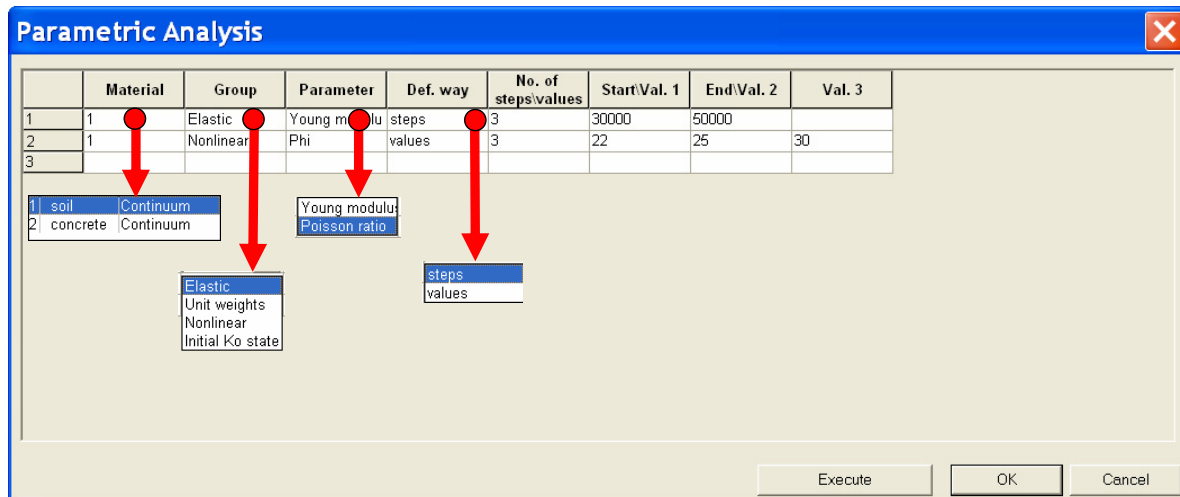


## 2.1.1 PARAMETRIC ANALYSIS

Window 2-4: Parametric analysis



- To add a new property to be varied
  1. Click by mouse on first free row under column **Material** and select one of the materials already defined in the data
  2. Click by mouse next right column (under **Group**) in the same row and select subgroup of properties from which the property will be selected
  3. Click by mouse next right column (under **Parameter**) in the same row and select one of the properties to be varied
  4. Click by mouse next right column (under **Def. way**) in the same row and select one of the options **Steps** or **Values**; if you select **Steps** then the definition is completed by setting number of steps (in column **No. of steps/values**) and the two limiting values; if you select definition through **Values** then the definition is completed by setting number of values (in column **No. of steps/values**) and list of these values placed in next columns; in the example shown in the figure Young modulus will be equal to  $E = 30000, 40000, 50000$ , kPa while friction angle to  $\phi = 22^\circ, 25^\circ, 30^\circ$  respectively
- To delete property from the list put cursor in the first column (the whole row will be highlighted) and press button **Delete**
- To run all automatically generated data files press button **Execute**;

### Remark:

1. By pressing **Execute** program prepares all data files (\*.inp files) that include all possible combinations of parameters to be varied and runs them immediately; each data file inherits a name of the template data file and an automatic suffix is added to it; this suffix consists of a chain of numbers (as many as number of properties to be varied) separated by underscore character; if we take the data file **foot** for instance and will perform parametric analysis

of settlements assuming that Young modulus is parametrized by Steps (first row in the table shown in the Figure) and friction angle is parametrized by Values (second row in the table shown in the Figure) then the set of 9 data files will be created; the data file labeled as **foot\_2\_3** corresponds to the template data file **foot** second step of the first parameter ( $E$  modulus) and third value of the second parameter ( $\phi$ );

2. Plotting selected nodal (displacements / pressures) or element (stresses / forces) results vs one of the varied parameters can be made with aid of the graphical postprocessor (refer to section 2.5.1.3)

---

Window 2-4
------------