clicking on it and selecting “**Delete**”. Three history output requests can be made. The reaction output at set Crack ligament, and displacement output at sets Crack front and Tension.

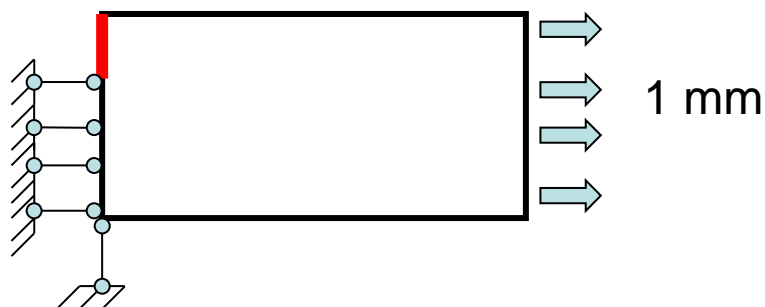




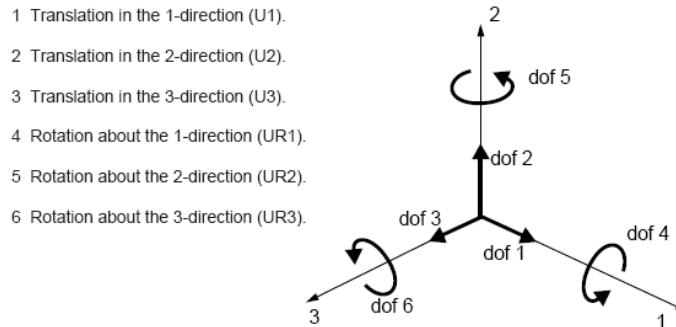
3.5 Apply the load and boundary conditions


Prescribed conditions, such as loads and boundary conditions, are step dependent, which means that you must specify the step or steps in which they become active. After that you have defined the steps in the analysis, you can define prescribed conditions.

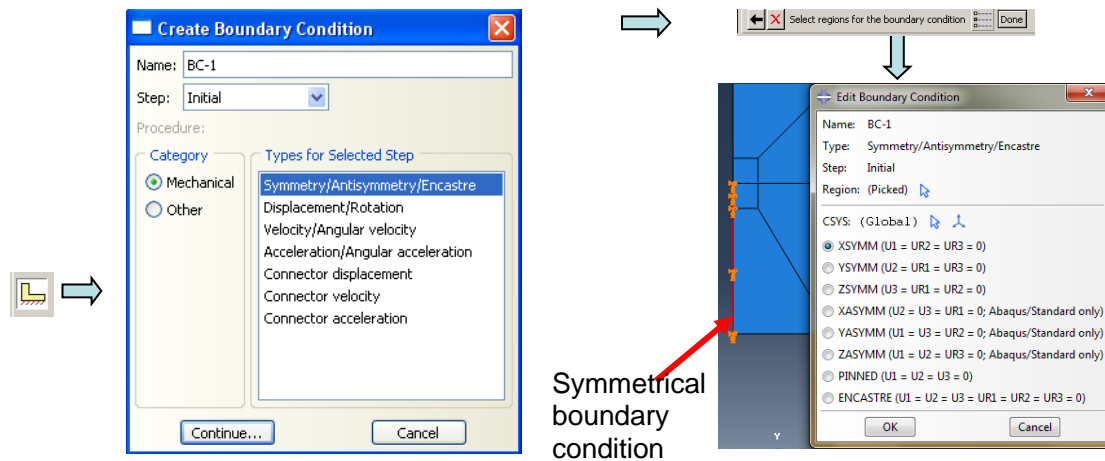
In structural analyses, boundary conditions BCs are applied to the regions of the model where the displacements and/or rotations are known. Such regions may be constrained to remain fixed (zero displacement and/or rotation) during the simulation or may have specified, non-zero displacements and/or rotations. The BCs for the half tension model are:




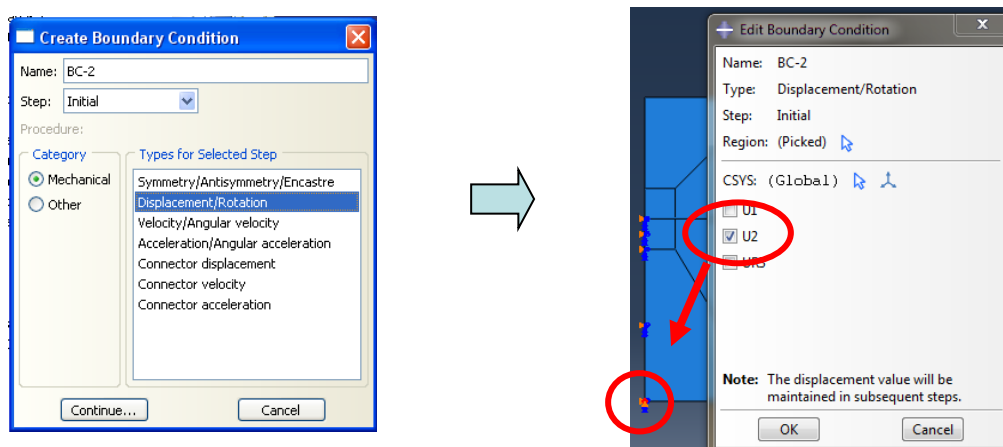
The labeling convention used for the displacement and rotational degrees of freedom in ABAQUS is shown in the following:




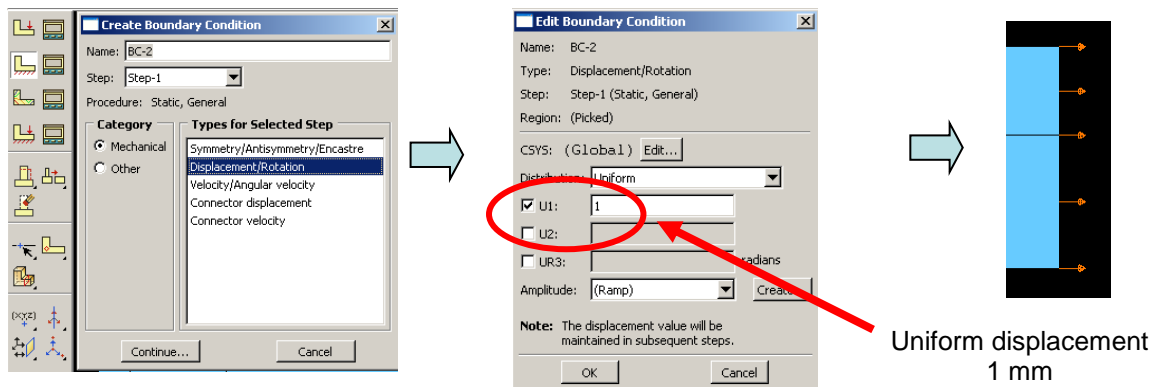
- In the module “Load” (or “BCs” in the left tree), use  to assign the **symmetric boundary conditions** at the crack ligament in the **initial step**. This symmetry condition is already implemented in Abaqus, and is called X-Symm.



- The model should have **no rigid body motion** in the vertical direction. Use  again to **fix the vertical displacement U2** at the left-bottom corner. This boundary condition should be defined in the **initial step**.




- Use  to create the **uniform displacement boundary condition** at right edge in the **static step**. Here “U1: 1” means that the displacement at the end of this analysis step will be 1 mm. The displacement increases from 0 mm (time =0) to 1 mm (time=1).




Final boundary conditions for the half tension model look like this:

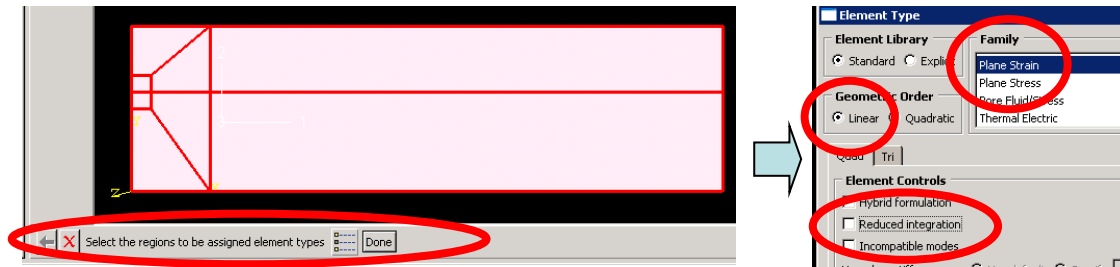



The loading in the tension model is controlled by the displacement and there is no concentrated/distributed loading in this tension model. If you wish, the horizontal displacement can be replaced by distributed loading by clicking , or double clicking “**Loads**” in the left tree.

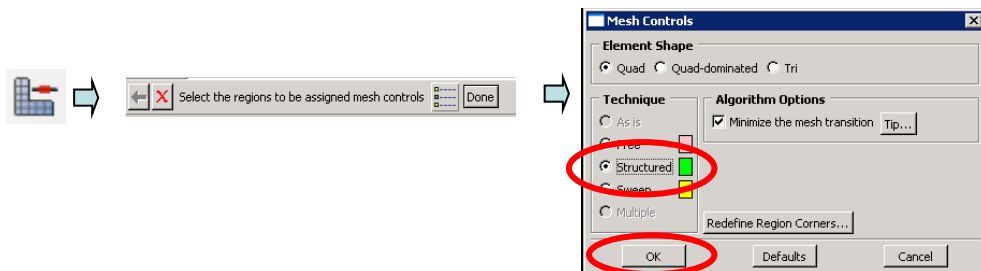
3.6 Mesh the model


You will now generate the finite element mesh in the module “**Mesh**”. Before you generate the mesh, you must specify the element type, the element size and the mesh generation technique.

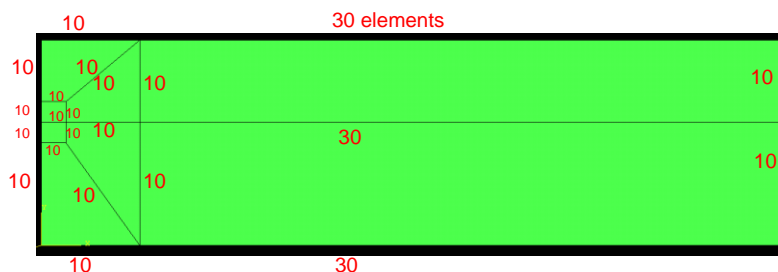
- Click  to choose the element type – “**plane strain**” with “**non-reduced integration**” for this exercise. This element is called CPE4.




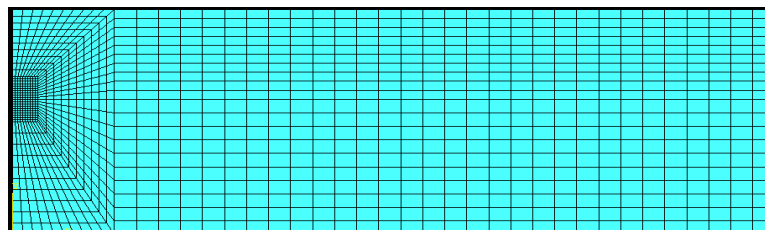
- Use  to specify the mesh generating technique – “**Structured**” technique for quad element shape. After “**OK**” the colors will shift to green which means the “**Structured**” technique can be applied in these regions.







- Click on the “**Seed edges**” icon  to assign the mesh seeds along the edges of all the regions. Select the edge(s) first and click on “**Done**”. Select “**by number**” in the “**Local seed**” windows newly opened and enter the number afterwards. The suggested numbers of elements along the edges are shown in the following figure:




- Click  to generate the mesh for the model. Finally you will get a mesh like this:

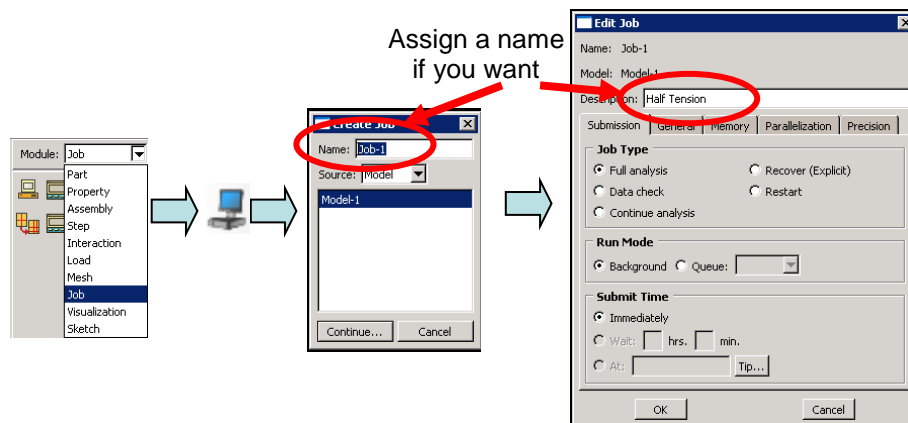



Note that if you click  for a while you will get more options with . The same is true for  → . In the left tree, the mesh can be found under “**Assembly**”, then “**Instance**”, and “**Part**”.

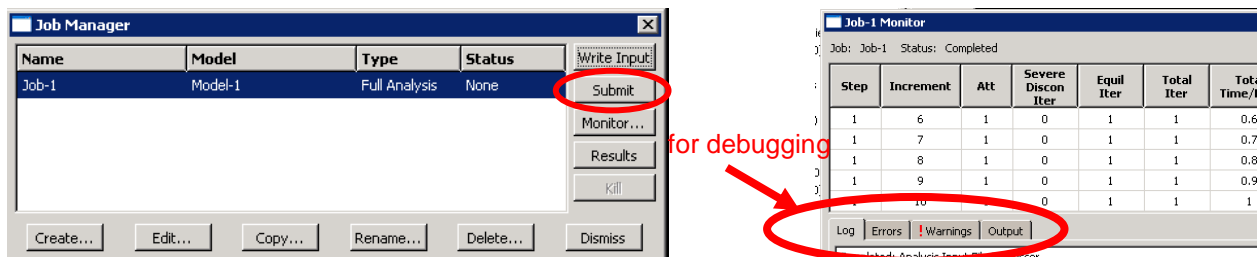
3.7 Create an analysis job

Now that you have configured your analysis, you will create a job that is associated with your model. Shift to the “**Job**” module.

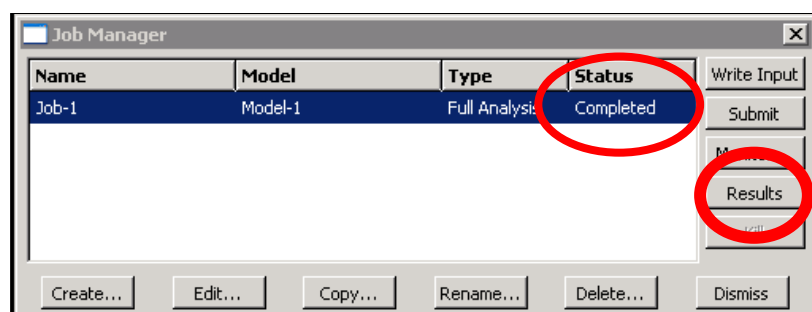
- Click  to generate the job, or alternatively, double click “**Jobs**” in the left tree.



- Choose job manager  to submit and monitor the job, or alternatively, right click on your job in the left tree, and select “**Submit**” and/or “**Monitor**”.



- If there is no error and the job finished properly the status will become “Completed” and click the “**Results**” to automatically enter the “**Visualization**” module. Alternatively, right click your completed job, and select “**Results**”.
- **YOU HAVE TO ASK FOR THE RESULTS**, entering the visualization mode is not enough. Failing to do that will induce post-processing results from a **previous** job!



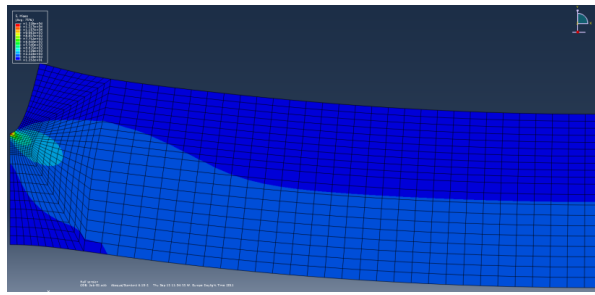
3.8 Post-processing the results

3.8.1 Graphical plots

The “**Visualization**” module of ABAQUS/CAE allows you to view the results graphically using a variety of methods, including deformed shape plots, contour plots, vector plots, animations, and X–Y plots. **You can save those plots to illustrate your report by using “print screen” or** In addition, it allows you to create tabular reports of the output data. For more information on any of the post-processing features discussed in this guide, consult ABAQUS/CAE User’s Manual.

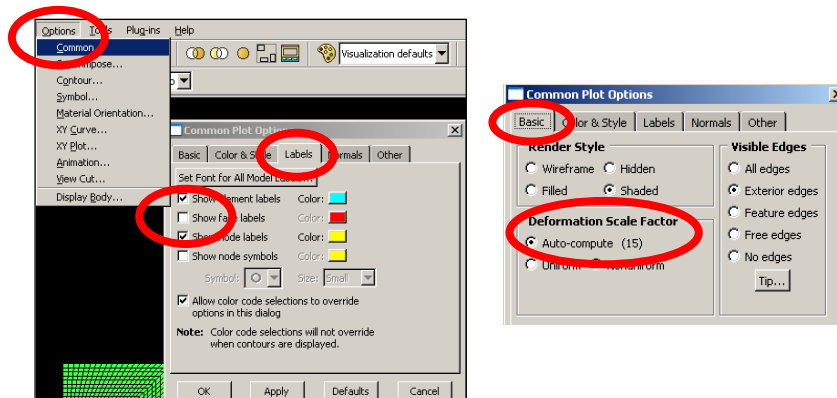
You can enter the “**Visualization**” module either by clicking the “**Results**” in the Job Manager window or click “**Visualization**” in the Module list located under the toolbar and select “**File → Open**”, select the **job.odb**, where job is the job name you assigned.

- You can use  and  to create fast contour and deformed plots.




The orientation triad indicates the orientation of the model in the global coordinate system. You can suppress the display and customize the title block, state block, and view orientation triad by selecting “**Viewport → Viewport Annotation Options**” from the main menu bar.

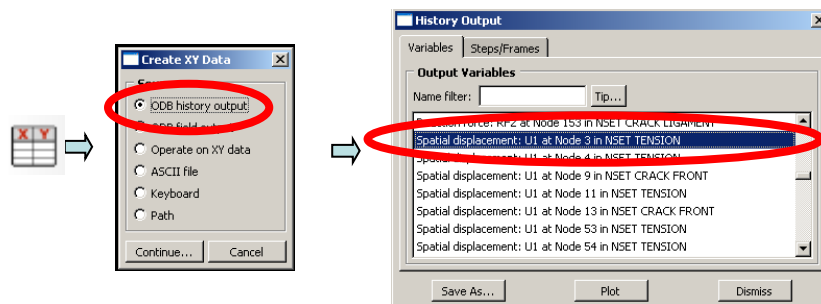
- If you want to display the node (or element) number in the visualization module, go to “**Options → Common → Labels**” and toggle on “**Show node labels**” (or element). Deformation in the deformed plots can also be scaled. You can go to “**Basic**” to change the deformation scale factor




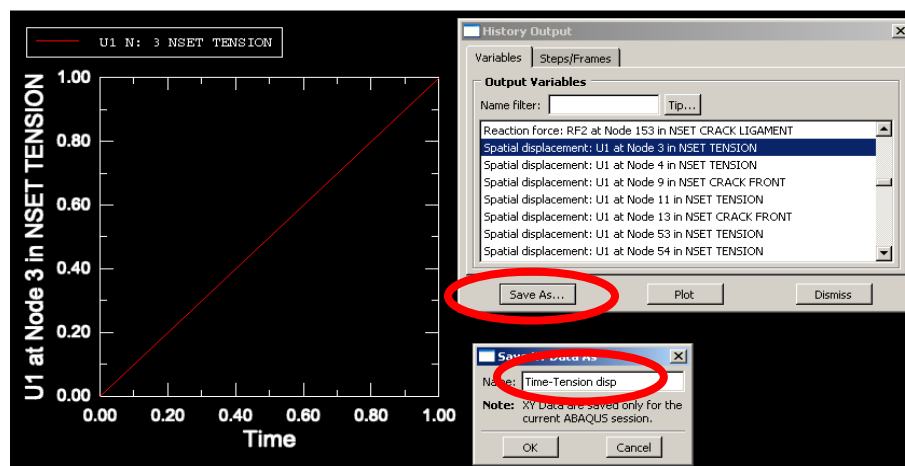
3.8.2 Global load-displacement curves

In finite element analysis it is important to plot the global load-displacement curve. When the job is completed, go to “**Create XY Data**”  in the “**Visualization**” module to plot the global load-displacement curves. It is very important that you **copy-paste** and **save** the curves you build to an **excel file**, since the created XY-data and XY-plots will not be saved in your results file. As soon as you close the Abaqus windows, they will be deleted, and you will have to create them once again afterwards.

- Choose “**Odb history output**”, select “**Spatial displacement U1**” at any node in the set “**Tension**” (because all the nodes in the **tension** set have identical displacement history) and **click “plot”** to get the time – remote displacement curve.

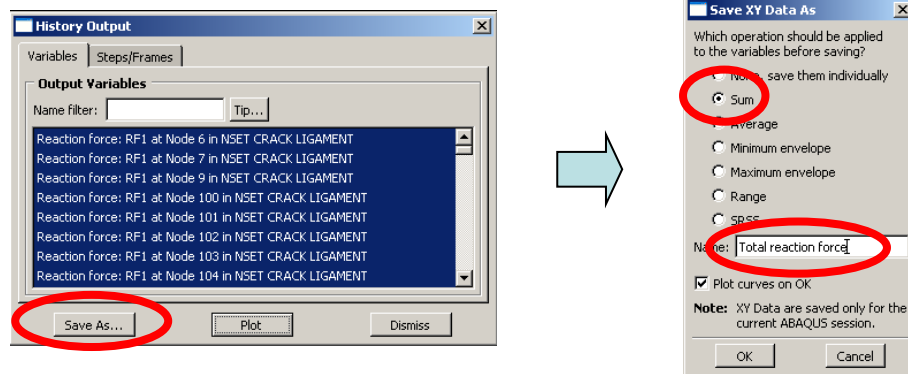



- “**Save as**” the curve with a name. We can use the XY Data Manager  to manipulate the curve later.

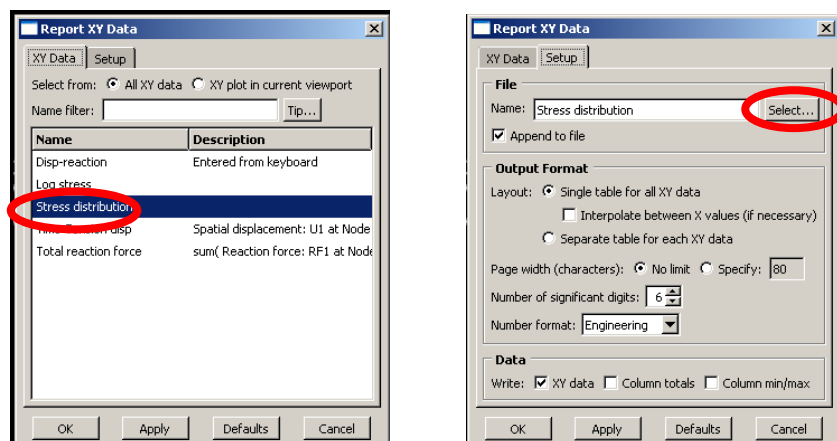


- Go to “**Odb history output**” again, select “**Reaction force RF1**” at all the nodes in the set “**Crack Ligament**” and plot. It is important to select all the nodes. Otherwise the total reaction force calculated will not be accurate. You will get a lot of time-reaction force curves. The individual reaction force at each node is not interesting for us. We are interested in the total reaction force. Choose “**Save as**” and “**Sum**” and assign a meaningful name to the quantity. Note: the total reaction force is negative in this model, because it points to the negative direction.

Now you have saved both the time-displacement and time-total reaction force curves for the model. From these two curves you can obtain the global displacement-load curve.

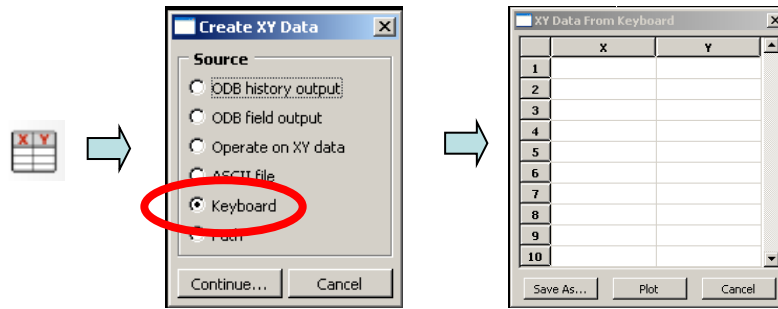


- You obtain the values for each data (columns) previously saved by going to the “**XY Data Manager**” . Choose the curve you are interested in and click on “**Edit**”. For the previously saved time-displacement curve for instance, the X column is the time and the Y column is the displacement.
- You can copy columns in an **Excel file** to generate new plots. For instance take the Y column of “time-displacement” and the Y column of “time-total reaction force” to later plot “displacement-total reaction force”. You can also directly export the data to a text file: “**Report → XY...**”

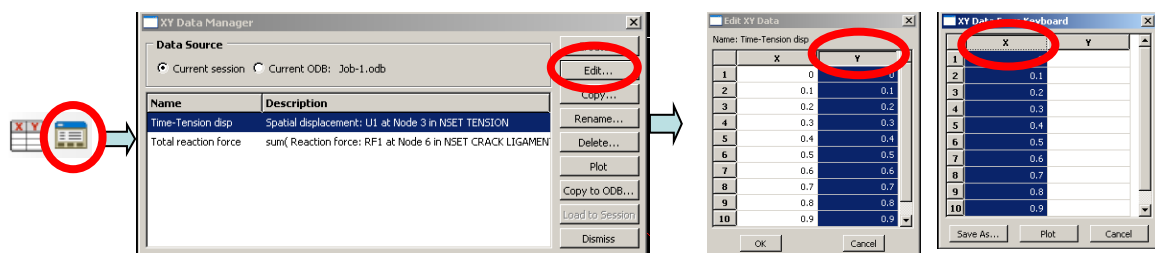




Select the XY-plot you want to export. Then select the tab “**Setup**” to tell ABAQUS/CAE where to store the file, and how the data should be presented. The first option is the filename. Note the check-box below the name-field “**Append to file**” is pre-checked, which means that the data will be added to the end of the file if it exists. The other options can just be as they are (in general). By clicking the button “**OK**”, the data is written to the file. The file containing the XY-data will be saved in C:\TEMP as default.

- If you want to work in the ABAQUS/CAE environment, prepare the plot of global displacement-load curve by opening a “**Keyboard**” table:



- **“Edit XY data”** for the time-displacement data and copy the Y column (displacement) to the X column in the **XY Data from Keyboard** window. In a similar way, **“Edit XY data”** for the time-total reaction force data and copy the Y column (total reaction force) to the Y column in the **XY Data from Keyboard** window. Use **“Plot”** and **“Save”** to plot and save the global displacement-load curve.



- You can use XY Plot Options  and XY Curve Options  to modify the appearance of the figures.

3.8.3 Nominal stress-strain curve of a tension specimen

Use the following equations to convert the global load-displacement curve of a tension specimen to its nominal stress-strain curve

$$\sigma = \frac{P}{A}, \quad \varepsilon = \frac{U}{L/2}$$

where P is applied load, A the loaded specimen area and L is the specimen length. Note that the applied load $P = -F$, where F is the total reaction force.

3.8.4 Bending stress of a bending specimen

Sometime it is beneficial to use bending stress instead of global load for a bending specimen. The bending stress can be calculated

$$\sigma_b = \frac{M}{I} \frac{w}{2}, \quad I = \frac{w^3}{12}$$

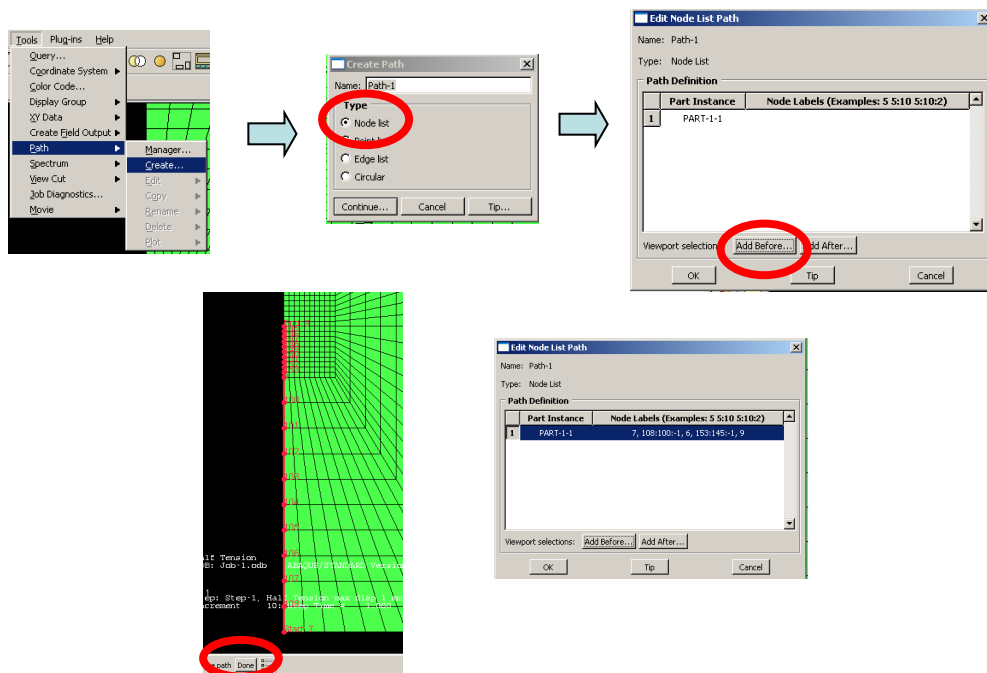
where M is the maximum bending moment, I the moment of inertia, and w the specimen width (see **Figure 1**). The bending moment can be calculated from the reaction force and half distance $M = P/2 * L/2 = PL/4$.


4. Elastic Fracture Parameter

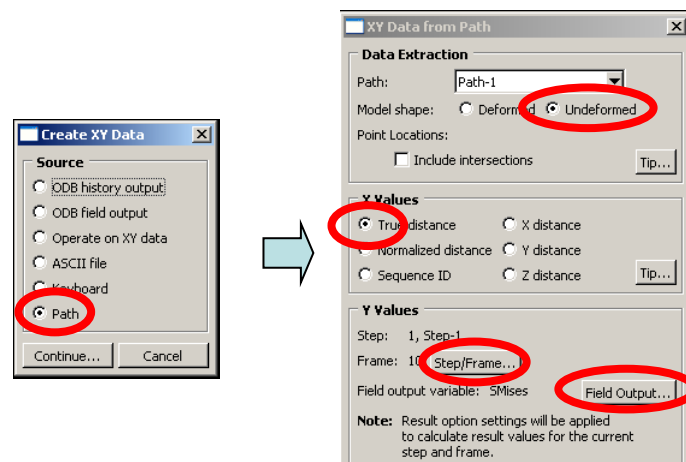
4.1 Calculate the K based on stress distribution along the crack ligament

Elastic fracture parameter - stress intensity factor (K) can be calculated from the stress distribution along the crack ligament. In order to display the stress distribution along a certain path, a path should be defined first in ABAQUS CAE **Visualization** module.

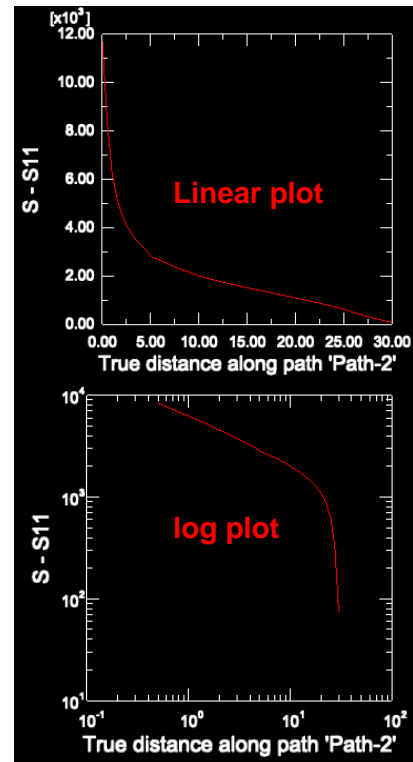
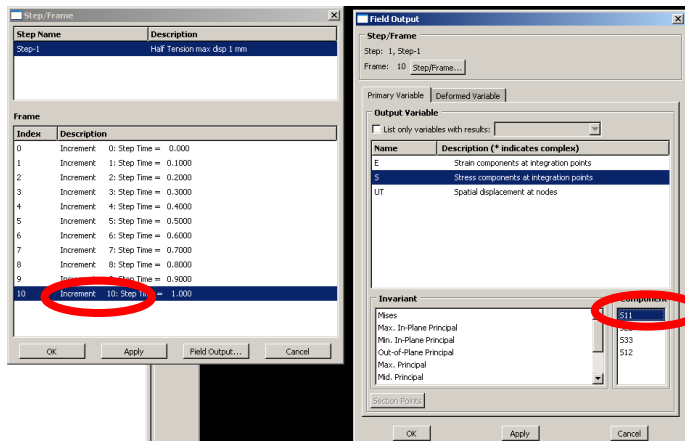
- Go to “**Tools → Path → Create → Node list**” to create the crack ligament path. After picking the correct nodes click “**Done**”. The created path with node numbers will be highlighted. You can edit the path using “**Path Manger**”.



- Once the Path is defined, you can go to Create XY Data  to plot the results. In the XY Data from path, you can first choose the right path (if there is more than one path available). Plot the results on the un-deformed shape and use true distance.
- Click the “**Step/Frame**” option to choose the results at the last increment. Click the “**Field output**” to choose the right crack opening stress (S11).



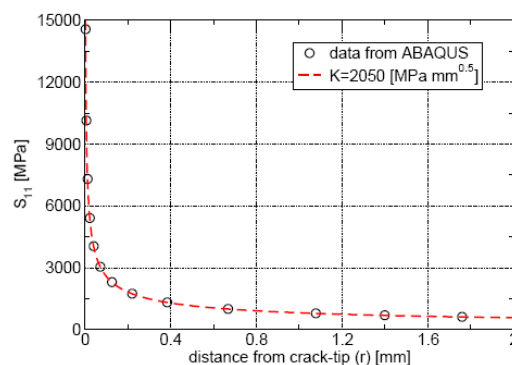
- **“Plot”** the stress distribution. You get typical stress distribution curve shown in the following figures. If the peak stress value appears on the right hand side, it means the path was not defined starting from the tip. The path should be re-defined.
- You can get logarithmic plot by using logarithmic type in **“XY Plot Options”**



- Save the stress – distribution plot for computing the K value.

Tips: The K can be either fitted from the stress distribution $\sigma_{11} = K_I / \sqrt{2\pi r}$ or determined from the tangent of the logarithmic plot. The following figure shows how the K can be directly fitted from the stress distribution. It should be noted that the K you fitted is strongly related to the applied load level to the model. The value shown in the following figure may have no relevance to your case. The K can also be determined from

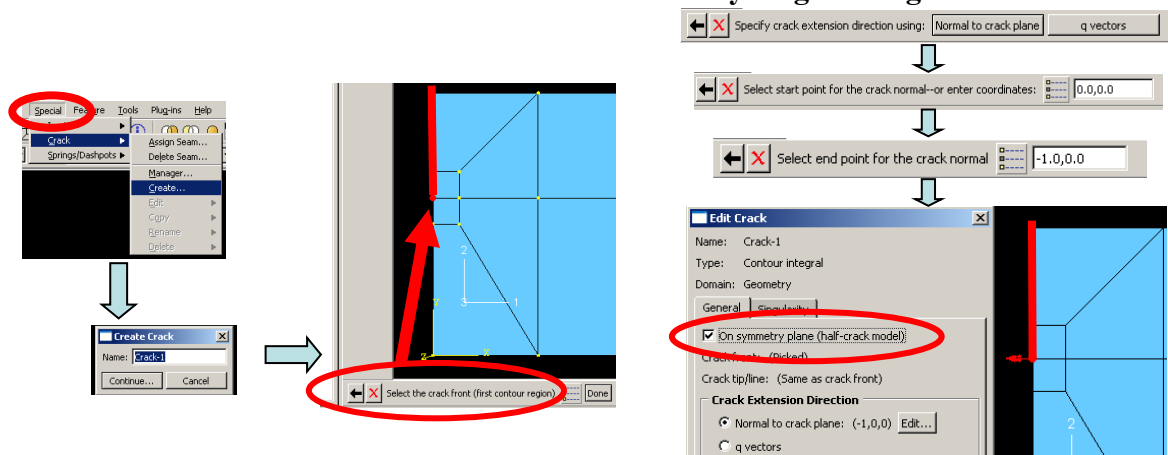
the $\ln \sigma_{11} - \ln r$ log plot by using the following relation: $\ln \sigma_{11} = \ln \left(\frac{K_I}{\sqrt{2\pi}} \right) - 0.5 \ln r$.



4.2 Define a crack to compute the J -integral


The elastic stress intensity factor (K) can also be calculated from the J -integral, which is used for most cases as an elastic-plastic fracture parameter. In order to calculate the J -integral, a crack entity should be defined in ABAQUS.

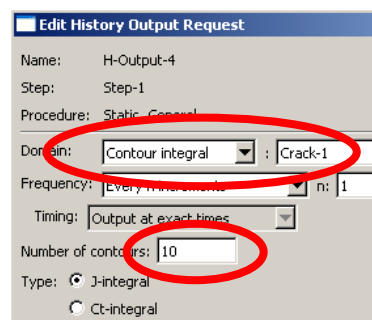
- Go to “**Interaction**” module, define the crack front (**Interaction** → **Special** → **Crack**)
- Define the “**crack normal**”. Make sure that the normal is pointing outwards. Please also tell ABAQUS/CAE that the crack lies on the symmetrical plane.
- The crack can be found in the left tree “**Assembly/Engineering features/Cracks**”



4.3 Create History Output for the J -integral

Once the crack is defined in the finite element model, you can create a new history output request for the J -integral.

- Go to “**Step**” module and click , or use the left tree. In the “**Edit History Output Request**” window, choose “**Contour integral**” and a “**Number of contours**” of 10.



- SUBMIT YOUR JOB AGAIN, the J -integral results will be available and go to “**Odb History output**” to post-process the J -integral- versus time data

4.4 K - J relation for elastic materials

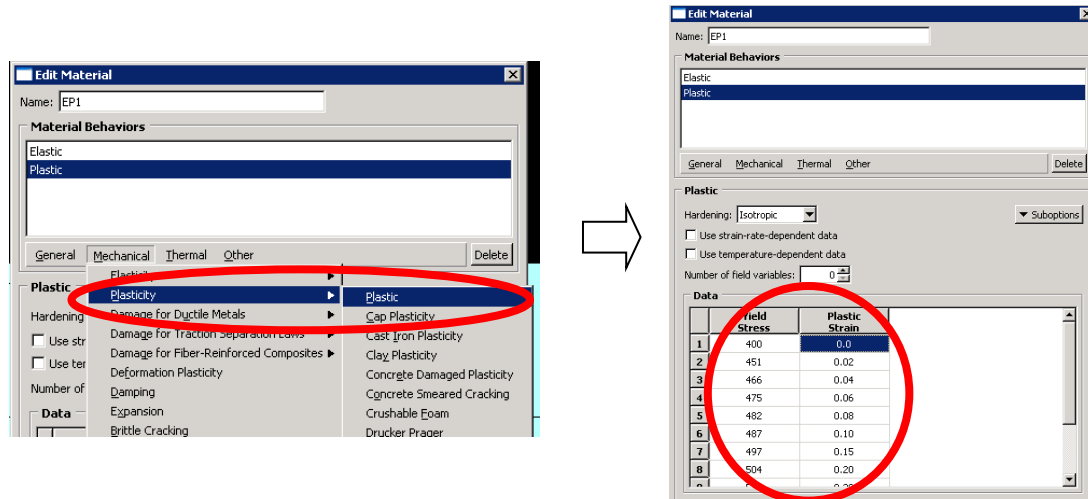
Following classical relation can be used to calculate the K from the J -integral: $J = K^2/E'$, where E' depends whether the problem is plane stress or plane strain. Consult the text book p.59 for details.

5. Elastic-Plastic Fracture Parameters

J-integral and CTOD are the two most common elastic-plastic fracture parameters. In order to calculate the elastic-plastic fracture parameters, you have to go to “**Property**” module to create a new elastic-plastic material (EP1, EP2 or EP3).

5.1 Create elastic-plastic materials in ABAQUS

- Input the elastic properties (Young’s modulus and Poisson’s ratio) first
- Input the plastic properties (stress in 1st column, plastic strain in 2nd column)



5.2 J-integral output

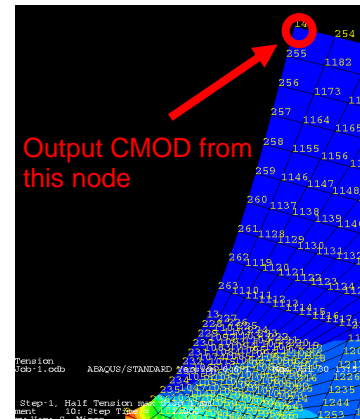
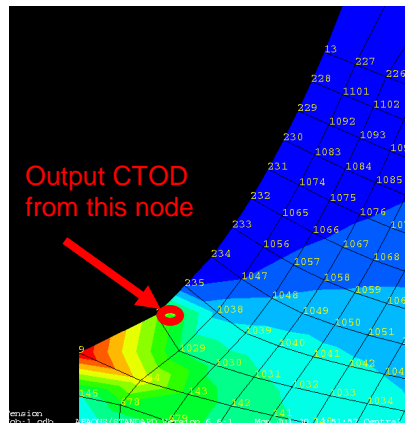
The same procedure as before can be used to output the J-integral for elastic-plastic materials.

5.3 Define the node for outputting CTOD/ CMOD

To output the CTOD and CMOD you need to identify the corresponding node numbers.

- In the “**Visualization**” module, go to “**Options** → **Common Plot Options** → **Show node labels**” to display the node numbers (as describe p. 23).
- Note down the CTOD node and CMOD node numbers. Use “**Odb history output**” to output the time-CTOD and time-CMOD curves. You can further plot the global load-CTOD/CMOD relations etc.
- Another way to get this output is to create two new “**Sets**”. The first called “CTOD” contains the node corresponding to CTOD, and the second called “CMOD” contains the node corresponding to CMOD. You can then create 2 corresponding “**History Output Request**” to ask for the displacement (UT: Translations) linked to the sets CTOD and CMOD. SUBMIT YOUR JOB AGAIN and once in the visualization module, create a new XY-data and/or XY-plot to display CTOD and CMOD relations.

The following figures are taken from the elastic case (E1 material), just to make it easier to spot CTOD and CMOD locations.

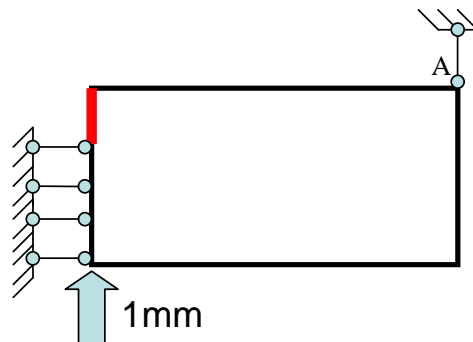


5.4 J-CTOD relations

You can plot the J -CTOD curves, which depend on the material strain hardening exponents.

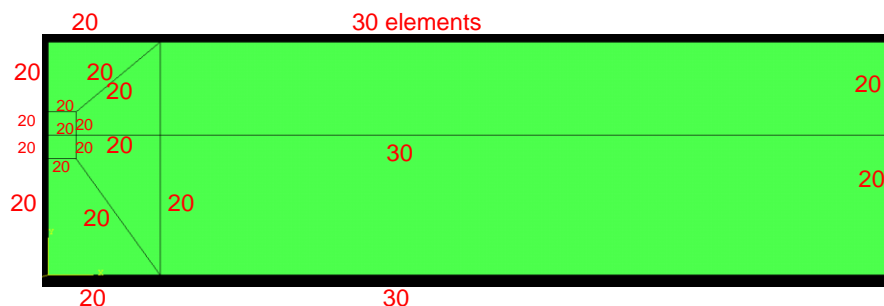
6. Tension versus bending

So far the tension model has been the focus. Select the **E1** and **EP1** materials, create a half bending model and compare the results with tension. The reaction force for the bending specimen can be easily calculated from point A in the following figure. To create that model, you should think about modifying the **Boundary Conditions** but also the **History Output**.



7. Mesh size effect

The finite element mesh size will possibly influence your results. Select the **EP1** material and make a new **tension** model with the following mesh specifications and compare the results.



You should keep in mind that a new mesh means new node numbering. Consequently, some of the output request and path need to be upgraded. The bending case can be treated only if you want to invest more but is not mandatory for the final report.

8. Computational Fracture Mechanics Report

8.1 Steps and time schedule (suggestion)

This section gives an outline of the expected progress of the work, including deadlines and content of the intermediate ABAQUS exercises. The more time you use on the intermediate exercises the less time you need for the final report. All the curves are not mandatory in the final report. They are suggested to help your understanding of the problem.

Week 36 to 37 - Create a FEM model with ABAQUS CAE

Linear elastic material E1, tension model

- Learn the ABAQUS CAE modeling concepts
- Create the geometry, assign the material, apply the boundary conditions
- Generate the mesh, perform the finite element analysis and view the results
- Global load – displacement curve
- Nominal stress – strain curve of the specimen (Young modulus?)

Week 38 to 39 - Calculate the stress intensity factor K

Linear elastic material E1, tension model

- Define the path to get “Sigma11 – distance from the crack tip” curve
- Calculate the K and discuss “ K – Distance from the crack tip” curve
- What about theoretical K in Anderson book?

Intermediate ABAQUS exercise 1 – deadline **Friday October 4th**

- **Describe the problem and theoretical background of the stress intensity factor, K**
- **Present and discuss the results you have so far**

Week 40 to 41 - Calculate the J -integral

Material E1 and EP0, EP1, EP2, tension model and bending model (EP1 only)

- Global load – displacement relation
- Define the crack to get the J -integral
- Discuss the J path independence (E1)
- Calculate the K value from the J -integral (E1, EP1?)
- “ J – global load or global displacement” curves

Week 42 to 43 - Calculate the $CTOD$

Material E1 and EP0, EP1, EP2, tension model and bending model (EP1 only)

- Define the $CTOD/CMOD$
- “ $CTOD/CMOD$ – global load or global displacement” curves
- $CTOD$ – J relations
- **Discuss the linearity of the previous curves (m parameter?)**

Week 44 to 45 - Effect of mesh

Elastic-plastic material EP1, tension specimen, finer mesh

- Nominal stress – strain curve
- Sigma11 – distance from the crack tip

- J – global load relation
- CTOD-global load relation
- CTOD-J relation

8.2 Graded report

Week 46 to 47 - Reporting

A final semi-technical report on the computational exercises is expected. The report must consist of not only figures but also **comments and discussions**. The report will be graded over 25 points. Here are some **suggestions** for the organization of the report:

- Problem definition, geometry and materials
- **Theoretical background (limited to the theory you need in the analysis of results)**
- Results and discussion
 - Tension (all materials)
 - “Global load – displacement” curves for tension specimen, all materials
 - Global and local deformed plots for tension E1, EP1
 - “Nominal stress – strain” curve for the tension specimen, all materials
 - “Sigma11 – distance from the crack tip” for E1 (EP1) and tension specimen
 - “K – Distance from the crack tip” for E1 (EP1) and tension specimen
 - “J – Nominal stress” curves for tension specimen, all materials
 - “CTOD / CMOD – Nominal stress” curves for tension, all materials
 - “CTOD – J” relations for tension specimen, comparison of four materials
 - Bending (E1 and EP1)
 - “Global load – displacement” curves for bending specimen, E1 and EP1 materials
 - Global and local deformed plots for tension and bending specimens, EP1
 - “K – Distance from the crack tip” for E1 and bending specimen
 - “J – Nominal stress” curves for bending specimen, E1 and EP1
 - “CTOD / CMOD – Nominal stress” curves for bending specimen, E1 and EP1
 - Mesh sensitivity (tension, E1 and EP1)
 - “Nominal stress – strain” curve, “Sigma11 – distance from the crack tip”, CTOD etc.
- Conclusions : you can finally draw conclusions about what you have understood from the computational fracture mechanics exercises.

Deadline ABAQUS final report: Friday November 15th

9. Computational Fracture Mechanics Networks

<http://cee-ux49.cee.uiuc.edu/cfm/home.html>
<http://www.cfg.cornell.edu/>