First, let's have a look at some information (from the **PSpice Reference Guide**) concerning the .STEP command and the capacitor device definition in PSpice (or generally in SPICE programs):

.STEP (parametric analysis)

Purpose:

The .STEP command performs a parametric sweep for all of the analyses of the circuit—such as .DC (DC analysis), .AC (AC analysis), and .TRAN (transient analysis) – they are performed for each step.
Once all the runs finish, Probe displays nested sweeps as a family of curves.

General form:

```
STEP LIN <sweep variable name>
+ <start value> <end value> <increment value>

.STEP [DEC |OCT] <sweep variable name>
+ <start value> <end value> <points value>
```

`.STEP <sweep variable name> LIST <value>*` **– this is your case**

The *<sweep variable name>* can be one of the types described below:

Sweep Variable Name	Meaning
source	A name of an independent voltage or current source. During the sweep, the source's voltage or current is set to the sweep value.
1. model parameter	A model type and model name followed by a model parameter name in parenthesis. The parameter in the model is set to the sweep value.
temperature	Use the keyword TEMP for *<sweep variable name>*. The temperature is set to the sweep value. For each value in the sweep, all the circuit components have their model parameters updated to that temperature.
2. global parameter	Use the keyword PARAM, followed by the parameter name, for *<sweep variable name>*. During the sweep, the global parameter's value is set to the sweep value and all expressions are reevaluated.

Capacitor

General form:

```
C<name> <(+) node> <(-) node> [model name] <value> [IC=<initial value>]
```

Model form:

```
.MODEL <model name> CAP [model parameters]
```

If **[model name]** is left out, then <value> is the *capacitance* in farads. If **[model name]** is specified, then the value is given by the capacitor model parameters; see **Capacitor value formula** below the table.

Capacitor model parameters table

Model parameters	Description	Units	Default
C	**capacitance multiplier**		**1.0**
TC1	linear temperature coefficient	$°C^{-1}$	0.0
TC2	quadratic temperature coefficient	$°C^{-2}$	0.0
T_ABS	absolute temperature	°C	
T_MEASURED	measured temperature	°C	
T_REL_GLOBAL	relative to current temperature	°C	
T_REL_LOCAL	relative to AKO model temperature	°C	
VC1	linear voltage coefficient	V^{-1}	0.0
VC2	quadratic voltage coefficient	V^{-2}	0.0

Capacitor value formula

If ***[model name]*** is specified, then the value is given by:

```
<value>·C·(1+VC1·V+VC2·V2)·(1+TC1·(T-Tnom)+TC2·(T-Tnom)2)
```

where *<value>* is normally positive (though it can be negative, but *not* zero). *Tnom* is the nominal temperature (set using TNOM option).

The solution

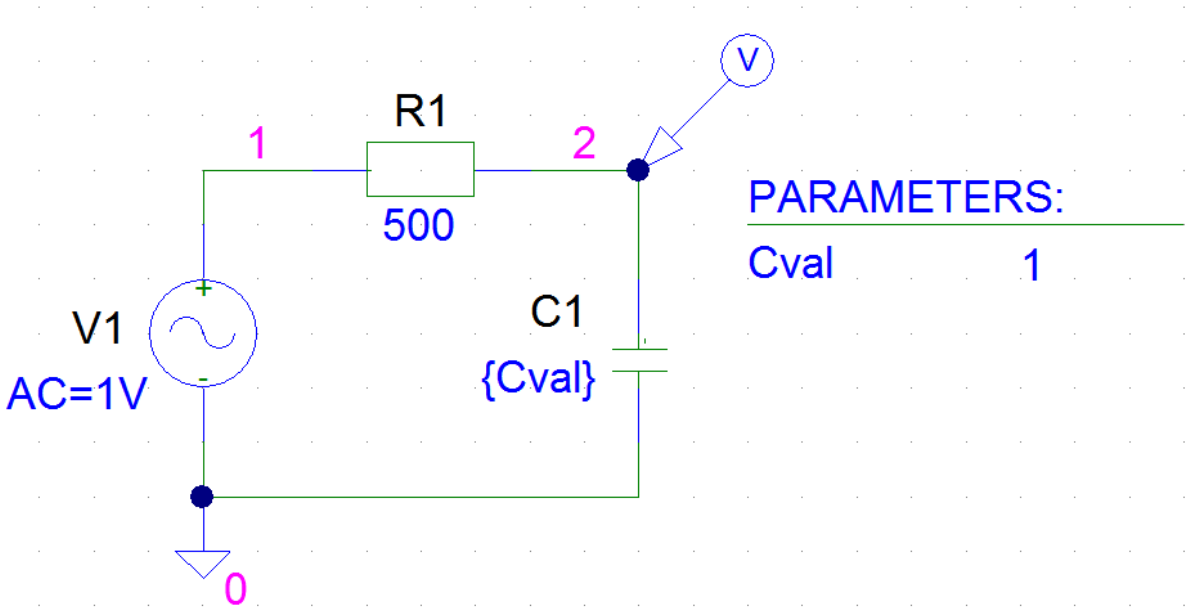
Now, finally to the task itself – there are two possible ways of doing it:

1. One way (the simpler one) of stepping a capacitor in the required steps 220p, 400n, 2u, using **a global parameter** is as follows: (item #2 in the above table of the **Sweep Variable Names**)

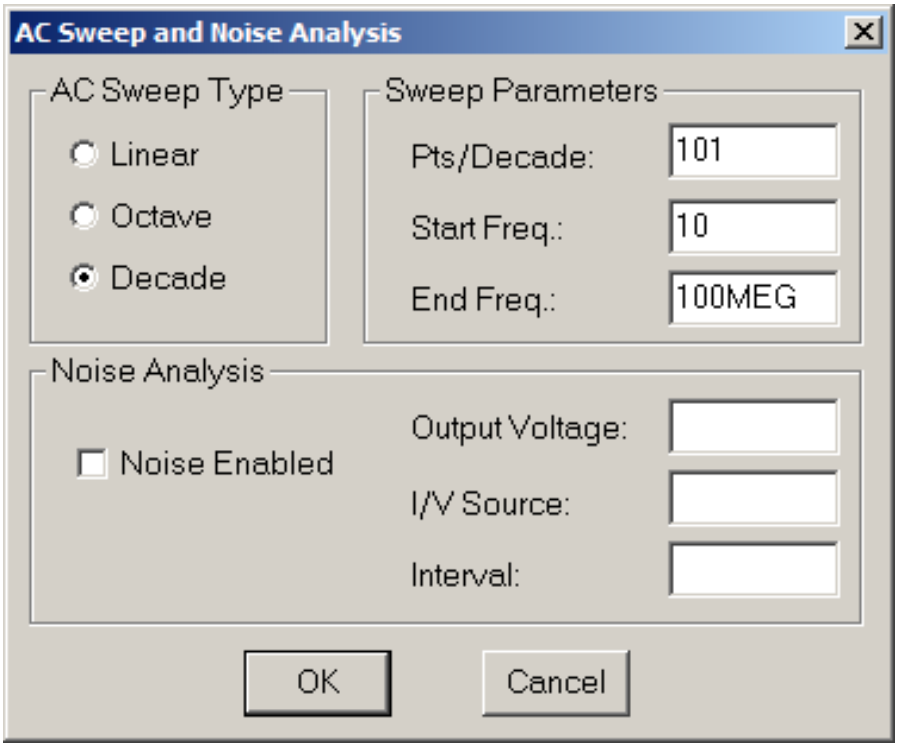
```
.PARAM Cval = 1
C1 2 0 {Cval}
.STEP PARAM Cval 220p, 400n, 2u
```

The parameter **Cval** is global and PARAM is the keyword used by the .STEP command when using a **global parameter**.

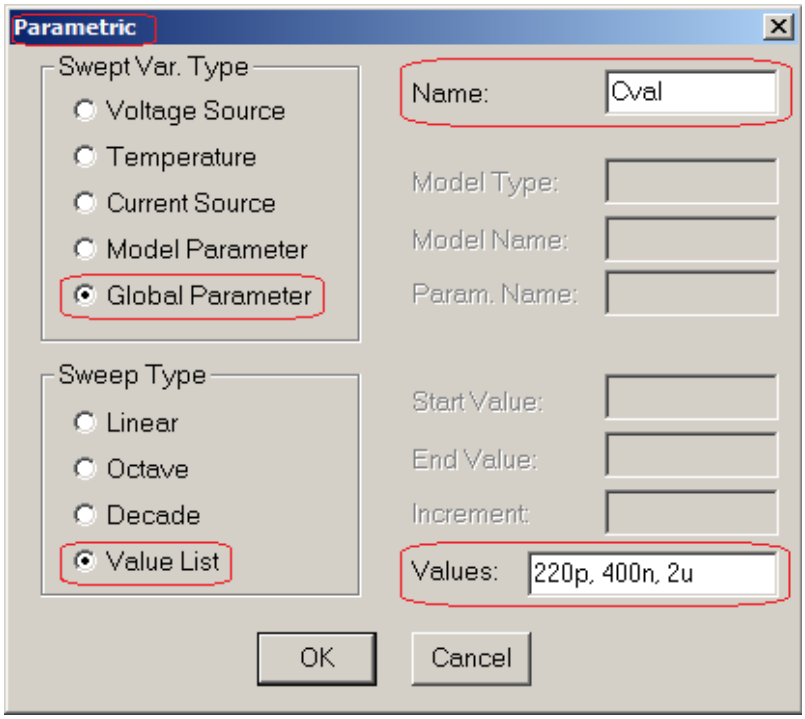
In your task, for instance, I have chosen the AC analysis to be done. The corresponding Schematics window content is shown in the picture Pic. 1. The AC Sweep parameters used in this example are shown in the picture Pic. 2, the parameters for the .STEP command in the picture Pic. 3, and the results of the analysis in the picture Pic. 4.



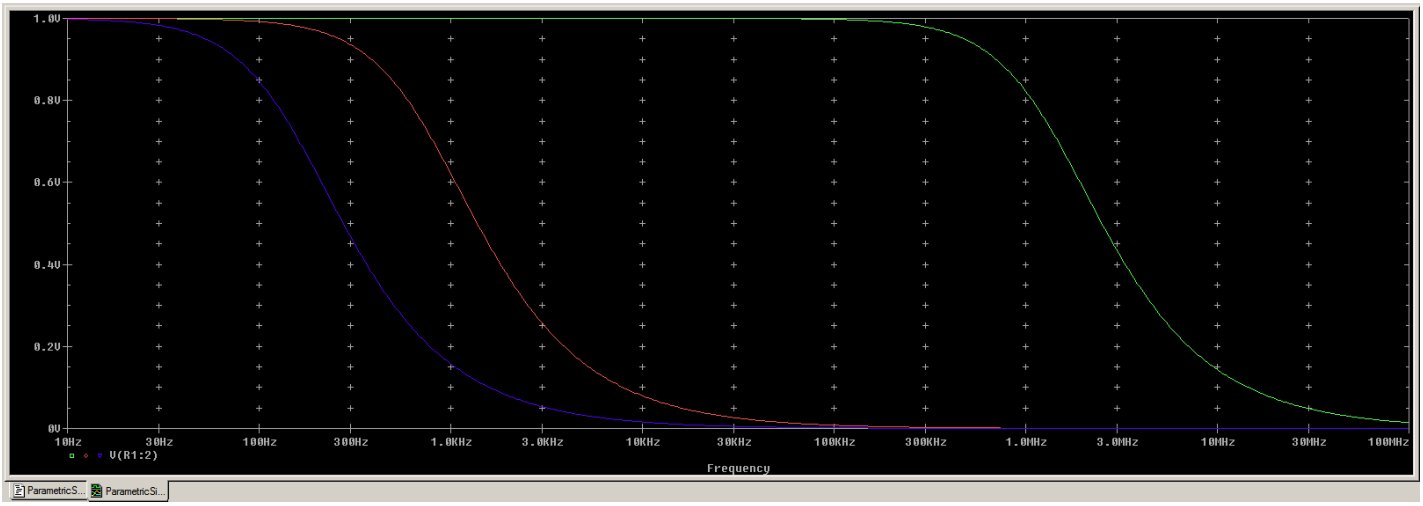
Pic. 1: The Schematics window content – case 1



Pic. 2: Input of AC analysis (AC Sweep) parameters



Pic. 3: Input of the parameters for the .STEP command – case 1 (global parameter Cval is stepped)



Pic. 4: The results of the AC Sweep in the Probe window (i.e., it contains the above mentioned family of curves)

2. The following example steps the capacitor **model parameter** C. This is another way of stepping a capacitor according to a list of chosen values (a bit more complicated way than the first one). (item #1 in the above table of the **Sweep Variable Names**)

```
C1 2 0 Cmod 1
.MODEL Cmod CAP(C=1)
.STEP CAP Cmod(C) 220p, 400n, 2u
```

It is the case when the capacitor **[model name]** is specified, see the Capacitor description above. Here **Cmod** is the **model name**, **CAP** is the sweep variable name (**a model type**), and **C** is the parameter within the model **Cmod** to step (see the **Capacitor model parameters table** above).

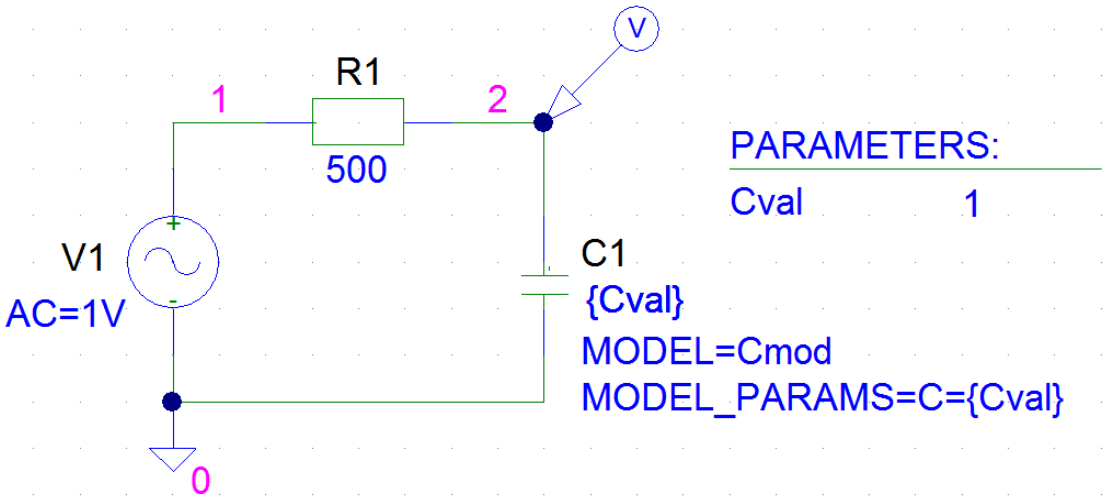
To step the value of the capacitor C1, the line value of the capacitor is multiplied by the **C** parameter value (that is stepped) to achieve the final capacitance value, that is:

```
final capacitor value = line capacitor value · C
```

Therefore, if the line value of the capacitor is set to **1** (assigned by the parameter **Cval** in the Schematics window; can be also simply assigned by entering **1** for the capacitor value instead), the final capacitor value is **1 · C** (i.e. **C**). Stepping **C** then steps the capacitor using the values 220p, 400n, 2u.

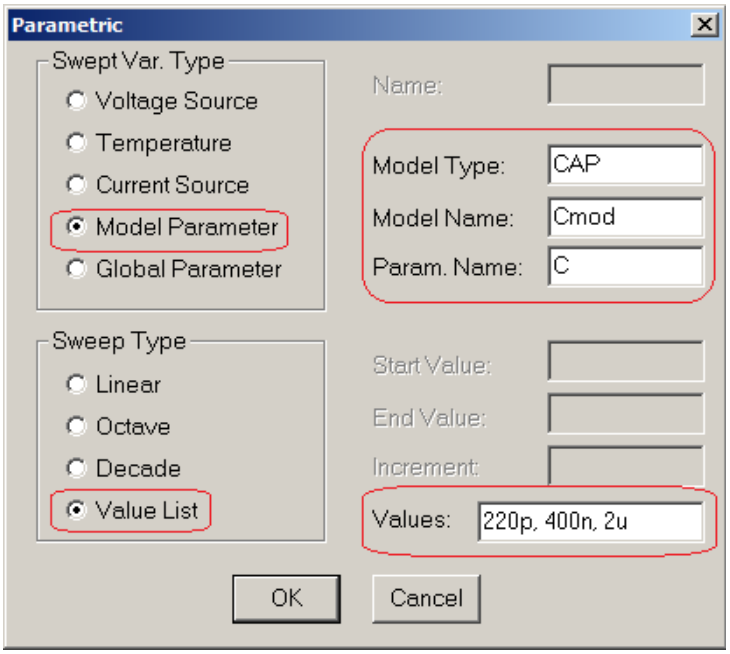
Note: For this purpose, the capacitor in the Schematics library (.slb) must have the ability to enter its **model name** and **model parameters** (this is not possible while using the standard analog.slb library where the device C is defined; I myself have defined particular devices named Cmod*), Rmod, and Lmod within my own special library made for these purposes, adding the empty attributes MODEL and MODEL_PARAMS to their definitions, editable in Schematics). The Schematics window content is shown in the picture Pic. 5 below.

*) They are **device names**, don't confuse the Cmod name here with the **model name** Cmod I have used in this example.



Pic. 5: The Schematics window content – case 2

The parameter to be swept is defined as follows, see the picture Pic. 6:



Pic. 6: Input of the parameters for the .STEP command – case 2 (model parameter C is stepped)

It generates the following lines in the appropriate circuit netlist (.net) and circuit description (.cir) files, respectively:

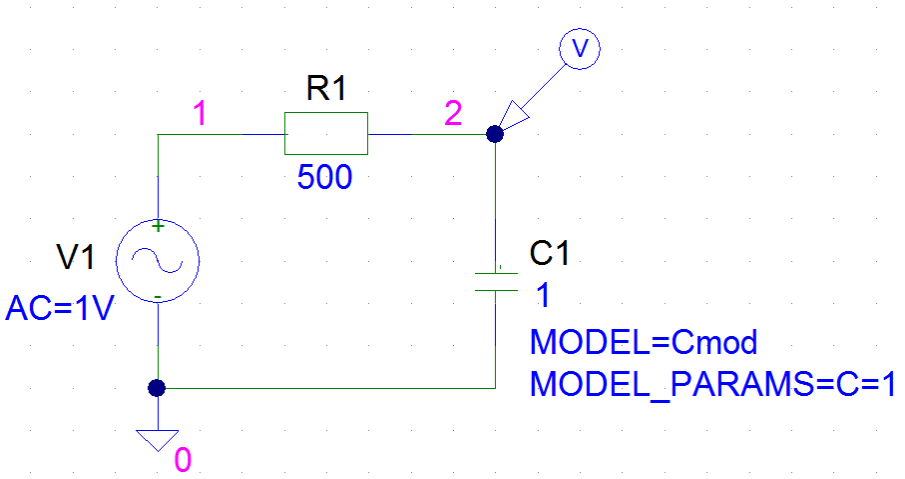
```
...
C_C1   2  0 Cmod {Cval}
.model   Cmod CAP(C={Cval})

...

** Analysis setup **
.ac DEC 101 10 100MEG
.STEP  CAP Cmod (C) LIST
+ 220p, 400n, 2u

...
```

or, in case of entering 1 for the capacitor value instead of using the parameter Cval, the netlist will contain:

```
C_C1    2 0  Cmod  1
.model   Cmod CAP (C=1)
```

The corresponding Schematics window content for this case is shown in the picture Pic. 7.

The simulation results are identical with those shown in the picture Pic. 4.



Pic. 7: The variant with „direct value“ of the capacitor C1, instead of using the parameter Cval