Wavenology SQIF Demo

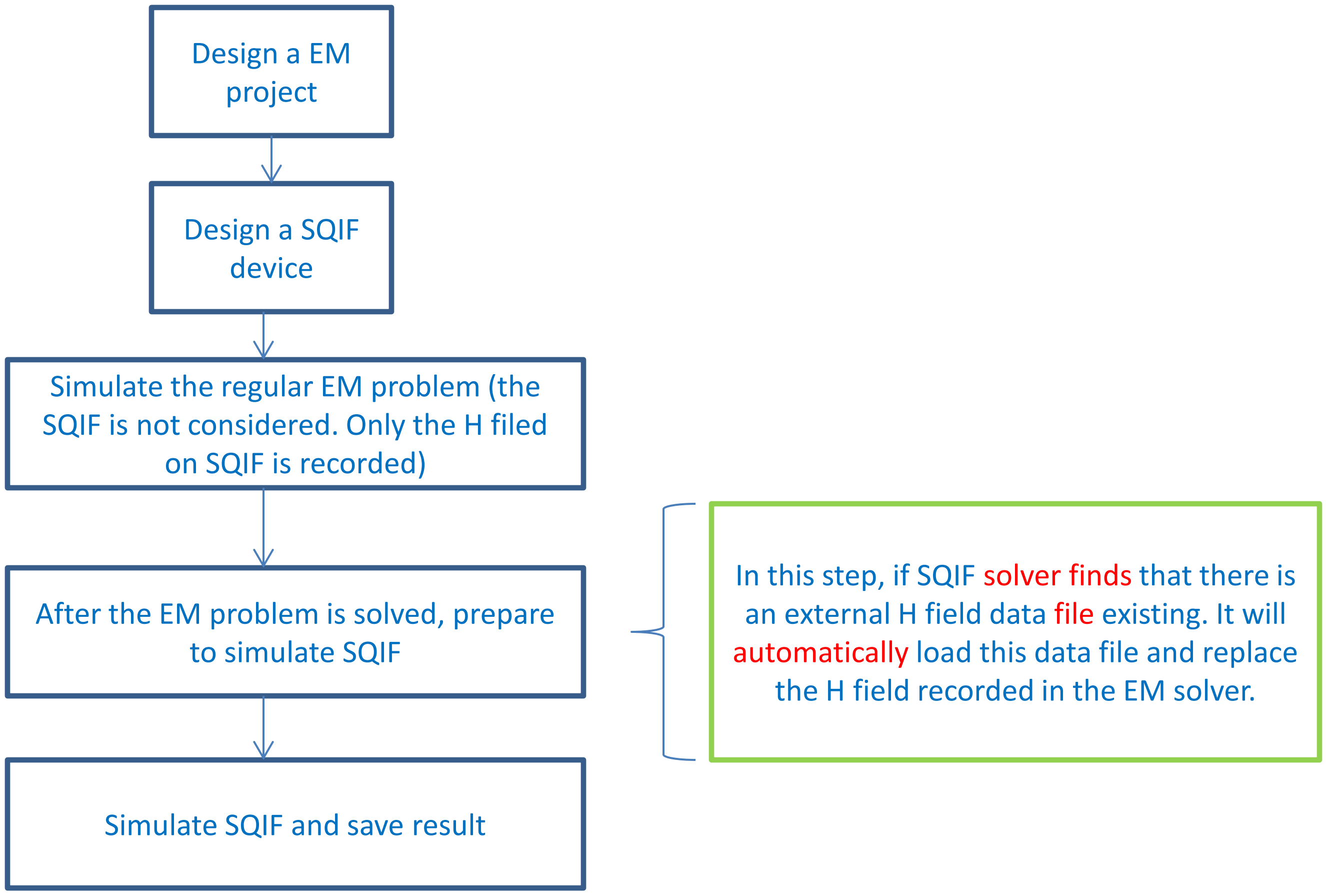
2014-07

Content

1. Demo cases
   1. Pure 1D Serial SQIF with user input magnetic flux
      - Show how to simulate a 1D serial SQIF and how to obtain the DC voltage response of a SQIF
   2. Pure 1D Parallel SQIF with user input magnetic flux
      - Show how to simulate a 1D parallel SQIF and how to obtain the DC voltage response of a SQIF
   3. Single loop dc SQUID with user input magnetic flux
      - Show how to simulate a dc SQUID and how to obtain the DC voltage response of a SQUID
   4. 1D Serial SQIF with plane wave incident
      - Show how to co-simulate SQIF and a regular EM problem
   5. 2D SQIF simulation with plane wave incident and dipole antenna
      - Show how to co-simulate SQIF with antenna and other sources

2. Procedure of building a SQIF and simulation problem
   - The example is “2D SQIF simulation with plane wave incident and dipole antenna”

Basic SQIF simulation scheme



Note: the external H field file must be in the same folder of project file, with a ***fixed name*** "SQIF_input.txt".

This means that, if user want to use external H field file, each project should have it's own folder. There cannot be multiple "SQIF_input.txt" existing for different projects in one folder, except all projects will use the same H field.

Example of the External H Field File

- Case file: SQIF\demo\1D SQIF\serial\dc_voltage_response\40-loop-gaussian-s\ SQIF_input.txt

```
1
0        0
10e-12   -41.18e0
1e-9     -41.18e0
61e-9    41.18e0
```

2 columns data

File format type is 1, means the data in this file is H field. All loops in the array will use the same H field.
The data will be represented by 2 columns. The 1st column is time in second. The 2nd column is the H field magnitude in A/m.

Here, we can see the H field is in the range of +/-41.18 A/m. Therefore, the equivalent magnetic flux quantum through a 200 um^2 loop is:
$u_0HS/\Phi_0=4\pi e^{-7}*41.18*200e^{-12}/2.0678e^{-15}=5$
As the flux plot in the case 1.

Simulation Time Step Control

There are two kinds of SQUID simulation setup. One is using text format Spice circuit. Another is using WCT graphic circuit editor (GUI).

- **Text format Spice circuit**: user can control the time step directly by Spice instruction, which is in the line ".tran xxxx", for example, ".tran 2p 61ns 0s 2p uic". Here, the 2p in red is the maximum delta time (Δt) in the SQIF simulation. More detail information please refer to the Spice manual in the following link:

  http://newton.ex.ac.uk/teaching/CDHW/Electronics2/userguide/sec4.html

  subsection "4.3.9 .TRAN: Transient Analysis"

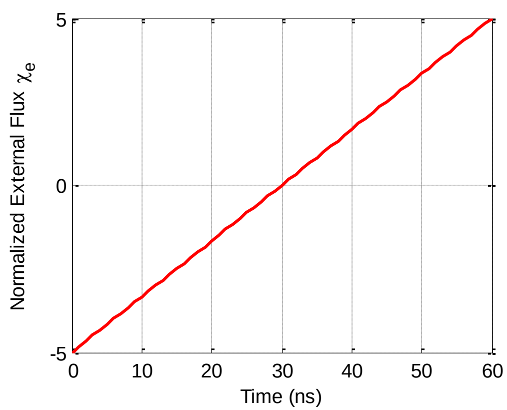
- For **GUI type** of simulation project, currently, there is not place to let user input time control in the circuit. So, we use Δt of the project automatically. The setting page is in the ***Time*** Tab of project design, as shown in the right-bottom figure of Page 36. In general, full wave EM solver will use a much smaller Δt compared to the Δt requirement in a SQIF simulation.

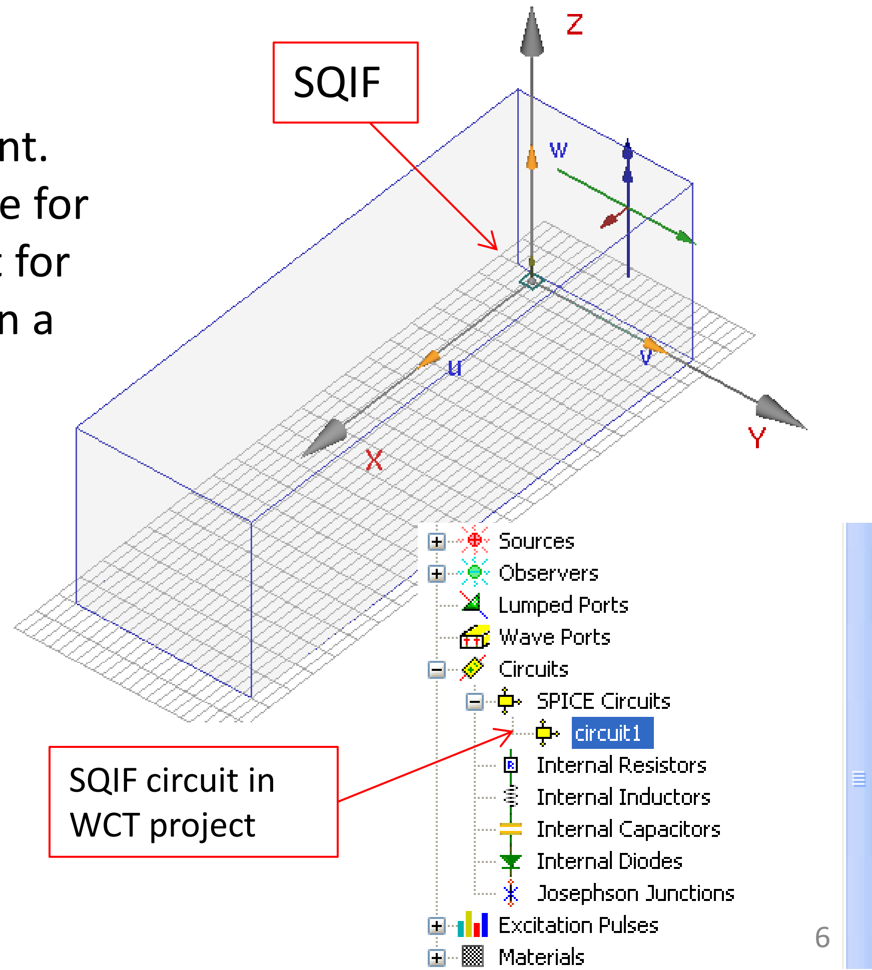
1. It is suggested to use "Text format Spice circuit" to simulate SQUID/SQIF.
2. According to our experience, it is suggested that the Δt in SQIF simulation will not larger than 6ps. A smaller Δt will have better accuracy, but cause longer simulation time. We find that, for most cases, 2p second will bring a good balance for accuracy and shorter simulation time.

Case 1: Pure 1D Serial SQIF with user input magnetic flux

- Case file: SQIF\demo\1D SQIF\serial\dc_voltage_response\40-loop-gaussian-s\s40_l1_lm.wnt
- This case is used to show how to obtain **the dc voltage response** on a 40 loops 1D serial dc SQIF

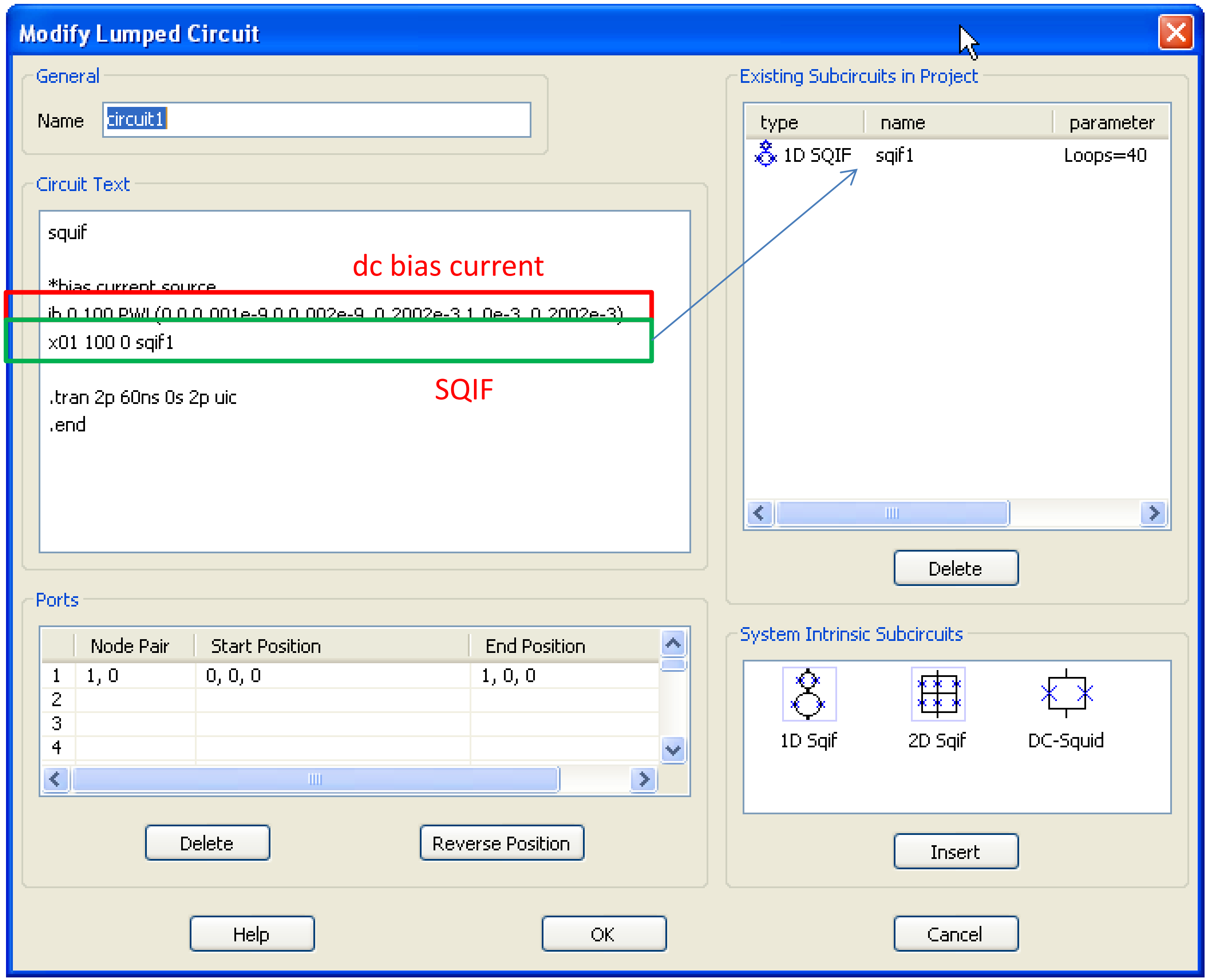
This case looks like using a plane wave incident. However, we define a external H field data file for SQIF. This data will be the actual H field input for SQIF. Following figure is the equivalent flux on a loop with 200 μm^2.



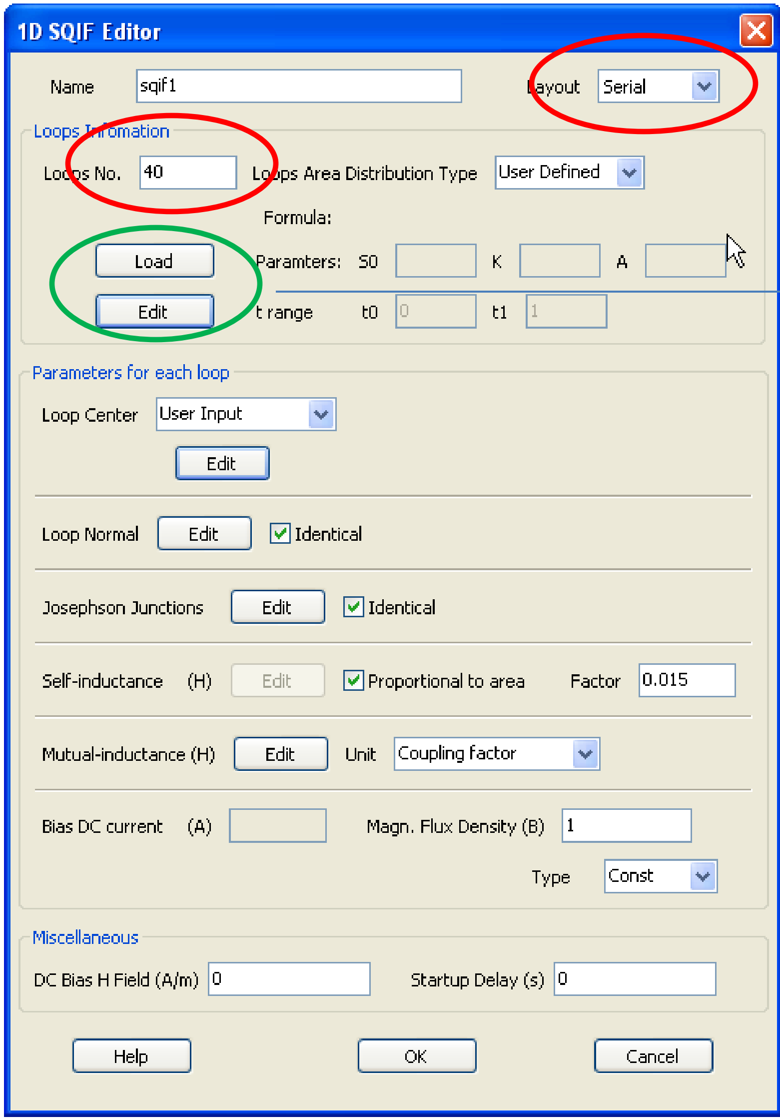


Full SQIF circuit Setting

(note: This case we use TEXT format circuit. However, WCT also support graphic format circuit, it can be also used to build a SQIF circuit)

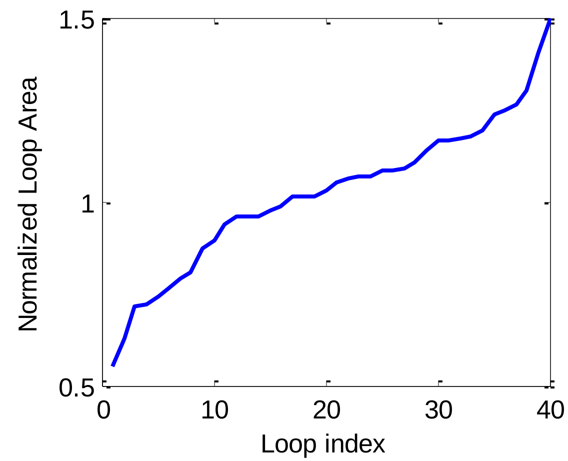


Serial SQIF parameters



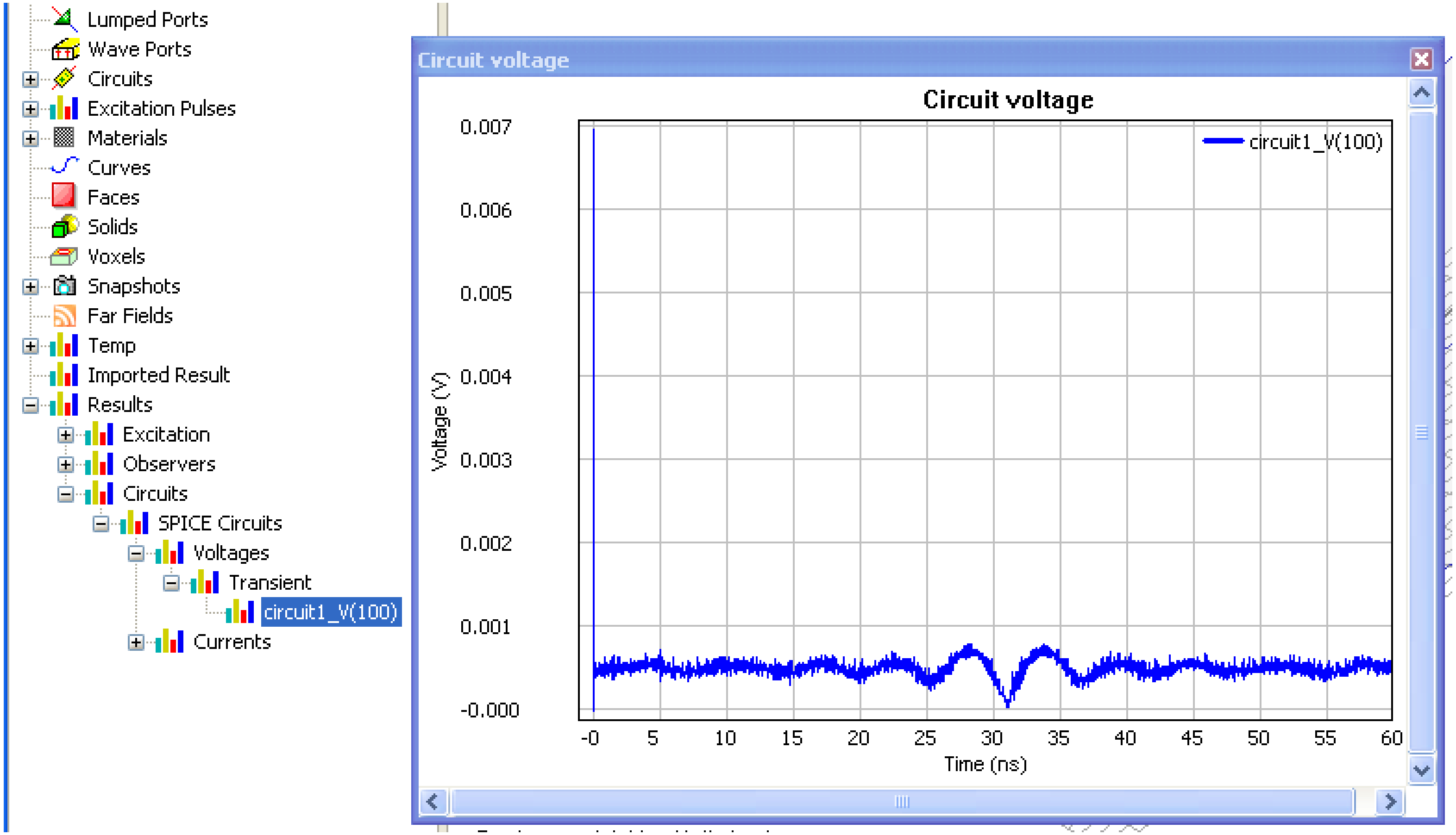
Loops area

1=200 μm^2

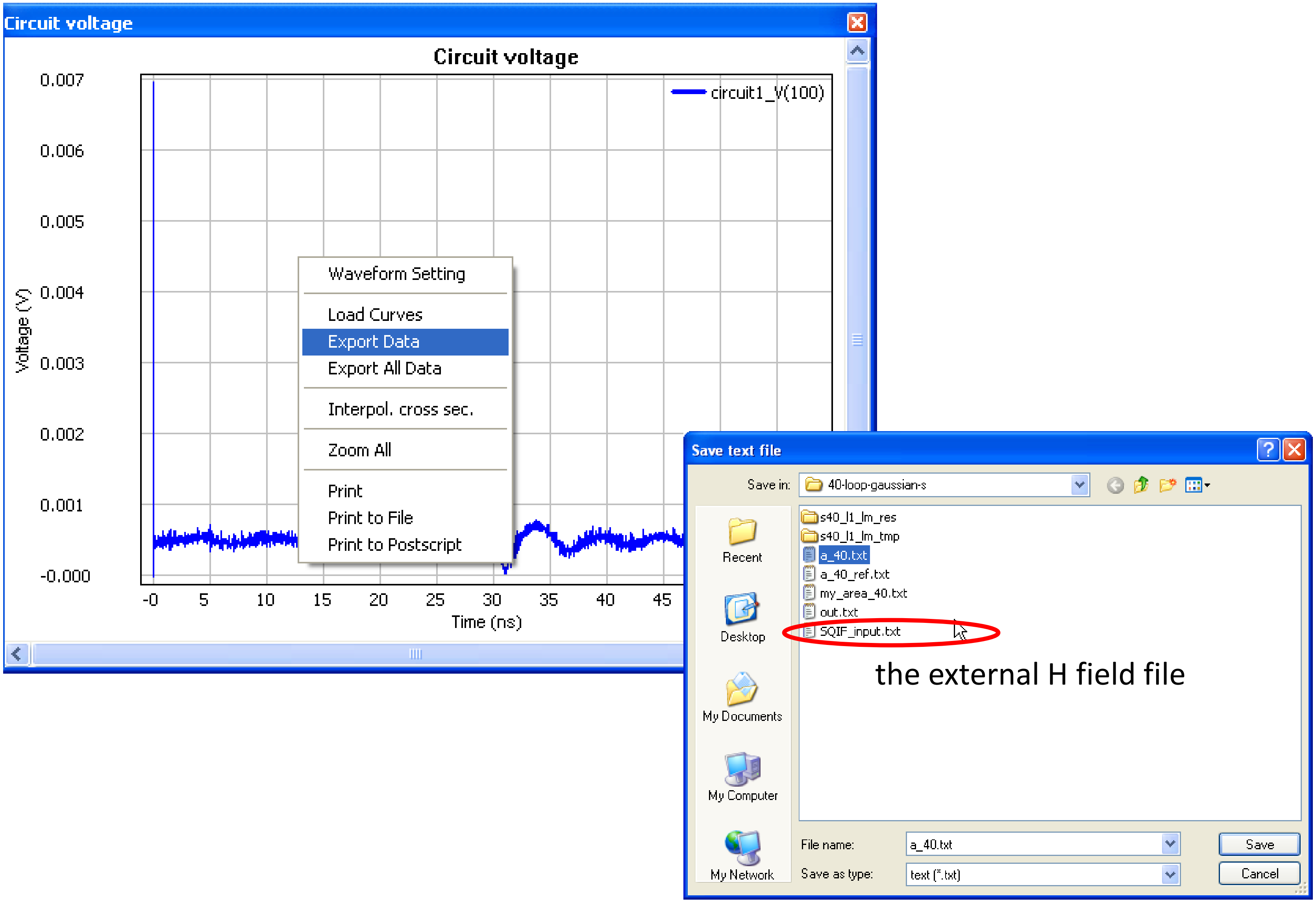


The loops size is imported from file: my_area_40.txt

Transient output voltage of SQIF



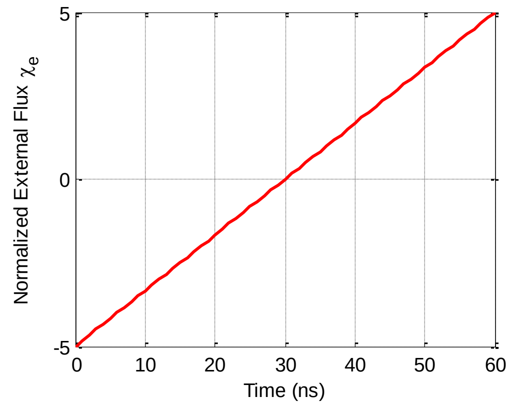
Right click mouse to export this voltage data as “a_40.txt”



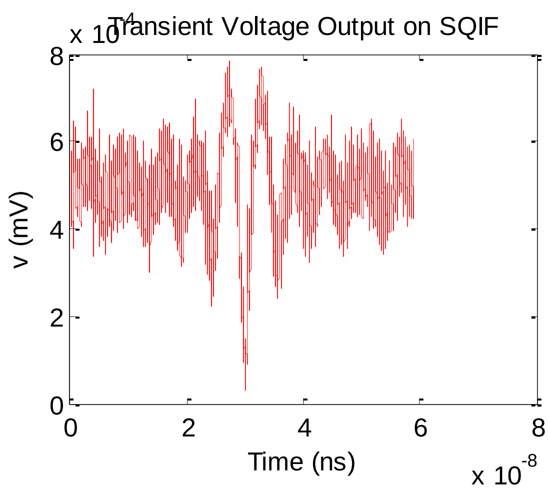
Use matlab code to filter this data to obtain the dc voltage response of SQIF.
The matlab code is under the WCT project folder as:
SQIF\demo\1D SQIF\serial\dc_voltage_response\40-loop-gaussian-s\ show_res_2.m

The processed results

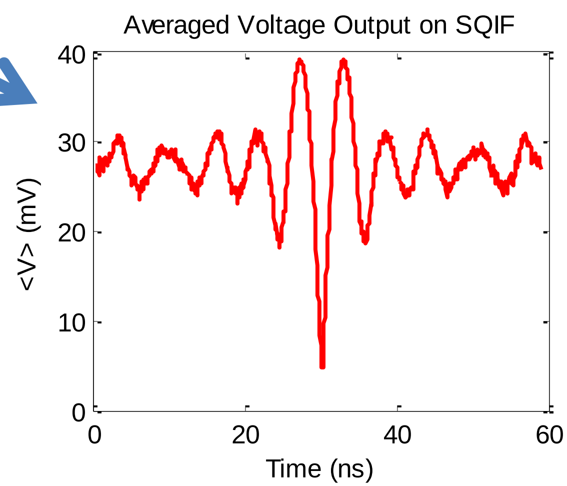
External flux input

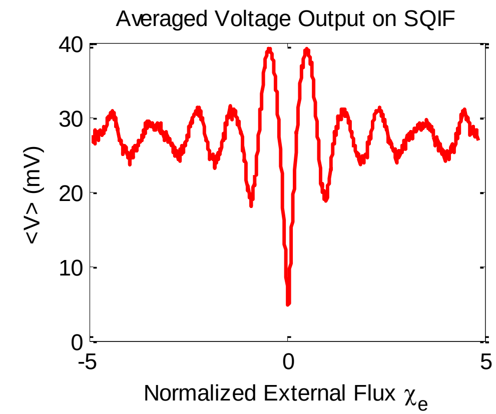


Transient SQIF output

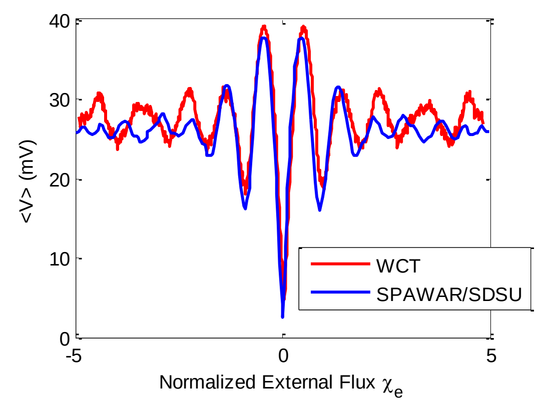


SQIF output
after filtered





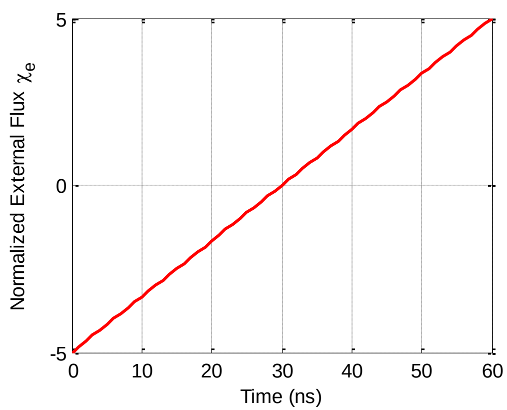
Comparison

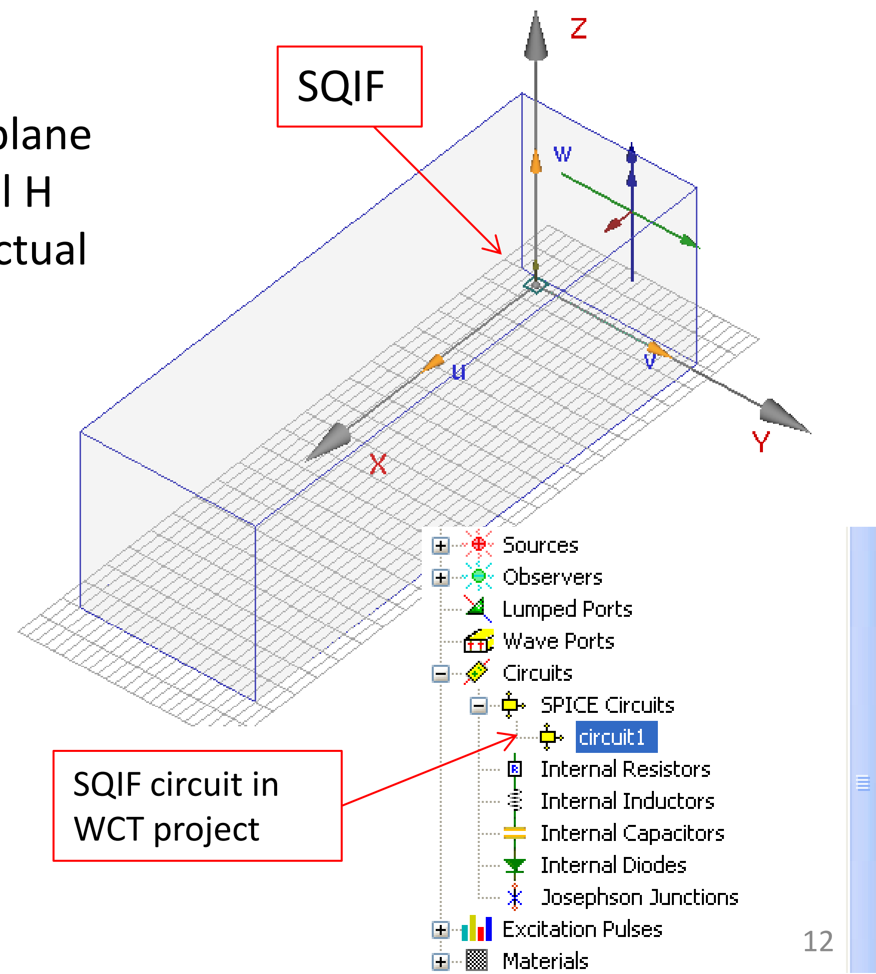


Case 2: Pure 1D Parallel SQIF with user input magnetic flux

- Case file: SQIF\demo\1D SQIF\serial\dc_voltage_response\40-loop-gaussian-p\s40_l1_lm.wnt
- This case is used to show how to obtain **the dc voltage response** on a 40 loops 1D parallel dc SQIF

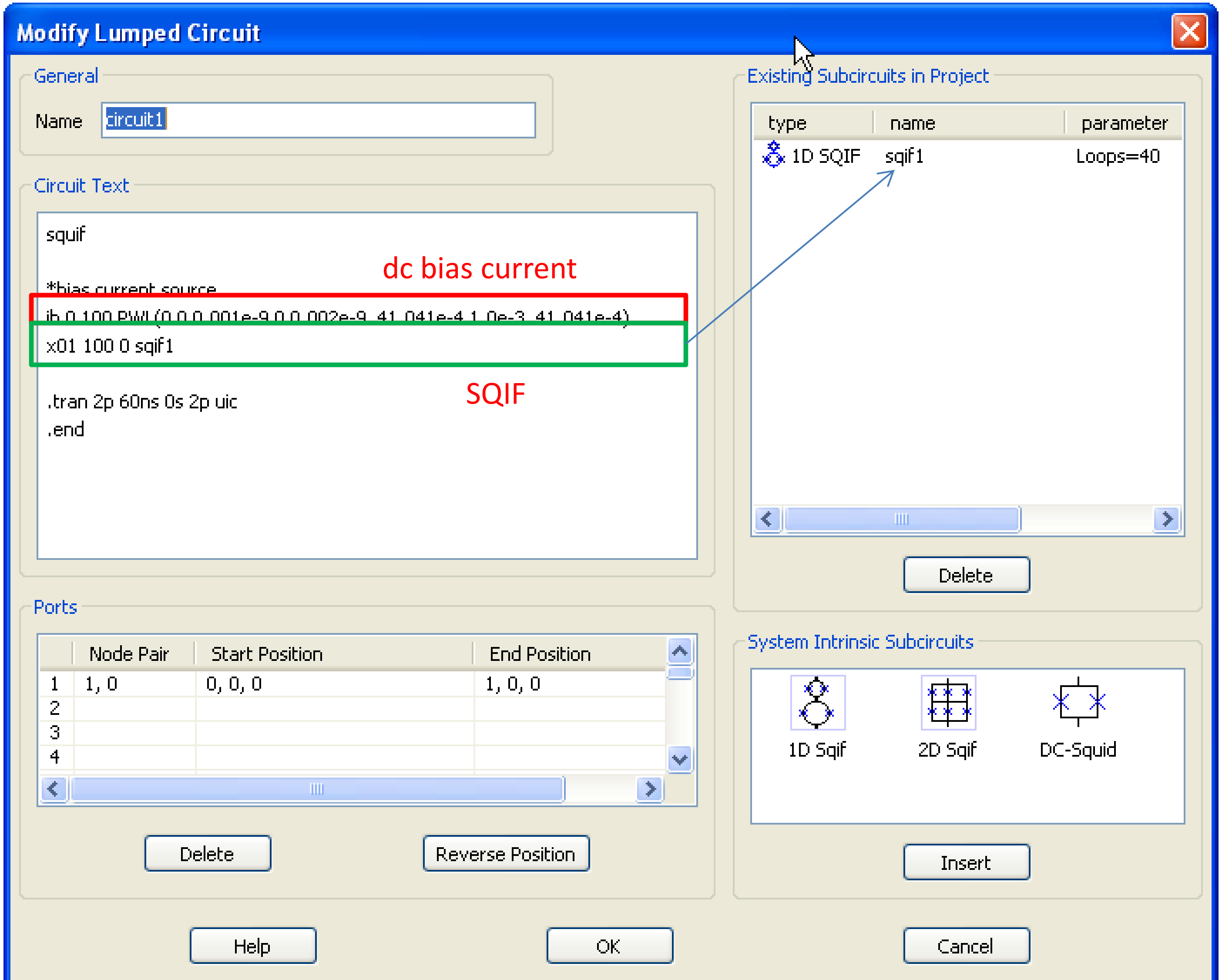
Similar to case 1, this case looks like using a plane wave incident. However, we define a external H field data file for SQIF. This data will be the actual H field input for SQIF. Following figure is the equivalent flux on a loop with 200 µm^2.



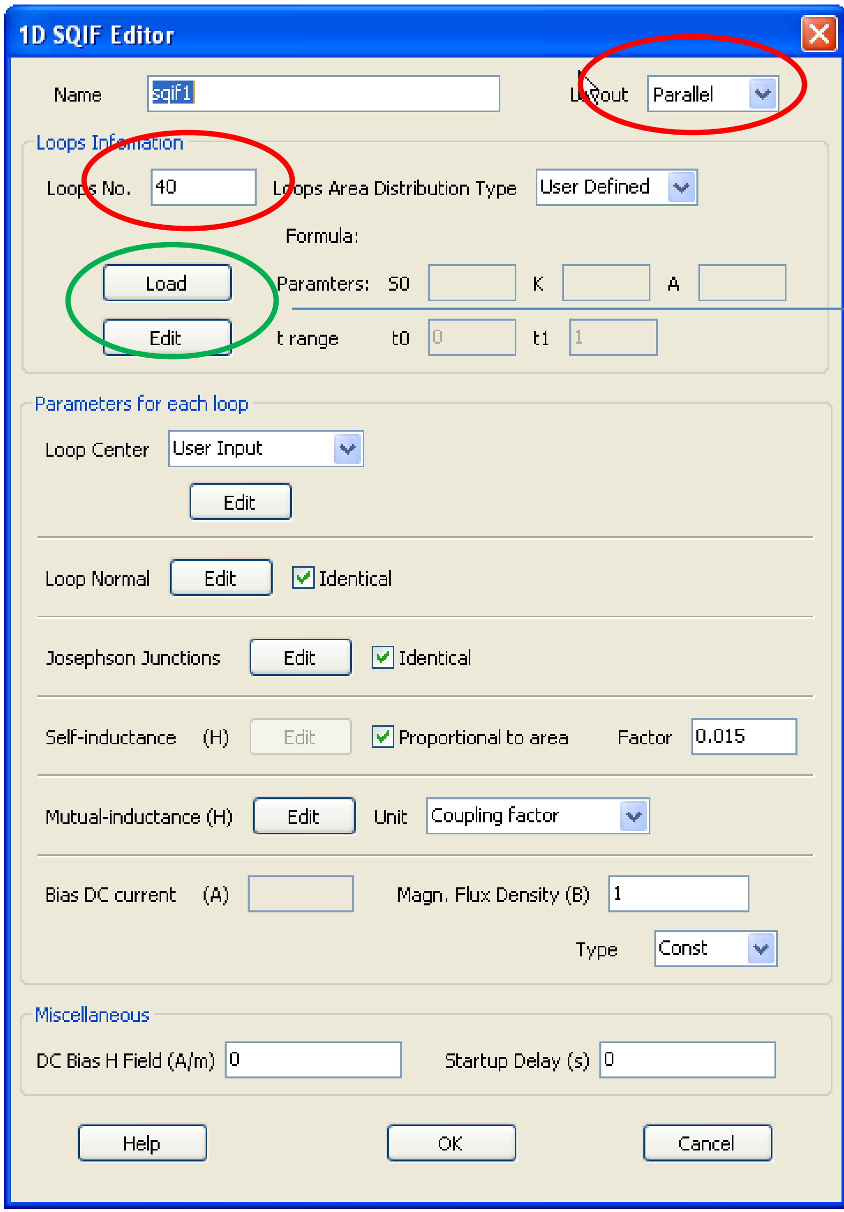


Full SQIF circuit Setting

(note: This case we use TEXT format circuit. However, WCT also support graphic format circuit, it can be also used to build a SQIF circuit)

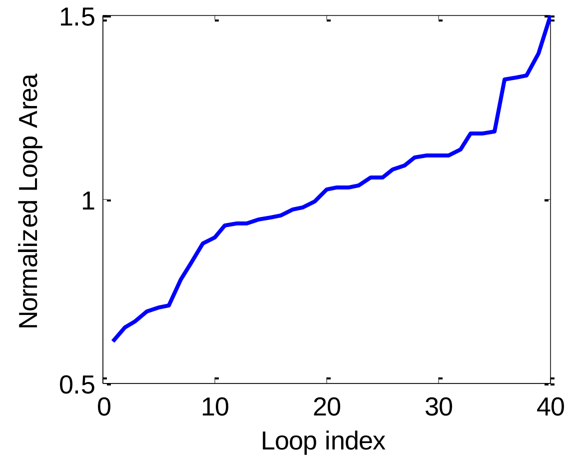


Parallel SQIF parameters



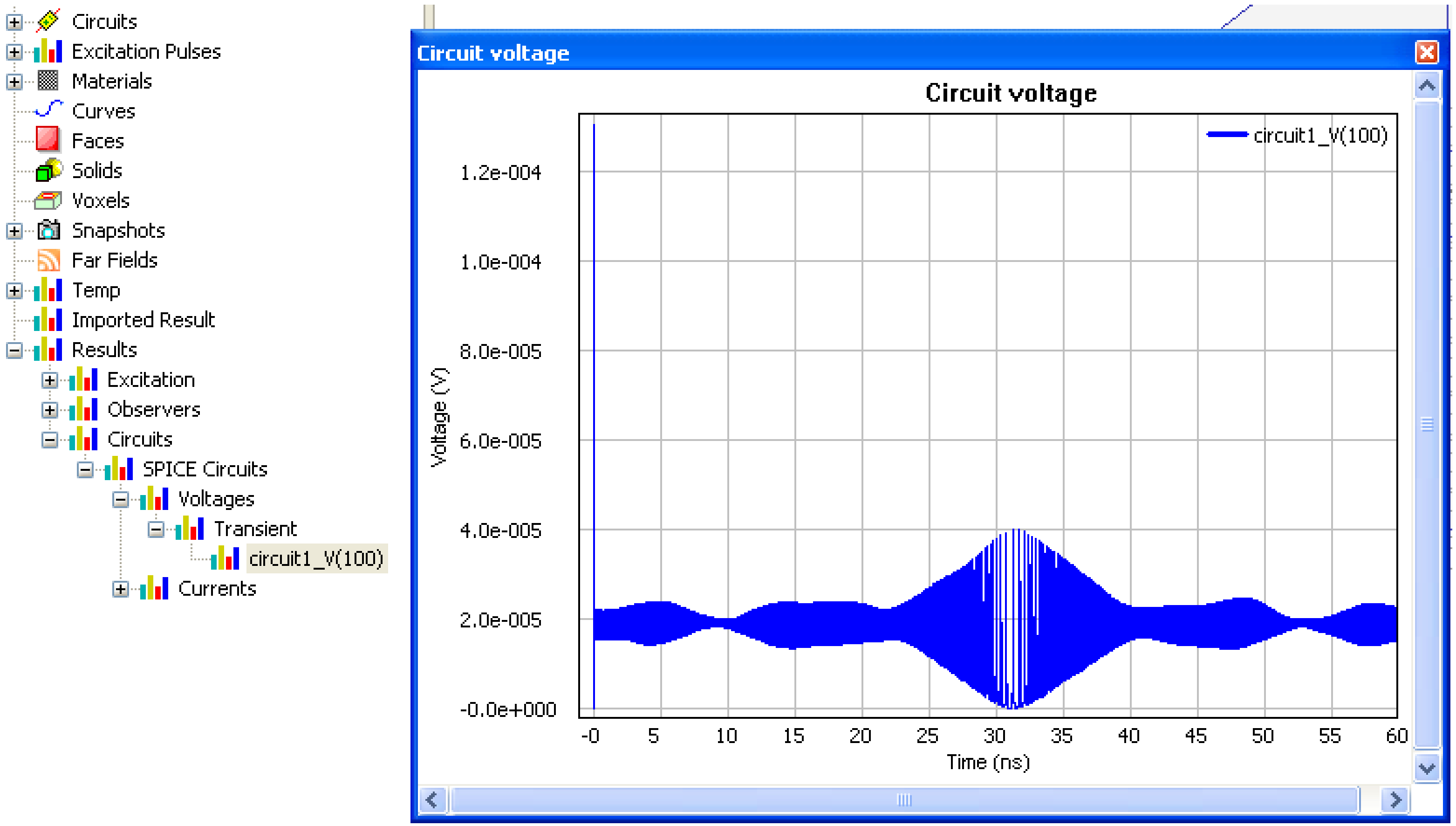
Loops area

1=200 µm²



The loops size is imported from file: my_area_40.txt

Transient output voltage of SQIF



Same as case 1, right click mouse to export this voltage data as “a_40.txt”

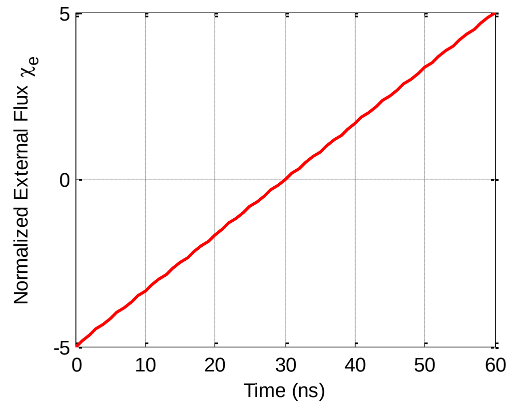
Use matlab code to filter this data to obtain the dc voltage response of SQIF.

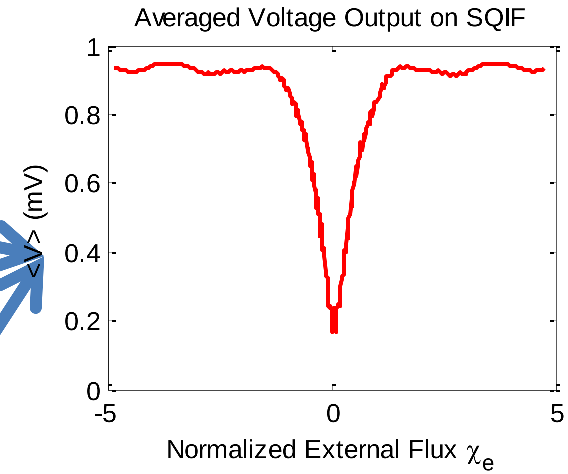
The matlab code is under the WCT project folder as:

SQIF\demo\1D SQIF\serial\dc_voltage_response\40-loop-gaussian-p\ show_res_2.m

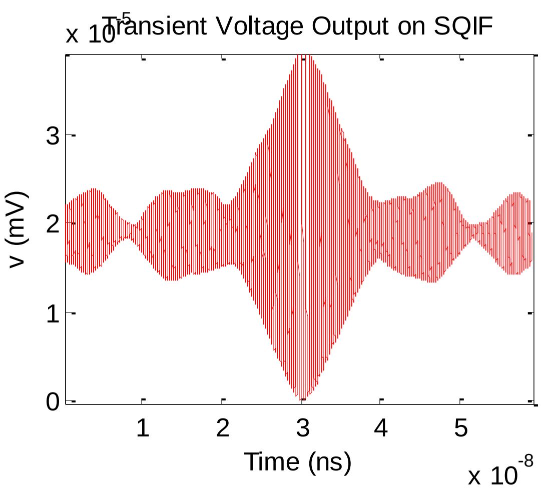
The processed results

External flux input

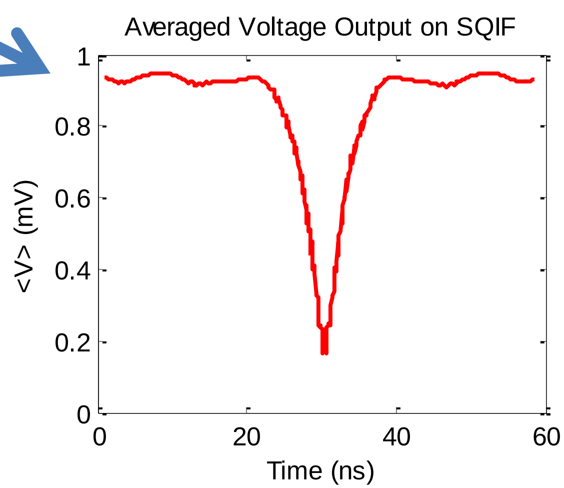




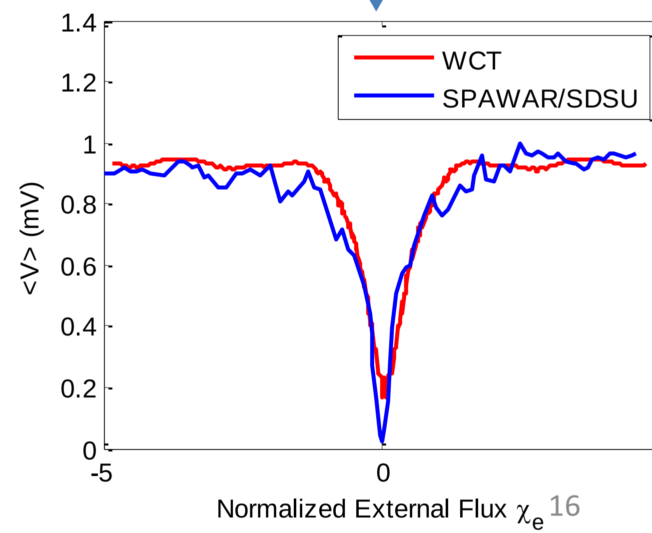
Transient SQIF output



SQIF output after filtered



Comparison

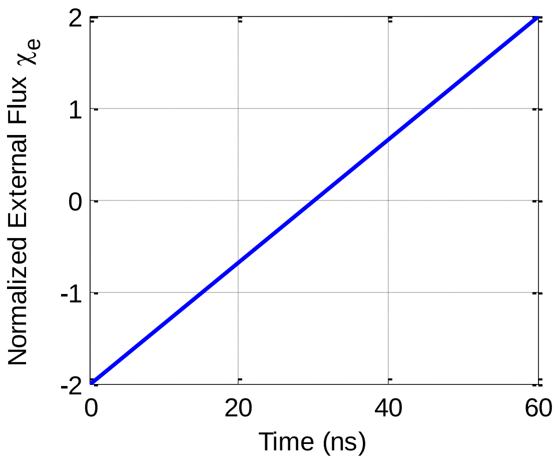


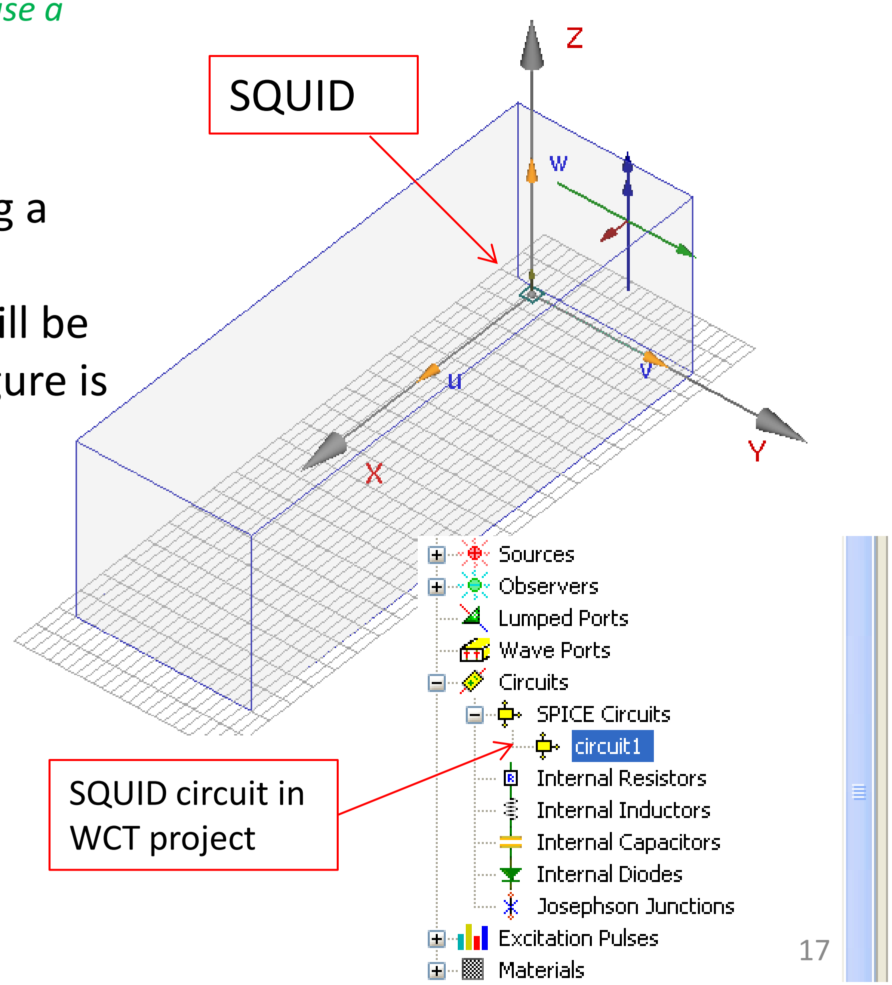
Case 3: Single loop dc SQUID with user input magnetic flux

- Case file: SQIF\demo\1D SQIF\single loop\dc_voltage_response\s1_2.wnt
- This case is used to show how to obtain **the dc voltage response** on a single loops dc SQUID

Due to a DC SQUID is a single loop SQIF. In this case, we use a single loop serial SQIF to represent a dc SQUID.

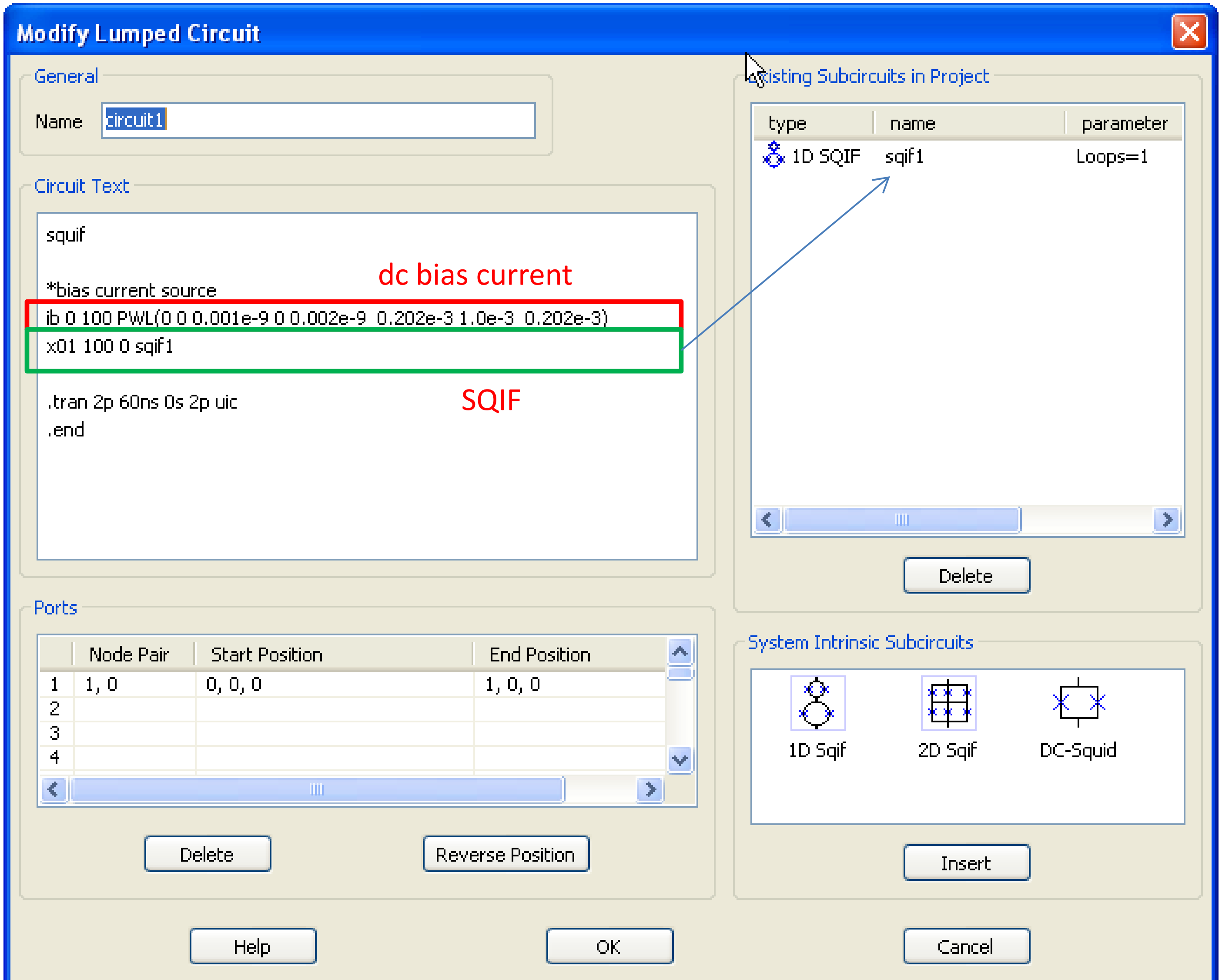
Similar to case 1 & 2, this case looks like using a plane wave incident. However, we define a external H field data file for SQIF. This data will be the actual H field input for SQIF. Following figure is the equivalent flux on a loop with 200 μm^2.



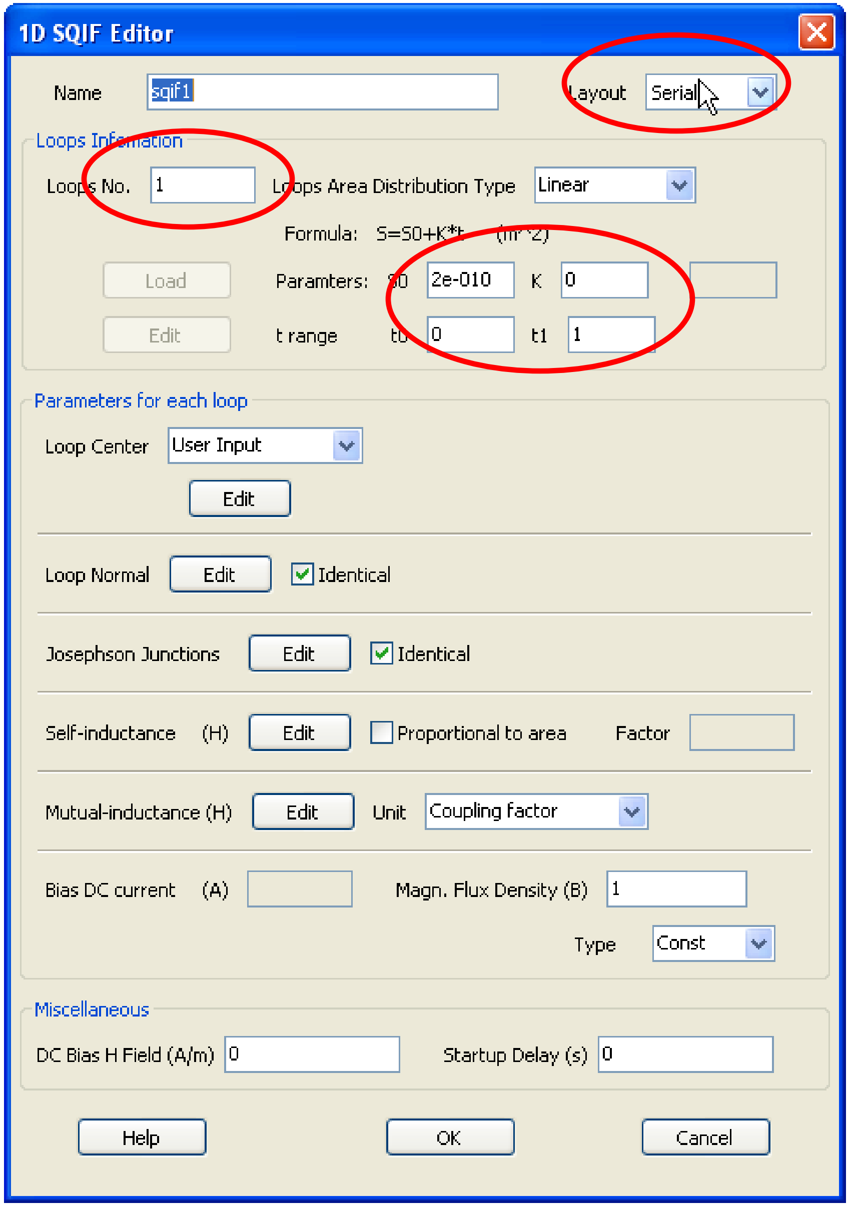


Full SQIF circuit Setting

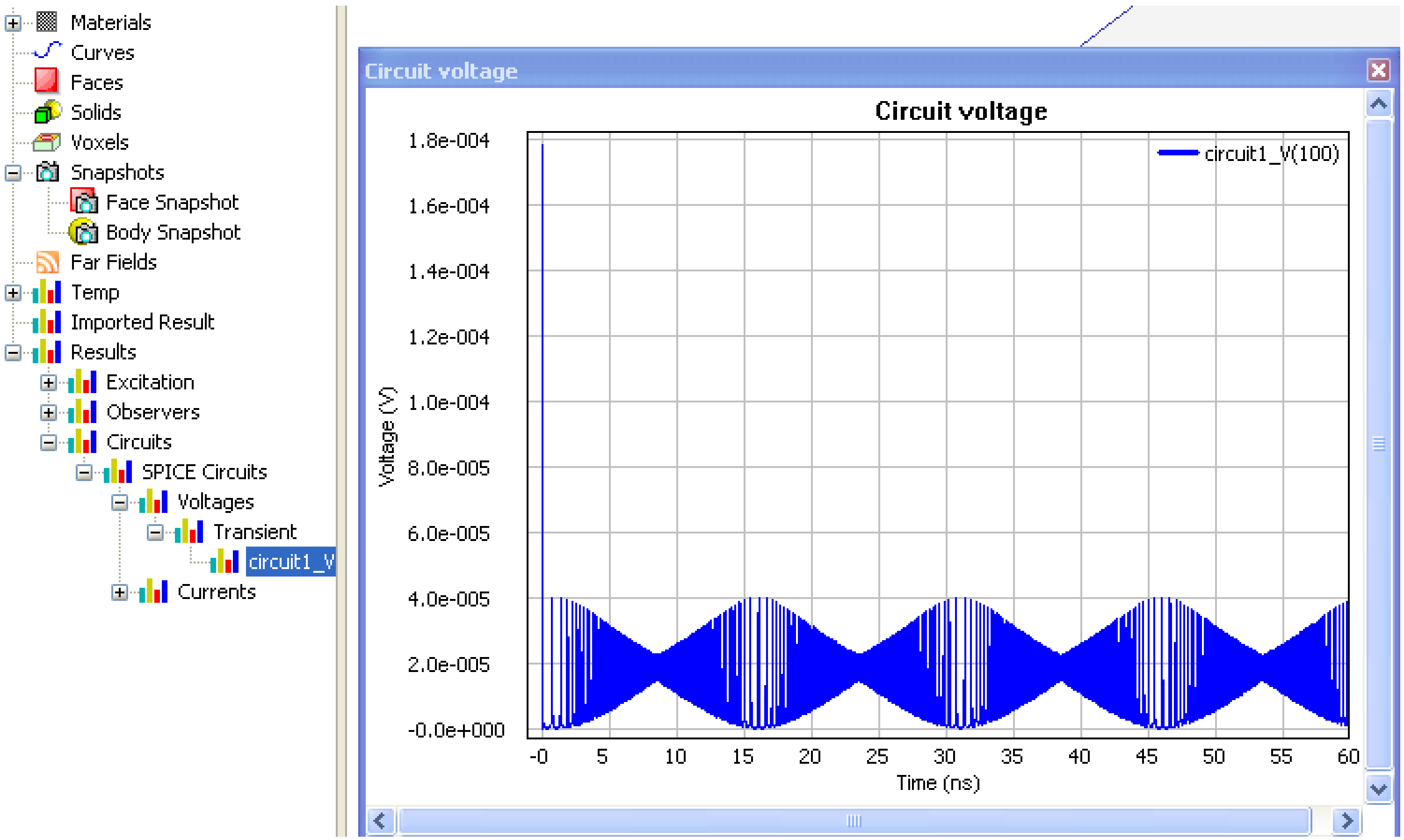
(note: This case we use TEXT format circuit. However, WCT also support graphic format circuit, it can be also used to build a SQIF circuit)



Parallel SQIF parameters



Transient output voltage of SQIF

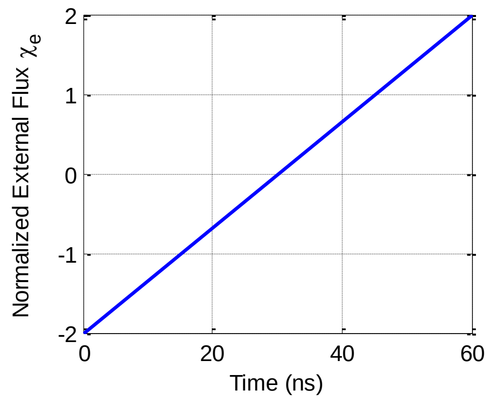


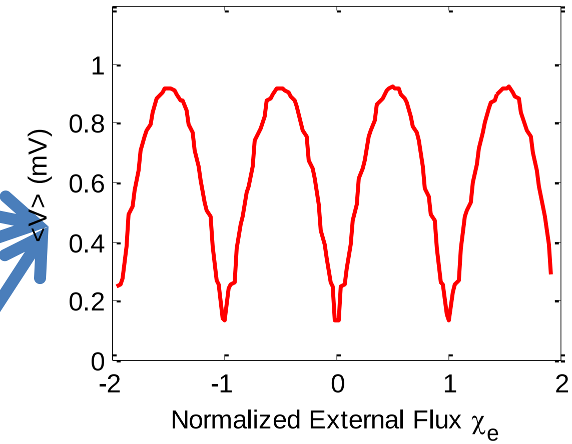
Same as case 1, right click mouse to export this voltage data as “a_1.txt”

Use matlab code to filter this data to obtain the dc voltage response of SQIF.
The matlab code is under the WCT project folder as:
SQIF\demo\1D SQIF\single loop\dc_voltage_response\show_res_2.m

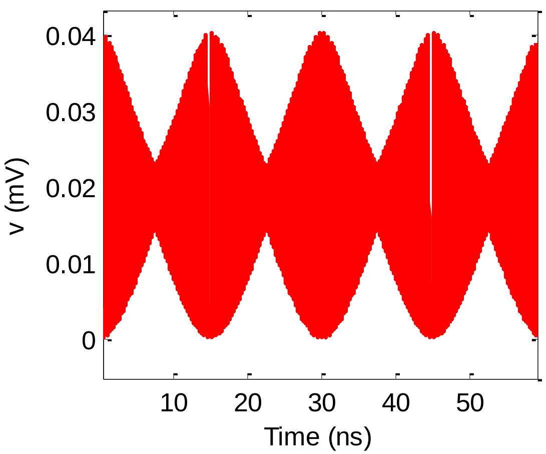
The processed results

External flux input

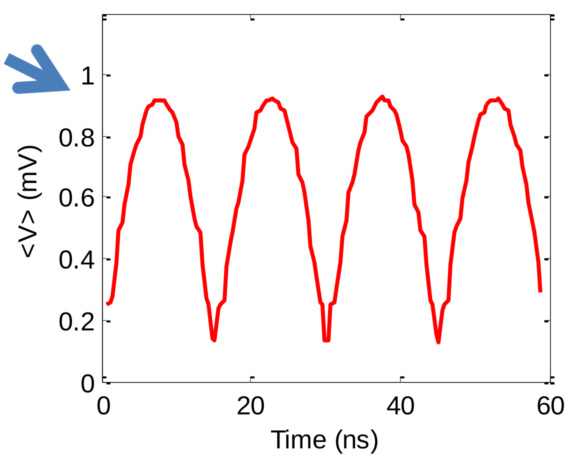




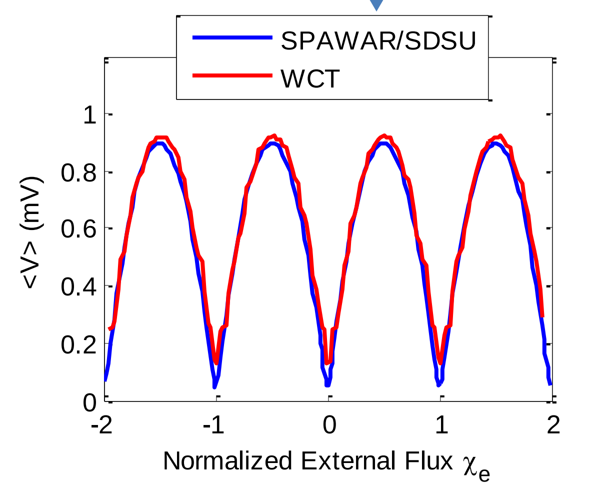
Transient SQIF output



SQIF output
after filtered

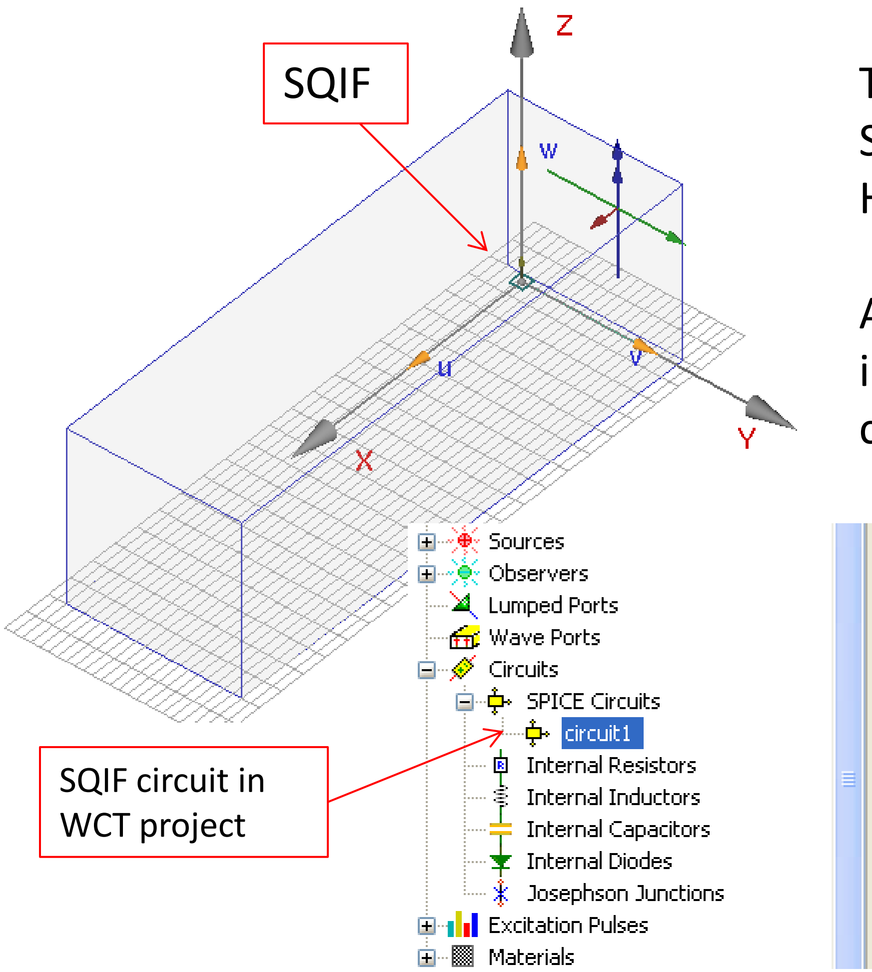


Comparison



Case 4: 1D Serial SQIF with plane wave incident

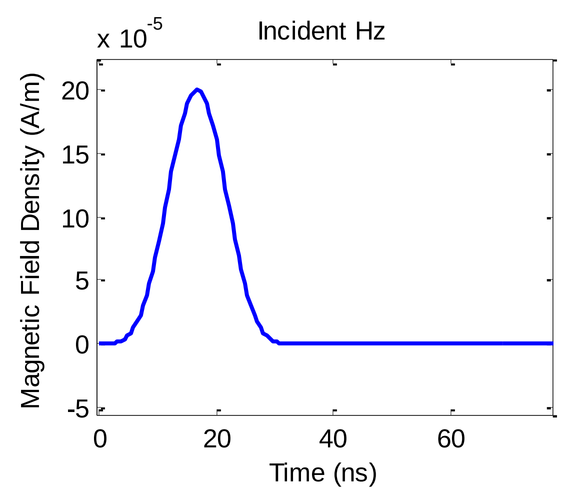
- Case file: SQIF\demo\1D SQIF\serial\plane_wave_incident\10_loops\case2_10loop.wnt
- This case is used to show how to use a 10 loops 1D serial dc SQIF to receive a incoming plane wave



This case use a plane wave incident. The SQIF device will convert received transient H field and export a transient voltage.

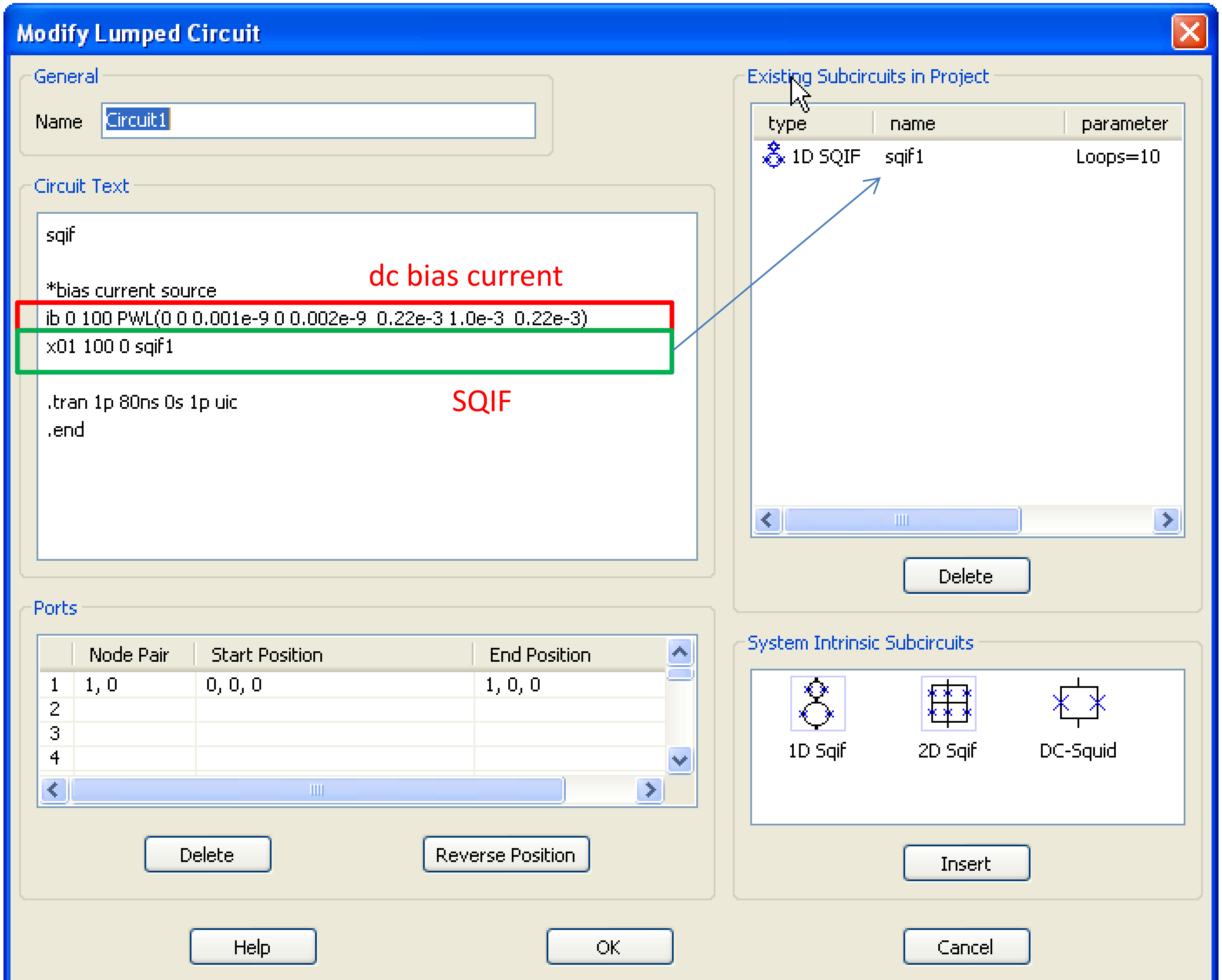
After filtered by a low-pass filter, the incoming wave can be re-constructed correctly.

Incoming plane wave

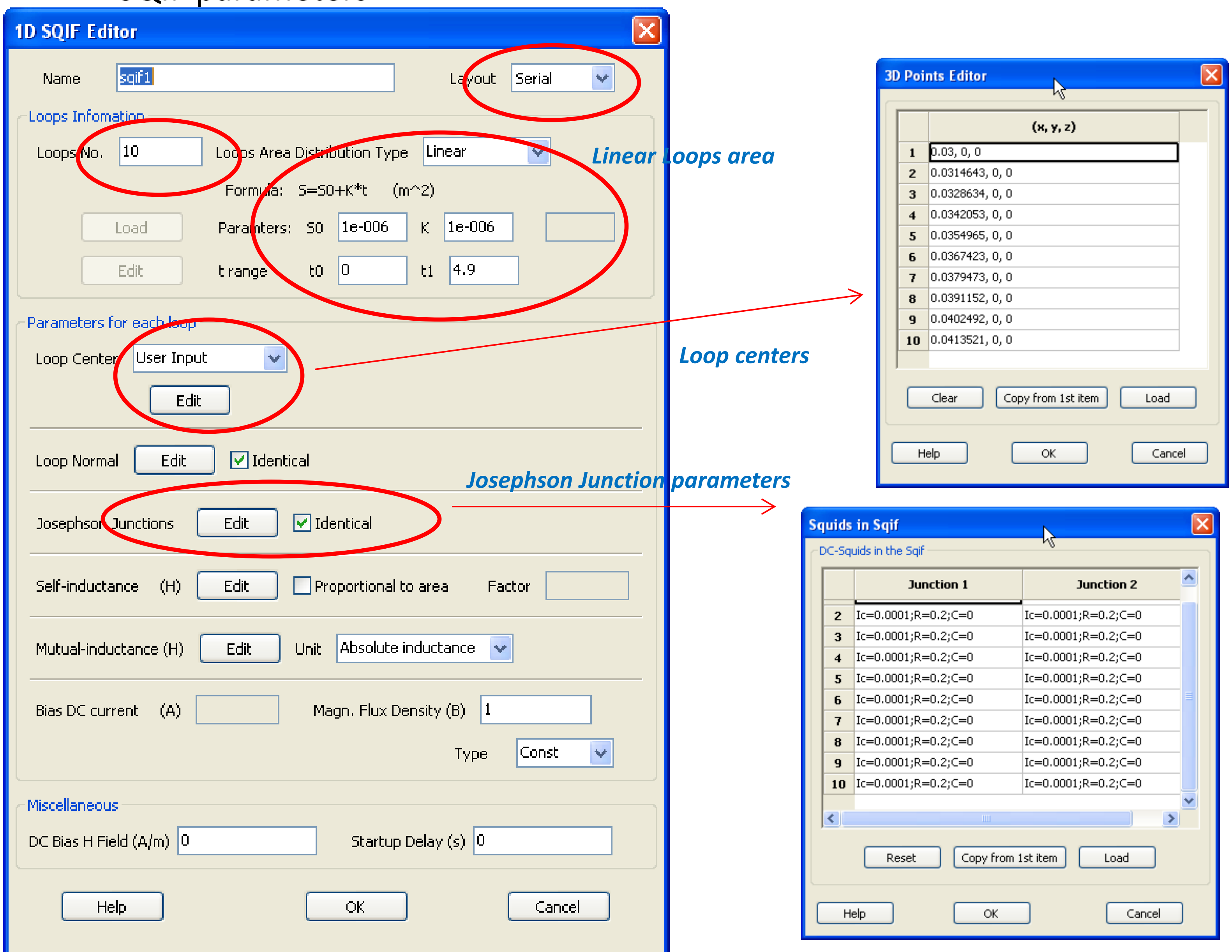


Full SQIF circuit Setting

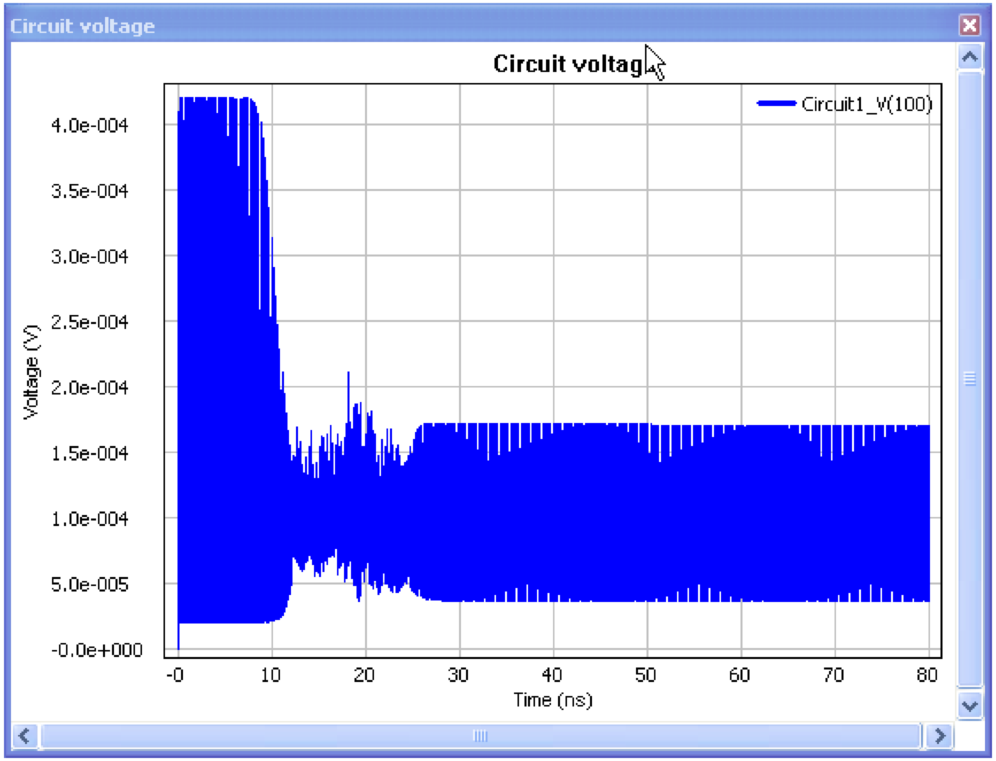
(note: This case we use TEXT format circuit. However, WCT also support graphic format circuit, it can be also used to build a SQIF circuit)



SQIF parameters



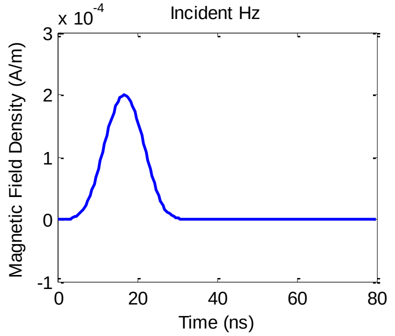
Transient output voltage of SQIF



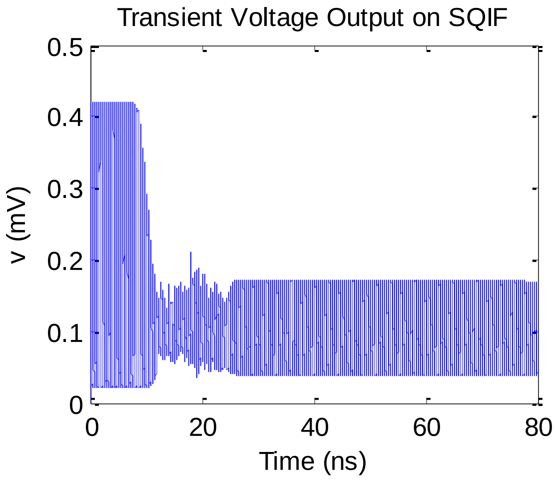
Same as case 1, right click mouse to export this voltage data as "v_out.txt"

Use matlab code to filter this data to reconstruct the received signal.
The matlab code is under the WCT project folder as:
SQIF\ demo\1D SQIF\serial\plane_wave_incident\10_loops\show_res_2.m

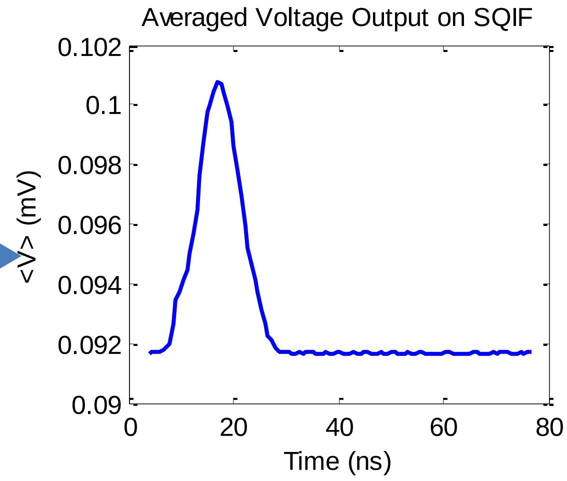
Incoming
plane wave



Transient SQIF output

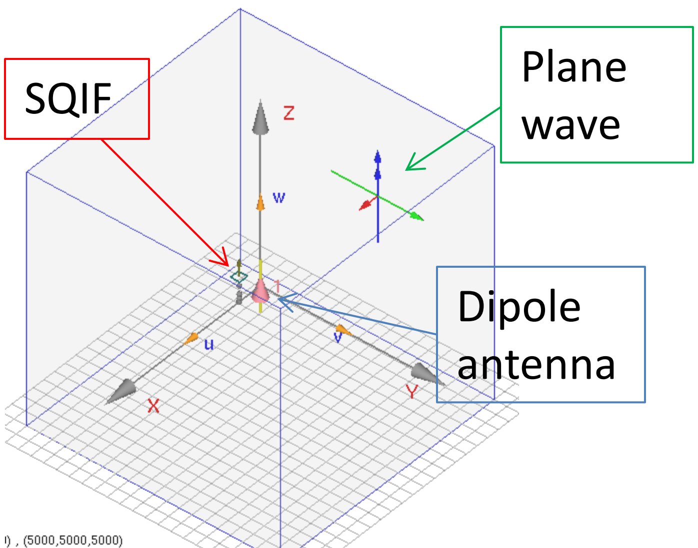


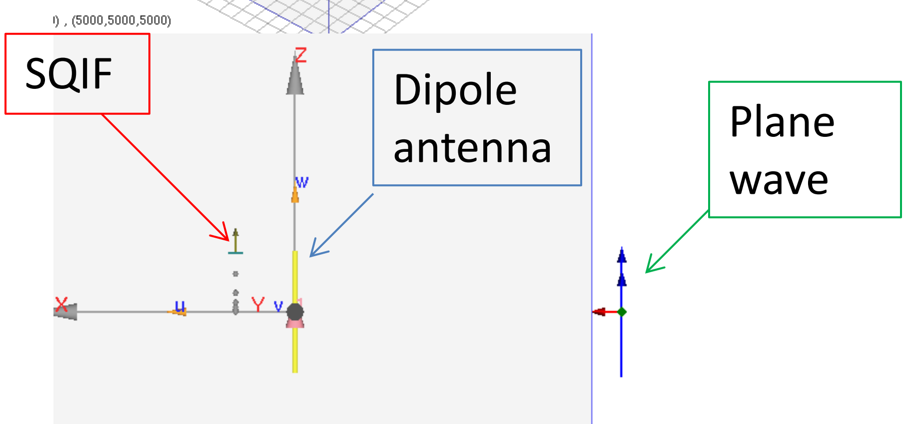
SQIF output after filtered



Case 5: 2D SQIF simulation with plane wave incident and dipole antenna

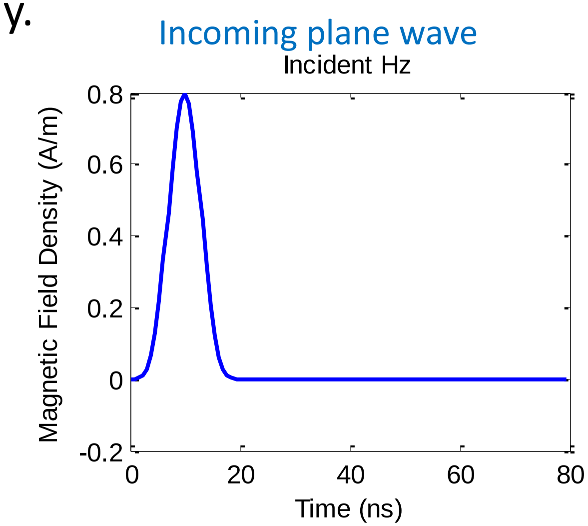
- Case file: SQIF\demo\2D SQIF\SQIF+PW+Dipole\system_sqif_10x10.wnt
- This case is used to show how to use a 10x10 loops 2D dc SQIF to pickup a incoming plane wave from the strong interference from nearby dipole antenna
- note:
- 1. the simulation time for this case is about 3 minutes on x32 version
- 2. the 2D SQIF array has not been verified with experimental data yet





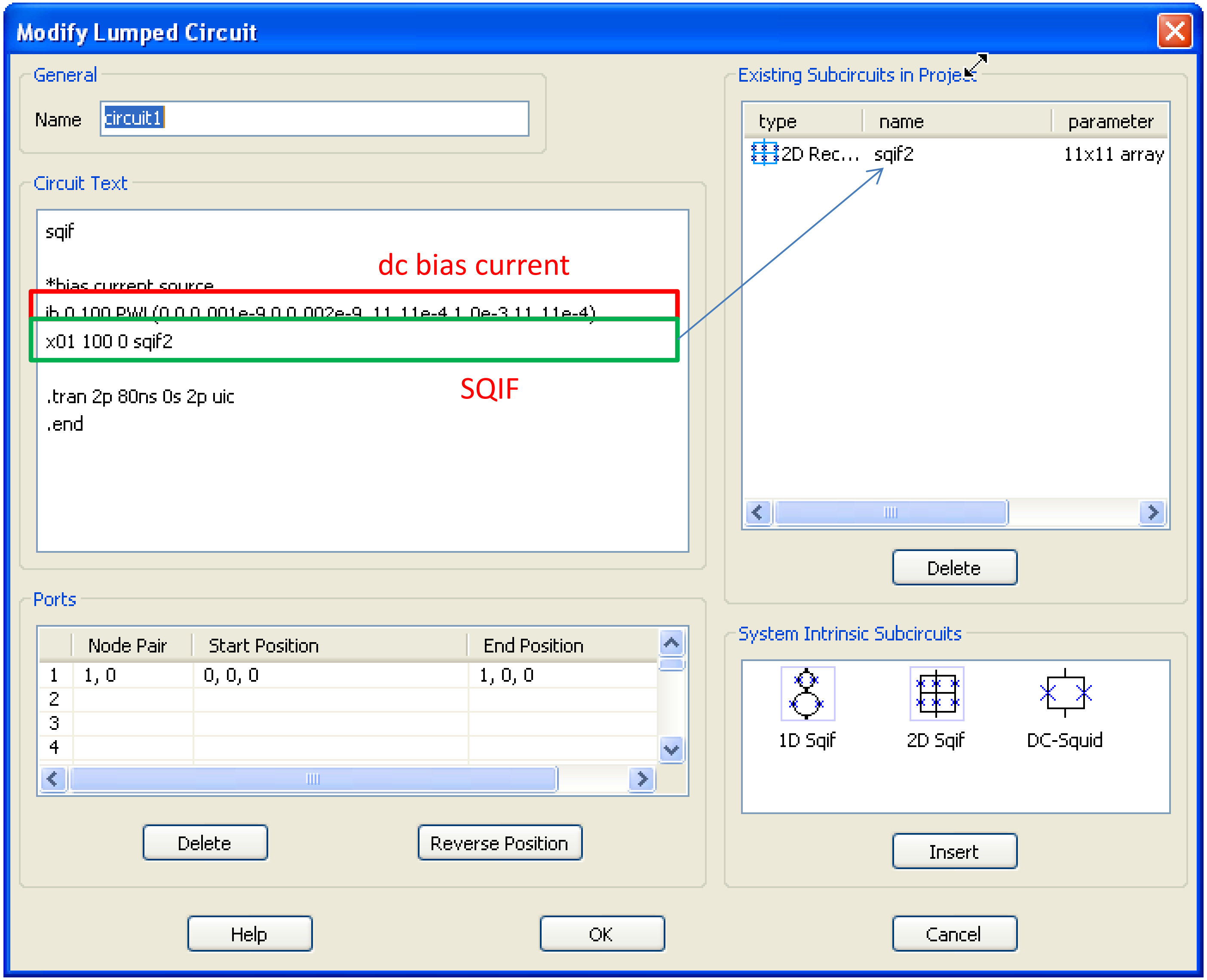
This case has two signal source: an incident plane wave and a dipole antenna. The SQIF device will receive the transient H field from the plane wave only.

After filtered by a low-pass filter, the incoming wave can be re-constructed correctly.

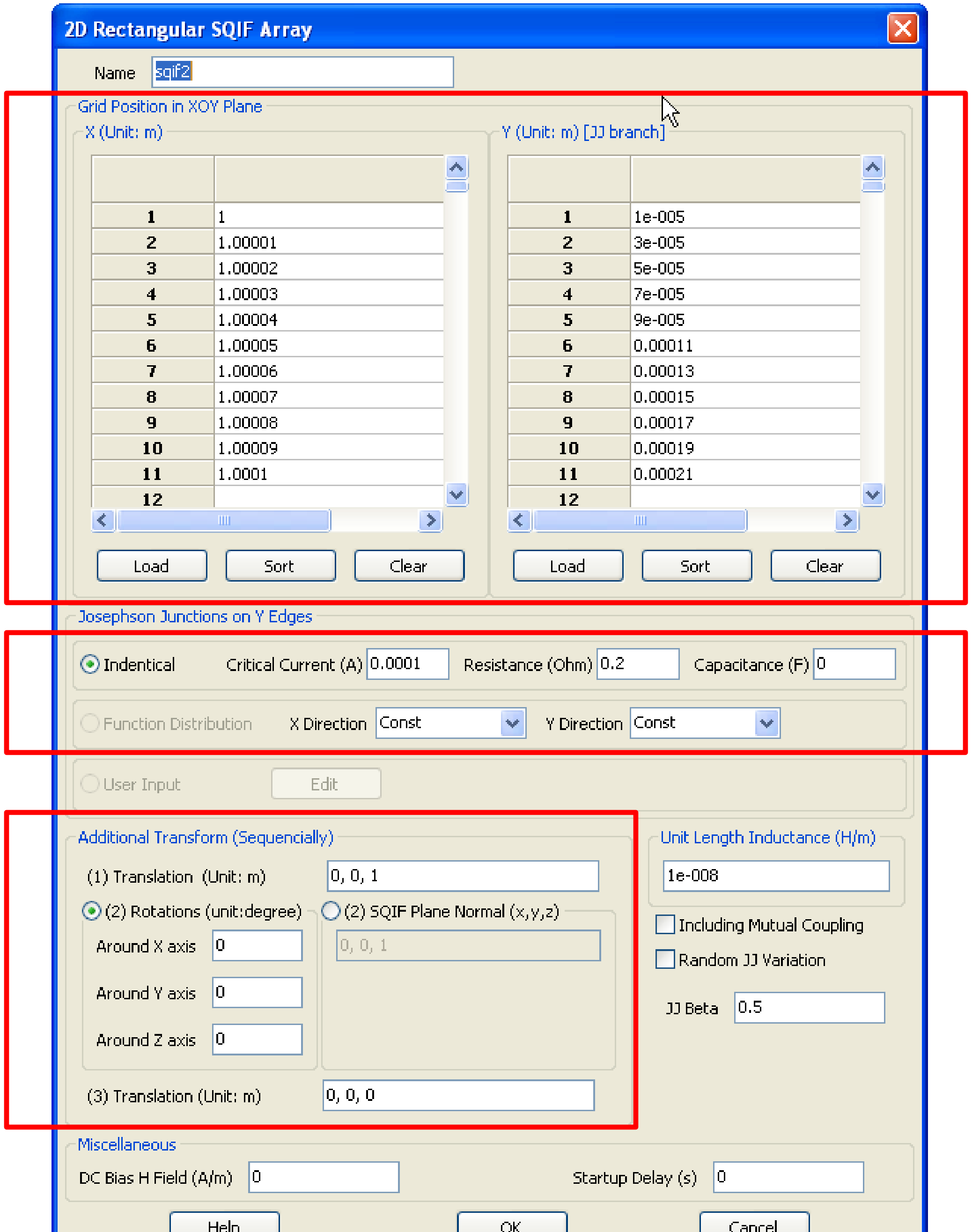


Full SQIF circuit Setting

(note: This case we use TEXT format circuit. However, WCT also support graphic format circuit, it can be also used to build a SQIF circuit)



SQIF parameters



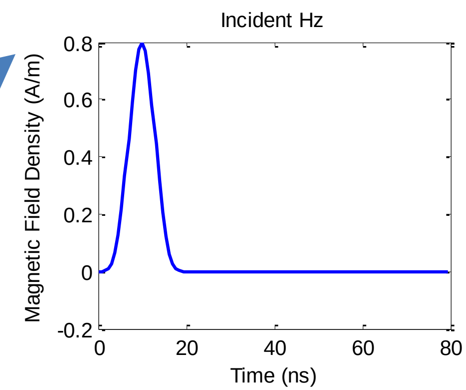
Loops grid

Josephson Junction parameters

3D position translation of SQIF

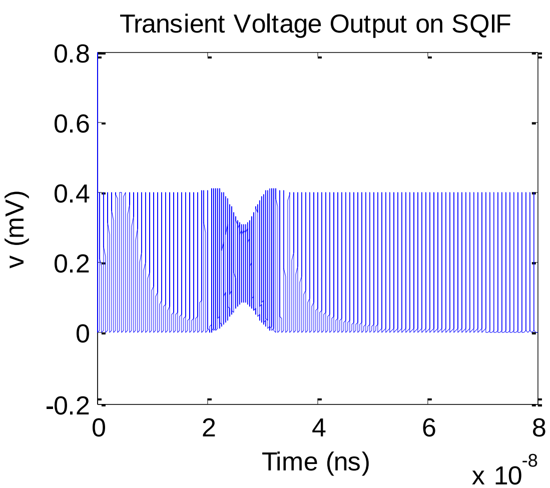
Use matlab code to filter this data to reconstruct the received signal.
The matlab code is under the WCT project folder as:
SQIF\ demo\ 2D SQIF\SQIF+PW+Dipole \show_res_2.m

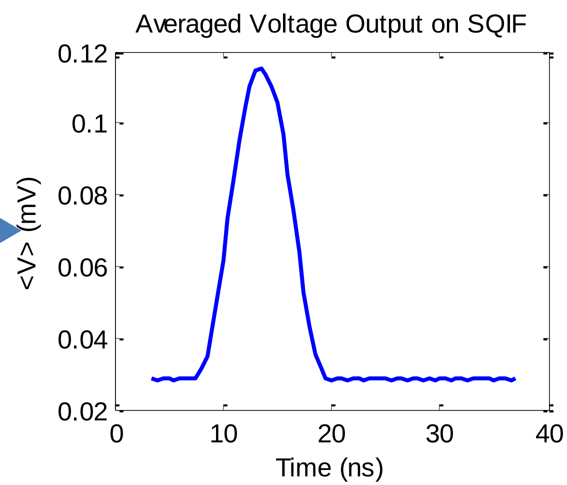
Incoming
plane wave



Transient SQIF output

SQIF output after filtered



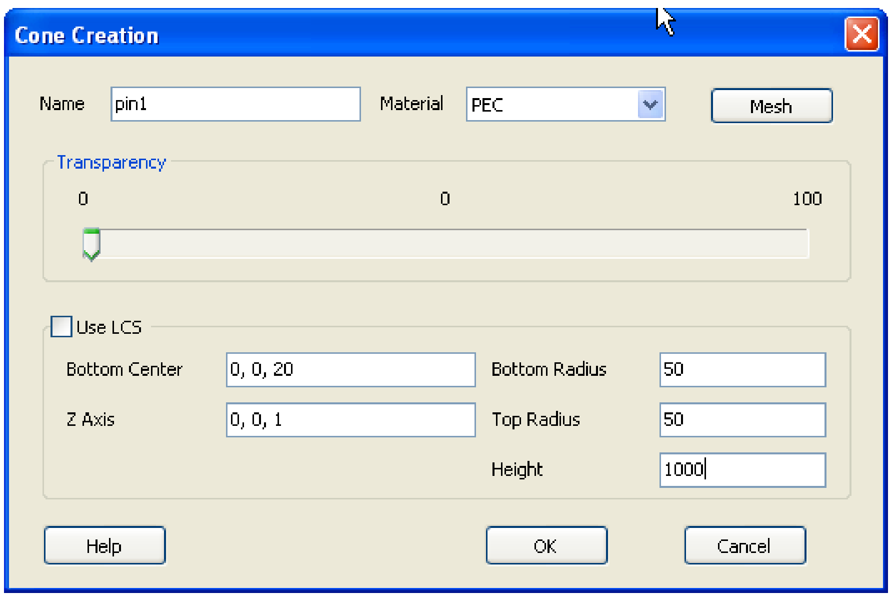


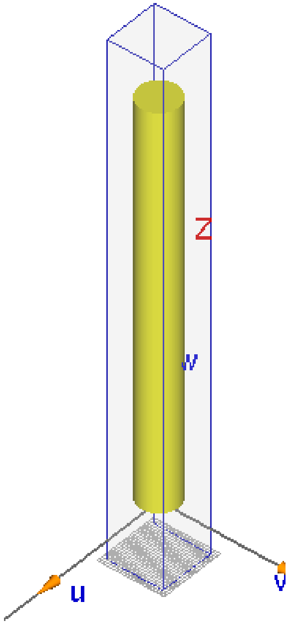
Set up a SQIF and EM Simulation using case 5

1. Start a default WCT EM case.

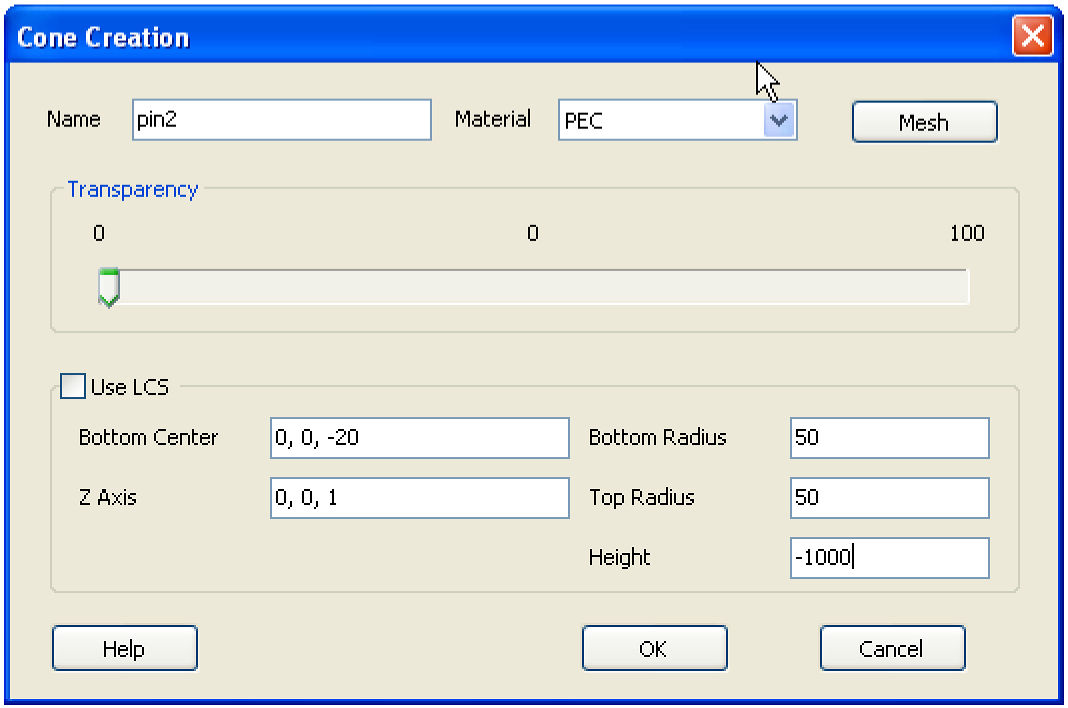
2. Build the dipole antenna

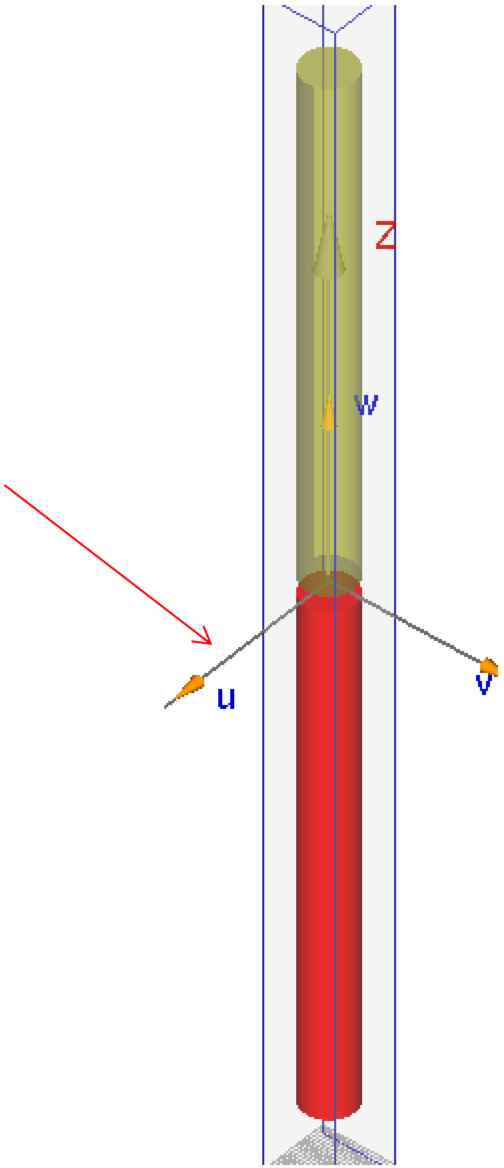
2.1 Top pin (use WCT ***CONE*** shape)



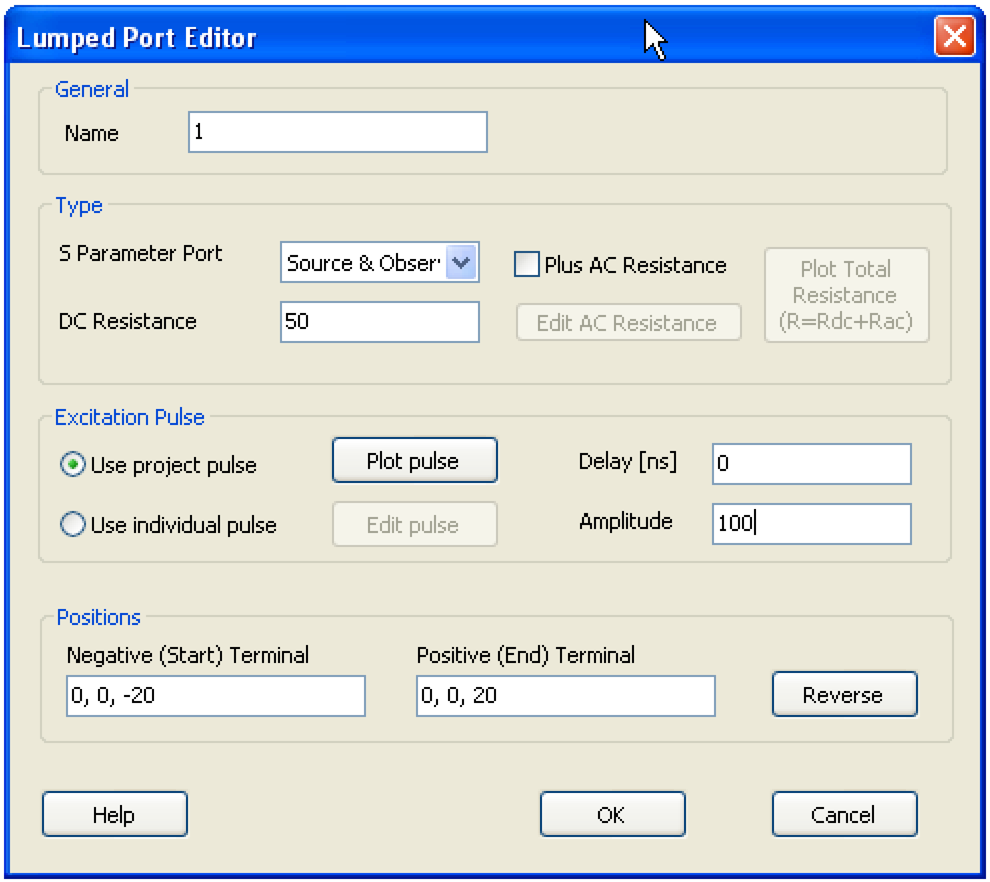


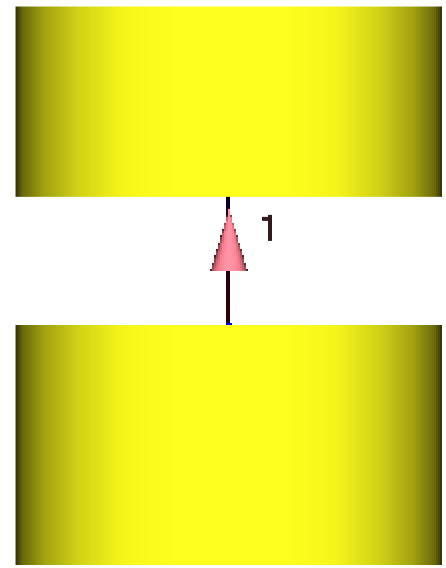
2.2 Bottom pin (use WCT *CONE* shape)





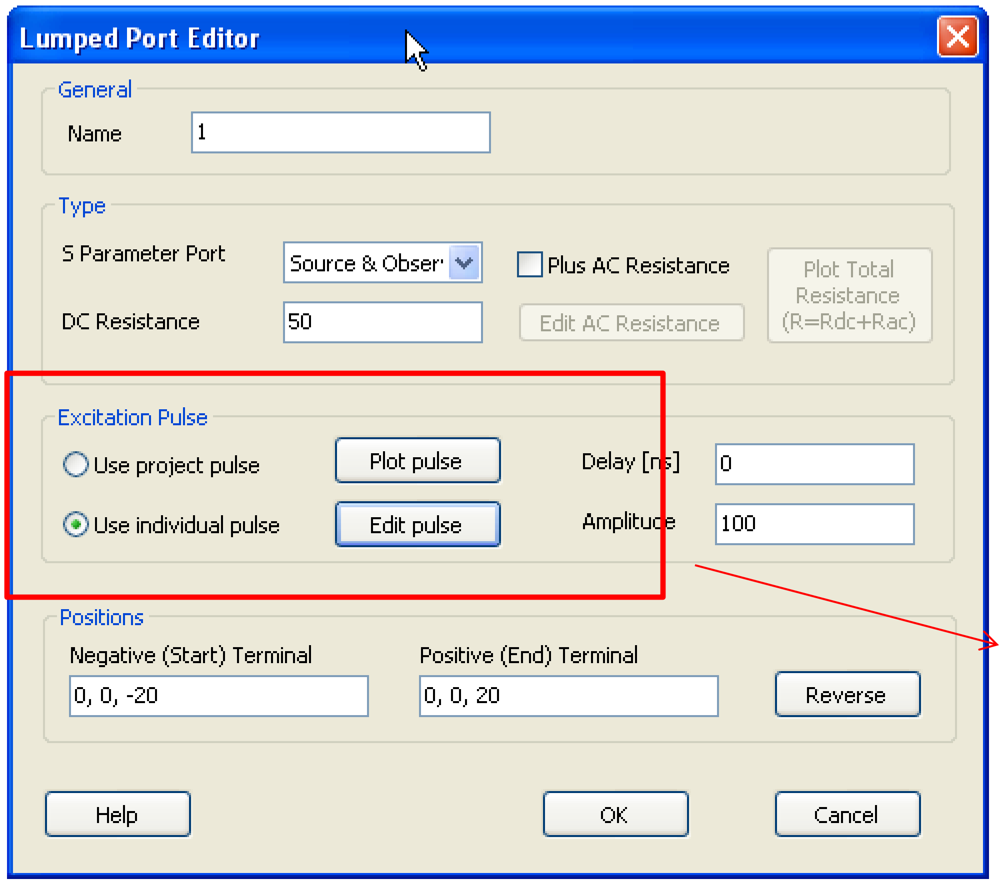
2.3 add a lumped port source which can excite the dipole antenna. This lumped port has an excitation voltage of 100 V.

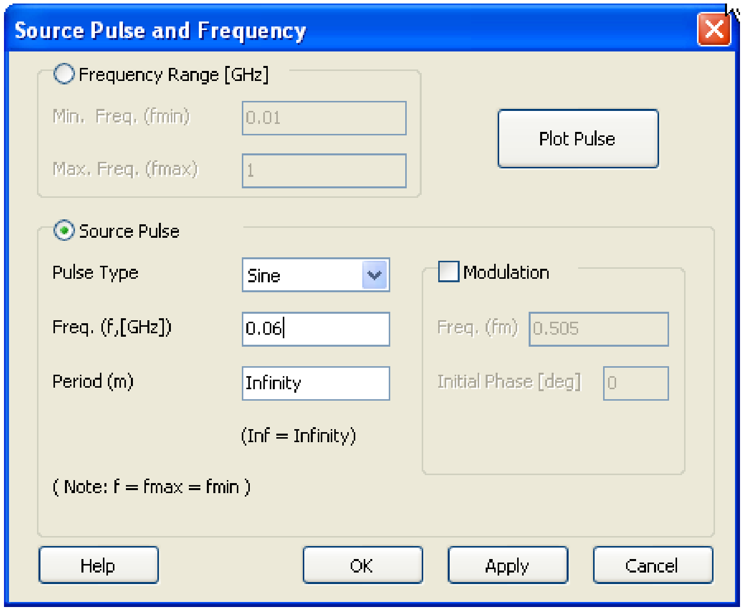




We will let the dipole antenna sending single freq. signal.

So, we change the lumped port excitation waveform to a infinite sinusoid wave.

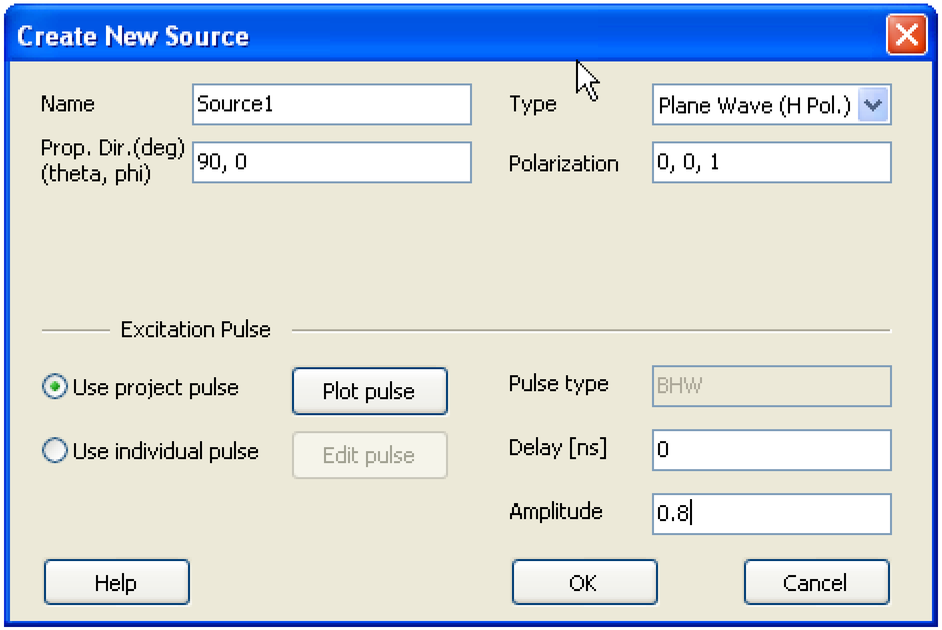


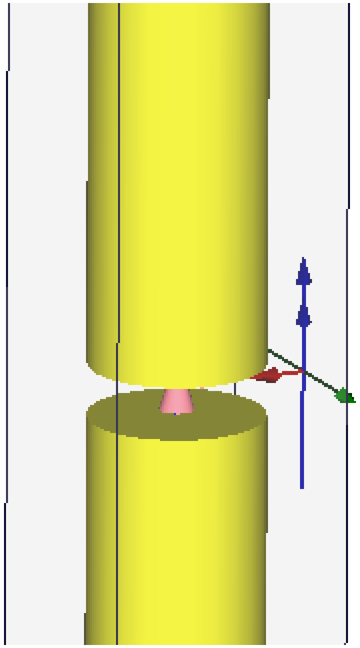


3. Add an incoming plane wave.

This plane is a *z* polarized H plane wave with a magnitude of 0.8 A/m.

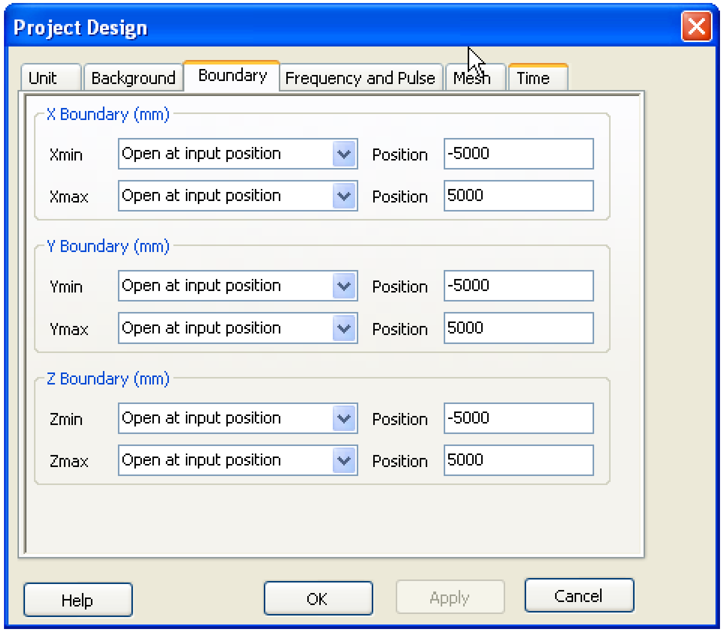
This plane wave use project pulse.

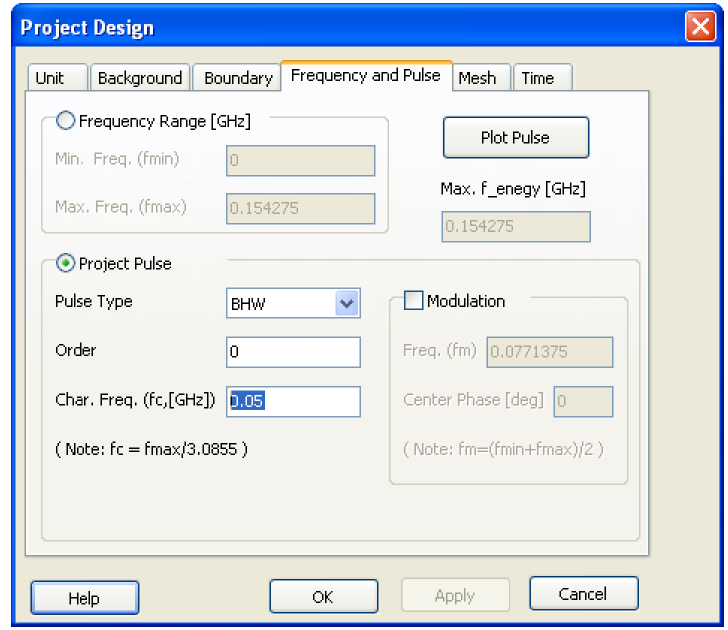


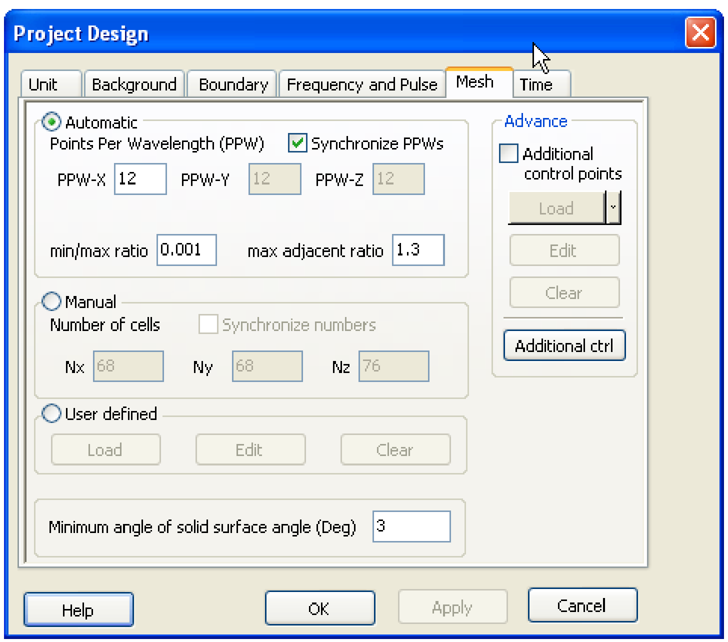


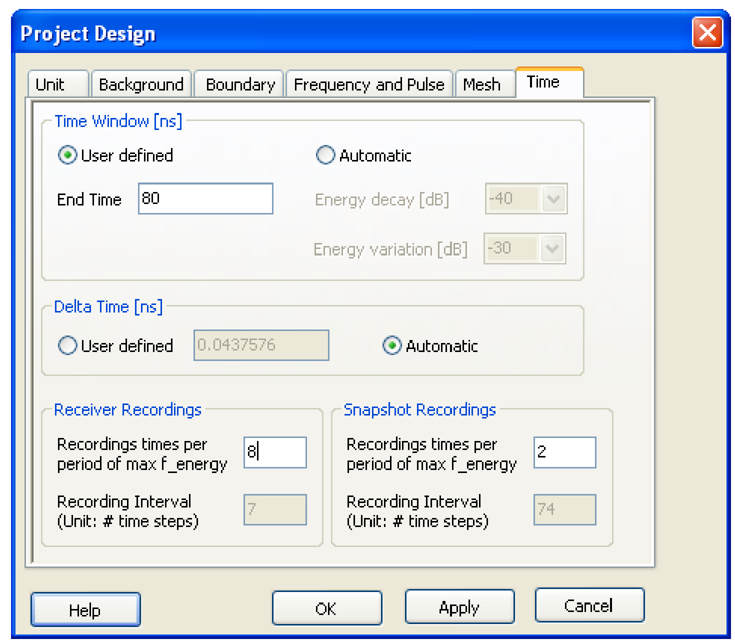
4. Change project setting.

Include project size, meshing density and simulation time.







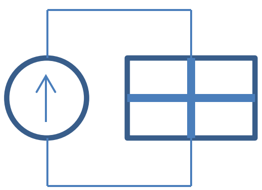


5. Setup 2D SQIF.

In this demo, we use TEXT format circuit. We will show how to use Graphic format circuit in the other cases.

A dc SQIF circuit has following structure.

Dc bias current source



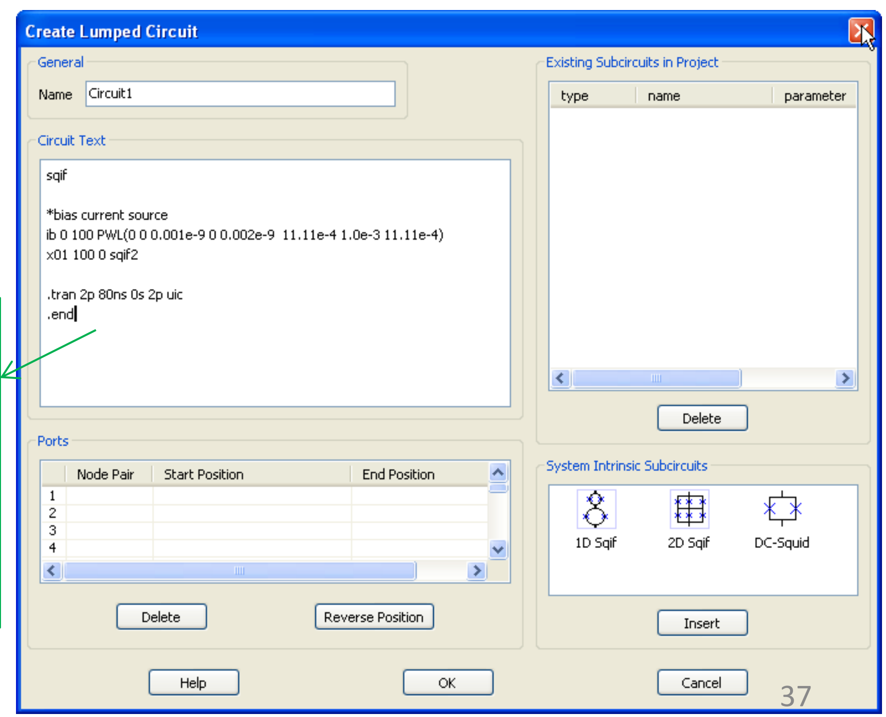
SQIF sub-circuit

Write a basic SPICE circuit with left structure.

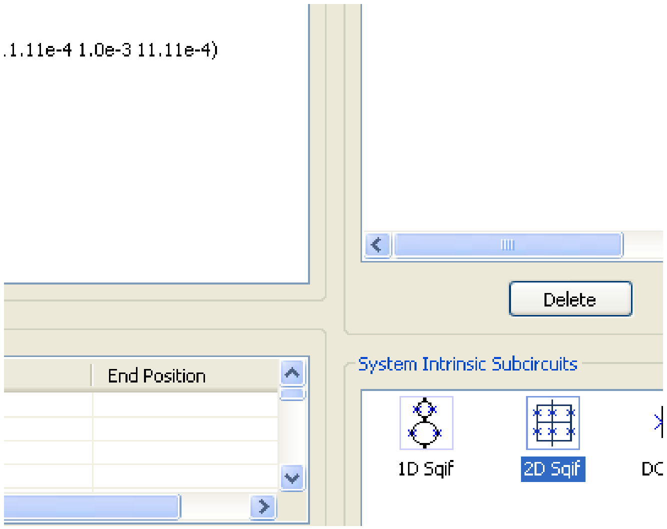
Because we use a 10x10 2D SQIF. It has 11 parallel Josephson Junctions. The I_c of Josephson Junction is 0.1 mA. Our setup use 1.01I_c for each Josephson Junction. So the total I_b=1.111 mA.

```
sqif

*bias current source
ib 0 100 PWL(0 0 0.001e-9 0 0.002e-9  11.11e-4 1.0e-3 11.11e-4)
x01 100 0 sqif2

.tran 2p 80ns 0s 2p uic
.end
```

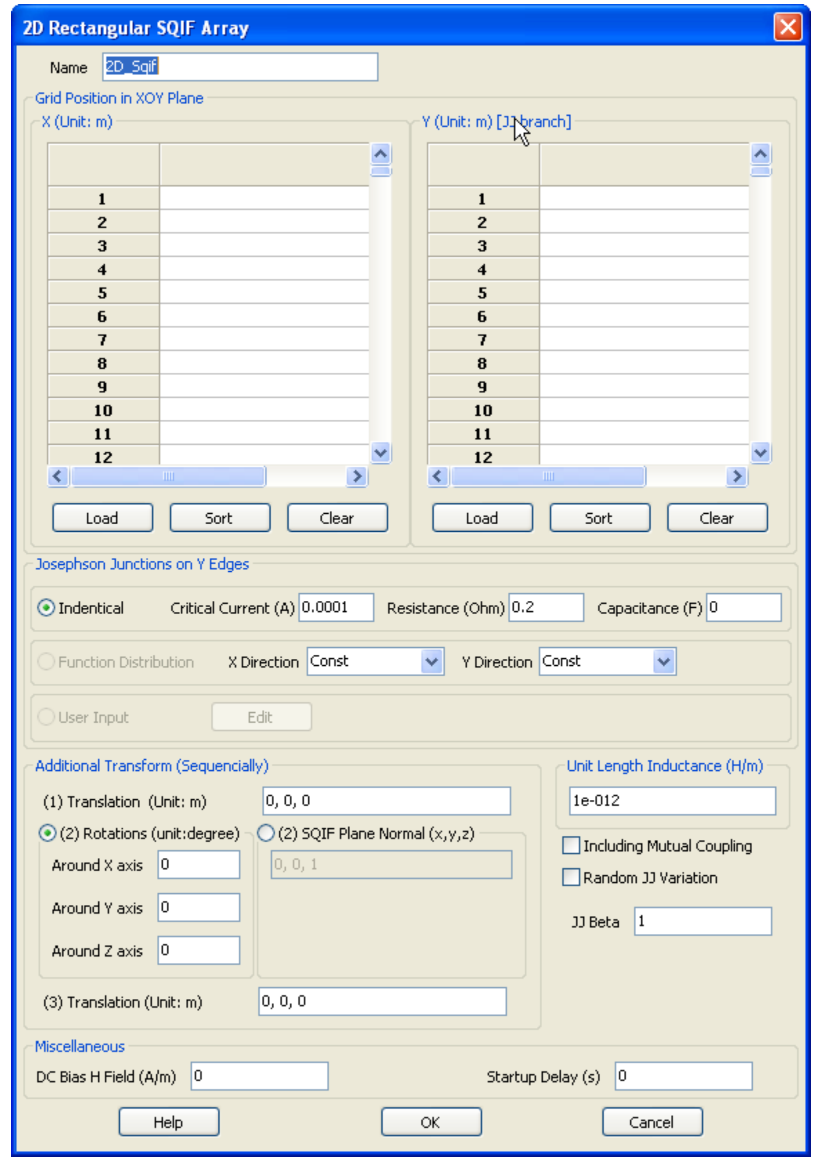
Set SQIF solver simulation time window as EM simulation time window.



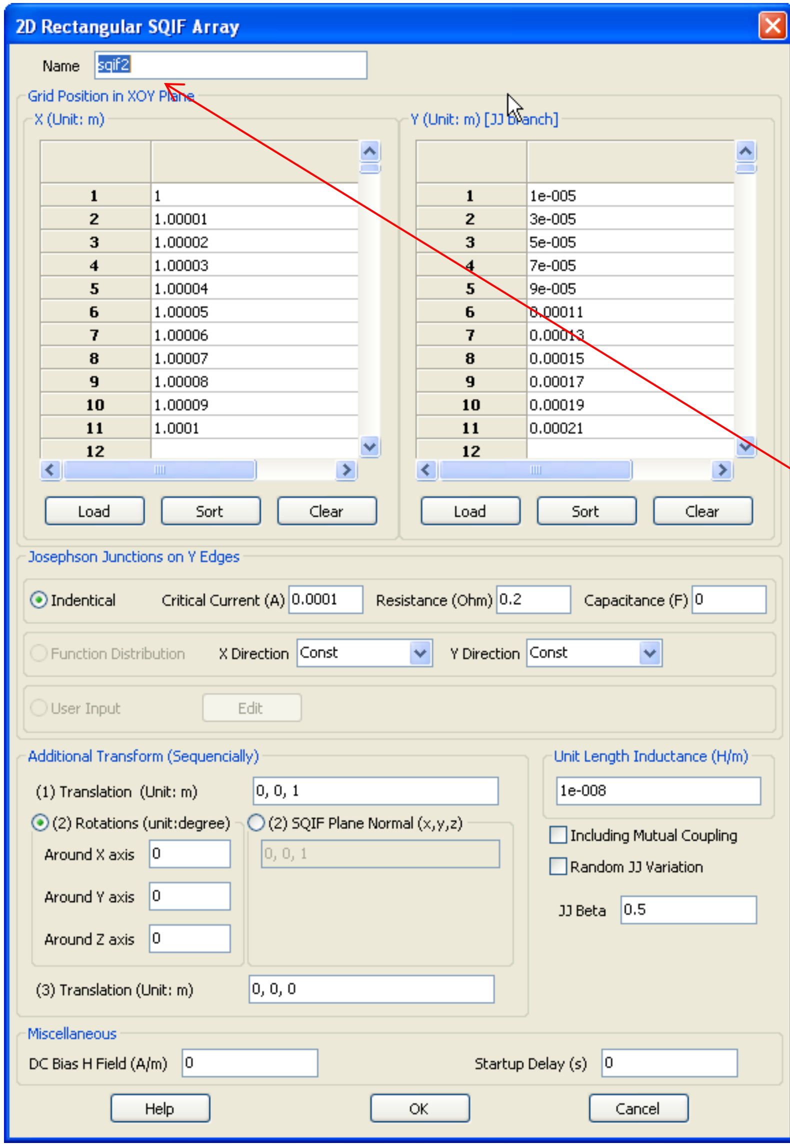
After the main circuit is built, we need to let the main circuit connect to SQIF.



Double click this 2D SQIF sub-circuit to popup a 2D SQIF editor.



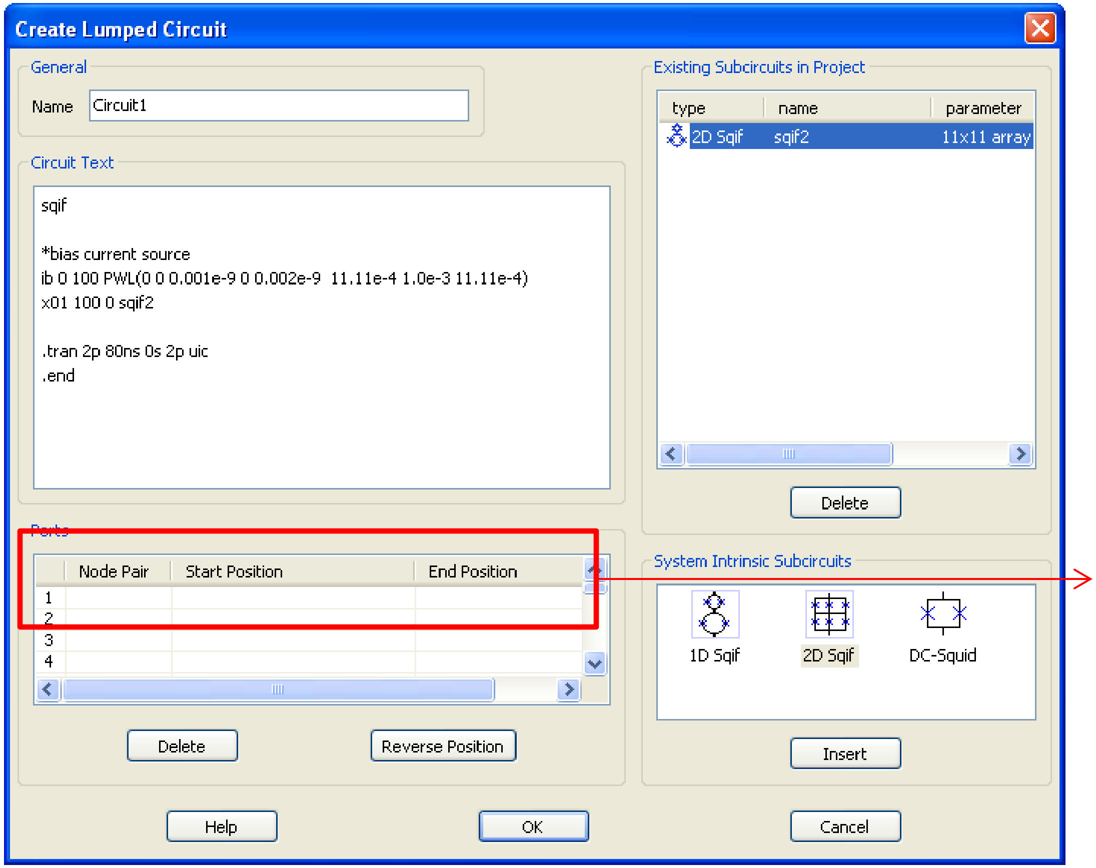
Setup 2D SQIF parameters



Note: make sure the 2D SQIF sub-circuit name is the same as the one used in main circuit.

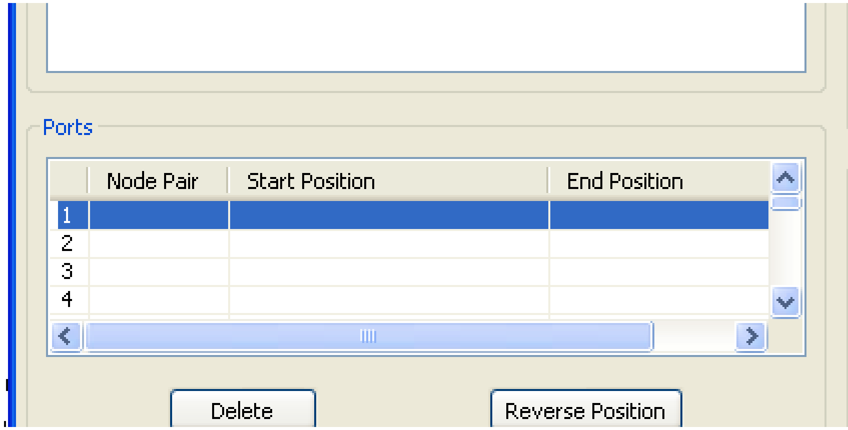
```
sqif

*bias current source
ib 0 100 PWL(0 0 0.001e-9 0 0.002e-9  11.11e-4 1.0e-3 11.11e-4)
x01 100 0 sqif2

.tran 2p 80ns 0s 2p uic
.end
```

After SQIF parameter is done.

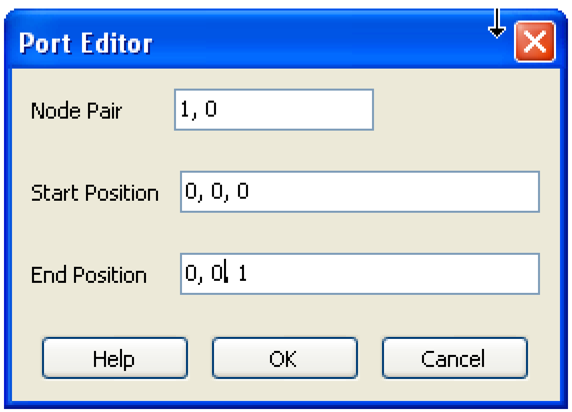


Need to add some other input to let this circuit can be fully support by WCT circuit solver.

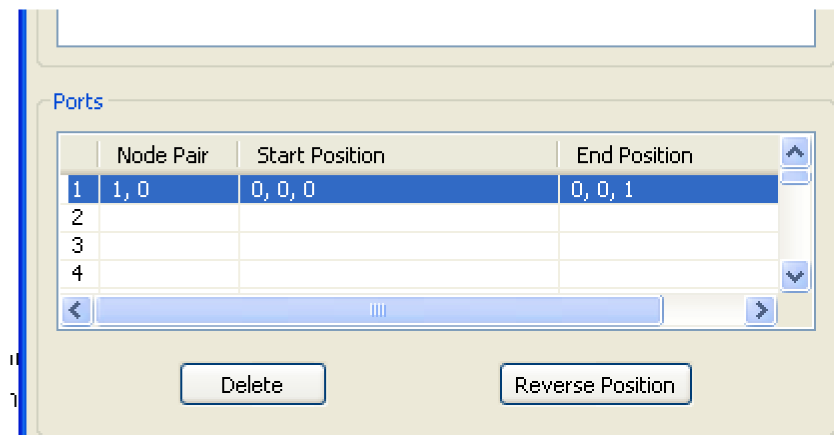
Actually, for a SQIF simulation, we don't need this information in SQIF solver. However, in current WCT version, user need to add this information to make circuit definition compatible with whole system.



Double click this item



Type in this information



5. Simulate to get result.