

# Plugin for Tunneling and Geotechnical Analysis in Abaqus

A. Fioranelli Jr, A. Cavalcante, T. Pessoa, A. Ferreira

SMARTTECH

*Abstract: Abaqus has one the best solutions for tunneling and geotechnical analysis due to the great non-linear capability. The Abaqus/CAE interface is not so friendly to define a tunneling analysis. It has a lot of steps and uses Model Change to remove and add elements, to simulate excavation a construction step). It's also difficult to change material properties during analysis. For non Abaqus users could be very hard to define a model. This paper shows a plug-in for Abaqus/CAE that helps the user to define a tunneling, excavation and temporary structures, with support facility and other geotechnical analysis. The paper explores the steps to define a geotechnical analysis and Abaqus/CAE capabilities. It also includes two examples of the plugin usage: a tunnel and a retaining wall.*

## 1. Introduction

Abaqus has a great solution to simulate geotechnical problems. Due to its non-linear capability, it can simulate properly problems involving rocks and soil materials. Additionally Abaqus can simulate concrete, steel and wooden materials, which are present in the most geotechnical problems. The software can also simulate those problems using implicit and explicit solver, depending on the problem, and has the capability to use both solutions (one for each STEP) on the same problem, if necessary. Flow analysis is also included, and can be used together to mechanical and thermal analysis.

It resumes the most important capabilities of Abaqus for geotechnical problems. Abaqus/CAE creates the geomechanical models for Abaqus/Standard and Abaqus/Explicit. But Abaqus/CAE could be improved, in order to make the model creation faster.

Due to its limitation, SMARTTECH geotechnical team decided to develop a plugin for Abaqus/CAE in order to make this process faster and easier.

This paper shows the plugin capabilities and also includes some examples to illustrates it.

## 2. Abaqus/CAE

Abaqus/CAE is a complete tool, very powerful and includes most of keywords for the .inp file. For people that have interest in geotechnical simulations in general, Abaqus/CAE is not an easy tool to create a model.

Here are some important limitations:

- \*MODEL CHANGE is used to simulate excavations and refill. This capability is used in Interaction Module where the user can activate or deactivate the elements. This is

included in Abaqus/CAE, but for a simulation with a large number of steps, it could take a long time to define all excavations steps. For tunneling analysis, when the user wants to consider several steps of rock excavation and then add concrete in place, the model change will be used a lot.

- \*FIELD is used to change the material property from a step to another. This tool can only be used editing the Abaqus keywords. Some other issues are detailed below.
- For each change (material or material removal/addition) is necessary a new step. Due to this problem, it is common in geotechnical problems that a simulation has a large number of steps. It is possible to create multiples steps in Abaqus/CAE, but joining the capability of creating multiple steps and define material change and removal/addition of material on the same window will help the users to define tunneling and geotechnical problems.

## 2.1 Material property - \*FIELD

To change the property material in some regions is necessary to use \*FIELD command.

This command is not supported by Abaqus/CAE. But this is not the most important issue. It is necessary to create a unique material with 2 different material properties. For example, in a simulation of a wall, is necessary to change the material property from soil to concrete. For that the user need to create a single material where all soil variables are included on field 1 and all concrete variables are included on field 2. All this information must be on the same material, once you can't define an element (a set) with 2 different Sections, or 2 different Materials.

It is quite confusing, especially for those that not used to Abaqus.

Due to this problem the plugin developed join different materials in a unique material (unique \*MATERIAL), separating each material (like rock, concrete, soil...) in different field variables. So all the model could have the same \*MATERIAL, and the changes during the simulation will be made using different field variables. The plugin will join all automatically.

## 3. The plugin

Based on these problems, it was decided to develop the plugin in order to minimize each one, using a unique window where the user can define in different steps which region will be deactivated, activated and also change it material properties.

Another important tool is the capability of joining different material in a unique \*MATERIAL, as explained on the previous chapter.

It is necessary for the user to know how to use Abaqus/CAE, the plugin does not replace it. The idea is just to help the creation of the geotechnical model.

The picture below shows the plugin window. It is necessary that that user define it area, each cell, which will be changed during the analysis. The user should also define the steps before starts the plugin, it could be used for any kind of step (for Abaqus/Standard and Abaqus/Explicit), but follows the Abaqus/CAE rules.

On the first column the user should define the Model that the change will be applied. The second column is used to define the set that will be changed.

The Initial Step column is used to define when the set will appear. In some cases the elements will appear during the analysis, such as reinforcement.

To define in which step the set will be changed, the user should use the Change Step. If the same set has more than one change during the analysis, the user should define a new line for the same set.

The Active column should be used to define if the set will be activated or deactivated. The two last columns are used to change the material of the set. For example, if initially a material is rock and then changes to concrete, the materials should be included in these columns.

Model	Set	Inicial Step	Change Step	Active	Initial Material	Material Change
Model-1	Set-meio	Initial	Escavacao_base	No	Rocha	rocha_concreto_b
Model-2	base	Initial	Escavacao_base	Yes	rocha_concreto_rr	Rocha
Model-1	material_base	Escavacao_base	Escavacao_meio	No	rocha_concreto_rr	Rocha
Model-1	material_base	Condicoes_iniciais	Escavacao_base	No	rocha_concreto_tr	rocha_concreto_rr
Model-1	material_base	Initial	Escavacao_base	No	rocha_concreto_rr	rocha_concreto_rr
Model-1	material_topo	Initial	Escavacao_meio	No	Concreto_tunel	rocha_concreto_b
Model-1	base	Initial	Escavacao_topo	No	Rocha	rocha_concreto_b

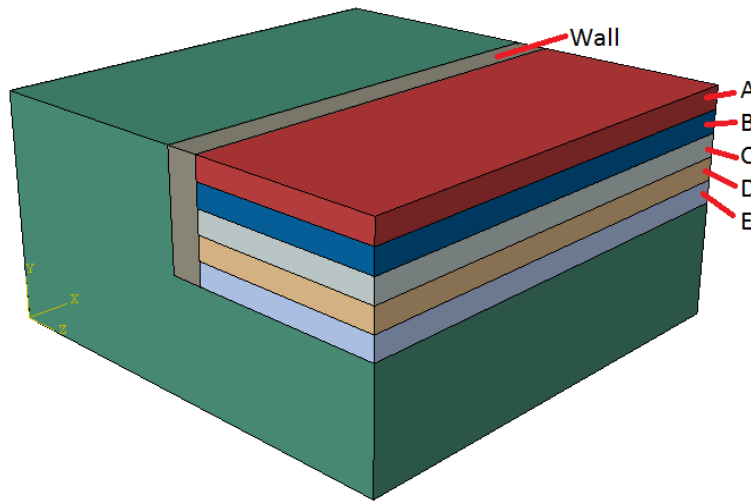
Figure 1. The plugin window

#### 4. Example 1

In this chapter we show a simple example of a construction of a retaining wall, it has 8 steps:

- Initial Steps where only gravity is applied (Geostatic Step) to consider the initial stresses.
- The second step is the wall excavation. A pressure is applied on the wall faces in order to simulate the mug pressure.
- For the third step the concrete wall replaces the wall mesh. The pressure is removed.
- The next 5 steps are used to simulate the excavation splitted into 5 steps.

The Picture below shows all the SETs defined on the model. One set it is for the wall, and there are other five sets (Step-A to Step-E) for each excavation step.



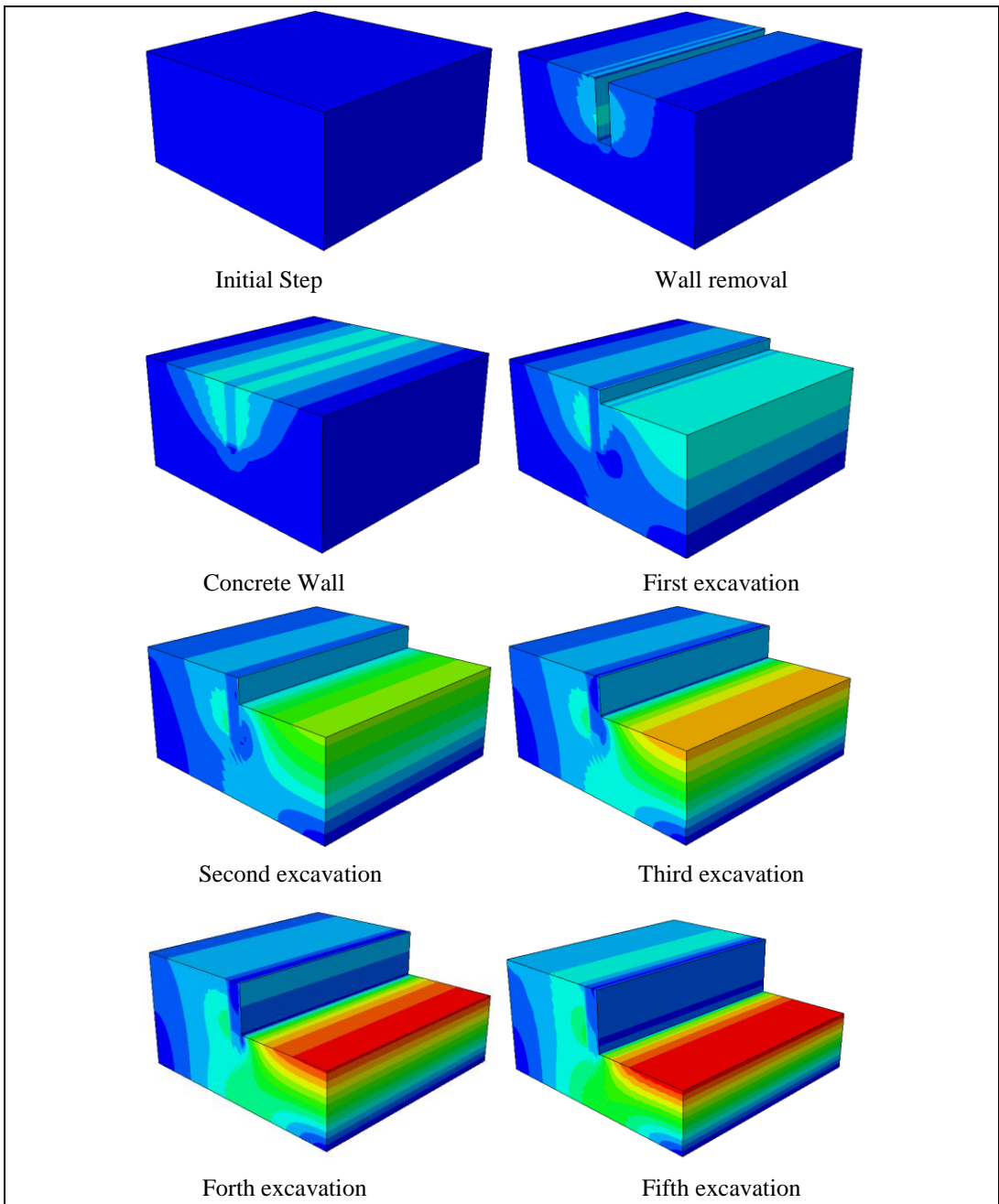
**Figure 2. The sets names**

The next picture shows how the model is defined using the plugin. Note that the materials, the steps and also the sets, must be defined.

Model	Set	Initial Step	Change Step	Active	Initial Material	Material Change
Model-1	Wall	Initial	Step-2	No	Rock	Rock
Model-1	Wall	Initial	Step-3	Yes	Rock	Concrete
Model-1	Step-A	Initial	Step-4	No	Rock	Rock
Model-1	Step-B	Initial	Step-5	No	Rock	Rock
Model-1	Step-C	Initial	Step-6	No	Rock	Rock
Model-1	Step-D	Initial	Step-7	No	Rock	Rock
Model-1	Step-E	Initial	Step-8	No	Rock	Rock

**Figure 3. The plugin window for Example 1**

The picture below shows the displacement results (U) for each step for this example. The objective of this paper is only to show the plugin application. So the results are not discussed.

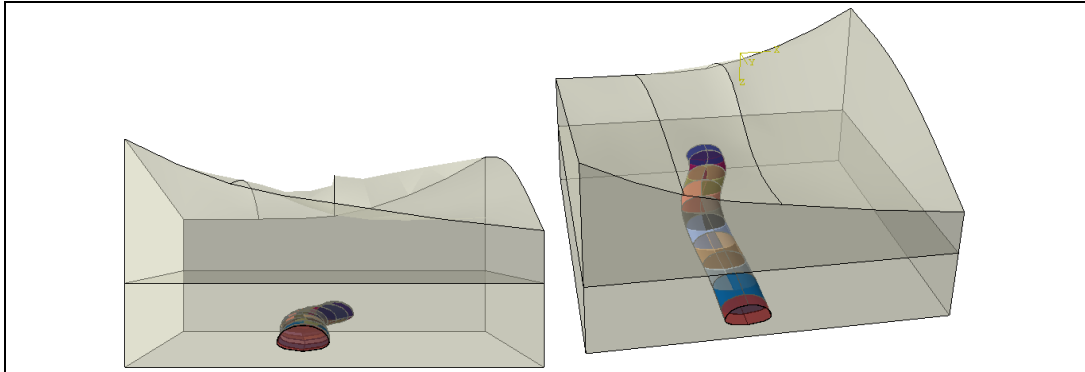


**Figure 4. The results for each step (U)**

## 5. Example 2

This next example shows a tunnel excavation simulation. It is not so complex, but it has a lot of repetitive steps. This kind of example is very appropriate to illustrate the plugin, once it has 21 steps.

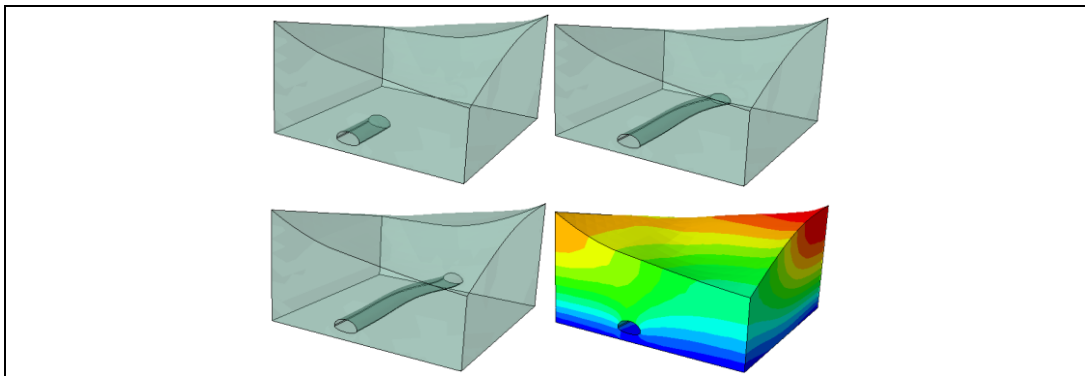
There are an initial step and 2 steps for each excavation step, the excavation and concrete addition on the tunnel border.



**Figure 5. The tunnel model**

The figure below shows some steps and the final results (U). The idea is this paper is not to discuss the results, but shows the plugin capability.

The excavation steps and concrete addition can be easily defined using the plugin.



**Figure 6. The tunnel model results**

## 6. Conclusion

This paper shows how the creation of a geotechnical model could be easier in Abaqus/CAE.

The plugin helps expert and new users of Abaqus/CAE that want to use Abaqus for geotechnical applications.

With this plugin the time consumption to prepare a model could be reduced considerably. It also helps to reduce error on model definition.

It can be used for most of the geotechnical problems:

- Tunnels.
- Dams.
- Retaining walls.
- Mining.
- Slope analysis.
- Wells.
- Excavation and construction problems in general.

The geotechnical team of SMARTTECH will continue developing this tool and creating new capabilities, such as the automatic definition of the tunnel profile.

## 7. References

1. S. Arndt, and B. Fillery, "The importance of simulation in geotechnical and mining applications," In Proceedings of Simulia Customer Conference. Barcelona, Spain. May 18-21, 2011.

[Visit the Resource Center for more SIMULIA customer papers](#)