I. Introduction

This laboratory experiment aims to utilize the fundamental numerical and analytical techniques of LTspice to accomplish two objectives:

(1) Perform time-domain simulations of a parallel LC resonant circuit and investigate how different initial conditions influence its voltage and current responses;

(2) Construct the SPICE netlist for a given linear resistive network and derive its matrix equations using the Modified Nodal Analysis (MNA) method, in order to understand the numerical solution process employed by circuit simulators.

II. Simulation of a parallel LC resonant circuit

In the first part of this experiment, the dynamic behavior of a parallel LC resonant circuit is analyzed through analytical derivation and LTspice simulations. As illustrated in Fig. 1, the procedure begins by deriving the differential equation of the LC circuit based on the constitutive relations of the capacitor and inductor. From this, the natural angular frequency, oscillation period, and general form of the free response are obtained, providing the theoretical foundation for subsequent simulation analysis.

Next, a lossless LC circuit with no external excitation and no initial stored energy is simulated in LTspice to examine its time-domain response. Finally, initial voltage and current conditions are applied to the circuit in LTspice, and the resulting free oscillation behavior is analyzed.
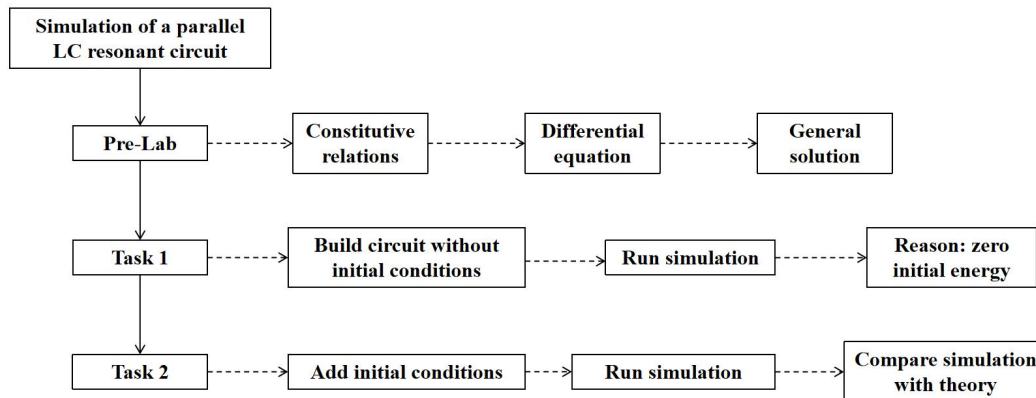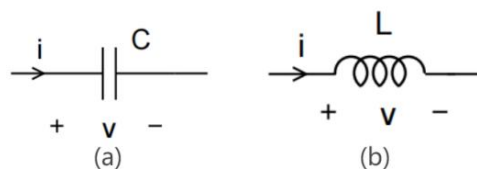

Fig.1 Part 1 Experimental flow

1. Pre-lab (1 mark)

1.1 Procedure

The circuit symbols for the capacitor and inductor are shown in Fig. 2(a) and Fig. 2(b).

Their time-domain constitutive relations are given by:

$$i_C = C\,\frac{dv}{dt} \tag{1}$$

$$v_L = L\,\frac{di}{dt} \tag{2}$$

When the capacitor and inductor form a closed loop, applying Kirchhoff's Voltage Law yields the second-order differential equation:

$$\frac{d^2 v}{dt^2} + \frac{1}{LC}\,v = 0 \tag{3}$$

1.2 Result

Let the natural angular frequency be: $\omega_0 = \frac{1}{\sqrt{LC}}$.

Then, the general solutions for the voltage and current are:

$$v(t) = V_0 \cos \omega_0 t + \frac{I_0}{C\omega_0} \sin \omega_0 t \tag{4}$$

$$i(t) = - C\omega_0 V_0 \sin \omega_0 t + I_0 \cos \omega_0 t \tag{5}$$

Where $v(0) = V_0, i(0) = I_0$.

For the selected component values $L = 1\,\text{H}, C = 25\,\text{mF} = 0.025\,\text{F}$, the natural angular frequency and oscillation period are calculated as: 6.325 rad/s and 0.99 s.

2. Task 1 (1 mark)

2.1 Procedure

According to the experiment requirements, the LC circuit is constructed in LTspice, and the following transient analysis directive is added: .tran 0 6 0 0.01.
The resulting circuit schematic is shown in Fig. 3.
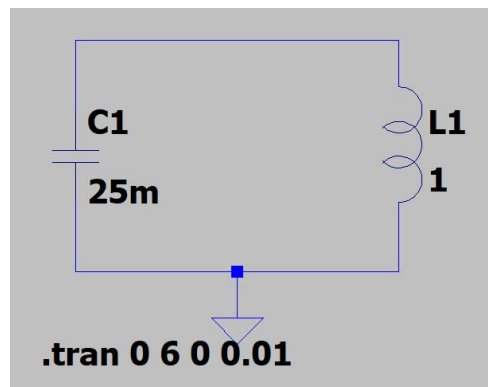
Fig.3 Task1 circuit construction

2.2 Results

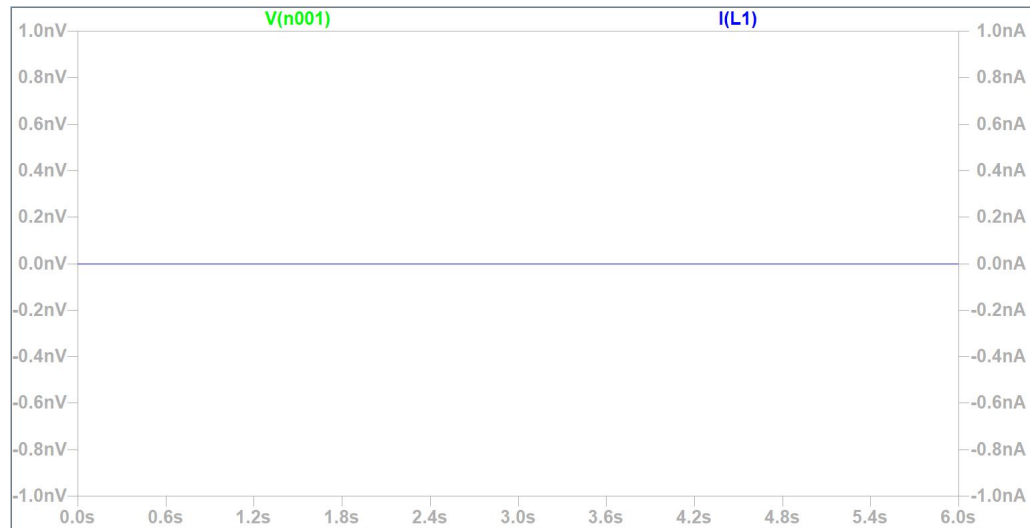The simulated voltage and current waveforms are shown in Fig. 4.

2

Fig.4 Task 1 simulation waveform

2.3 Analysis and comparison

As seen from the waveforms, both the voltage and current remain identically zero. Because the circuit contains no initial stored energy and no external sources are applied, the free response of the LC tank reduces to the trivial solution: $v(t) = 0, i(t) = 0.$ This result is fully consistent with the theoretical prediction.

3. Task 2 (2 marks)

3.1 Procedure

Initial conditions are added on the basis of Task 1, as listed in Table 1.

Table 1 Initial Condition Instructions

Instruction	Description
.ic i(L1)=0	sets the initial inductor current to 0 A
.ic v(N001)=10	sets the initial capacitor (node) voltage to 10 V

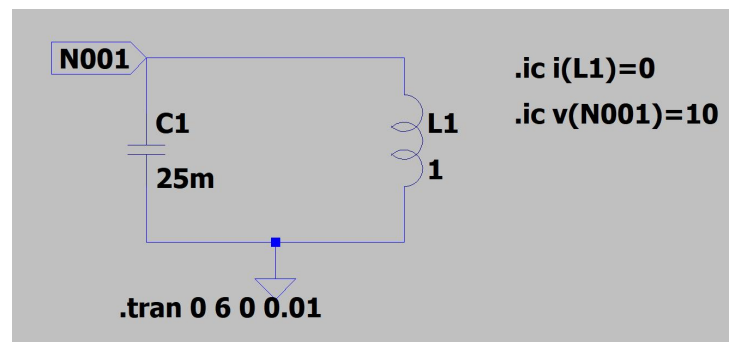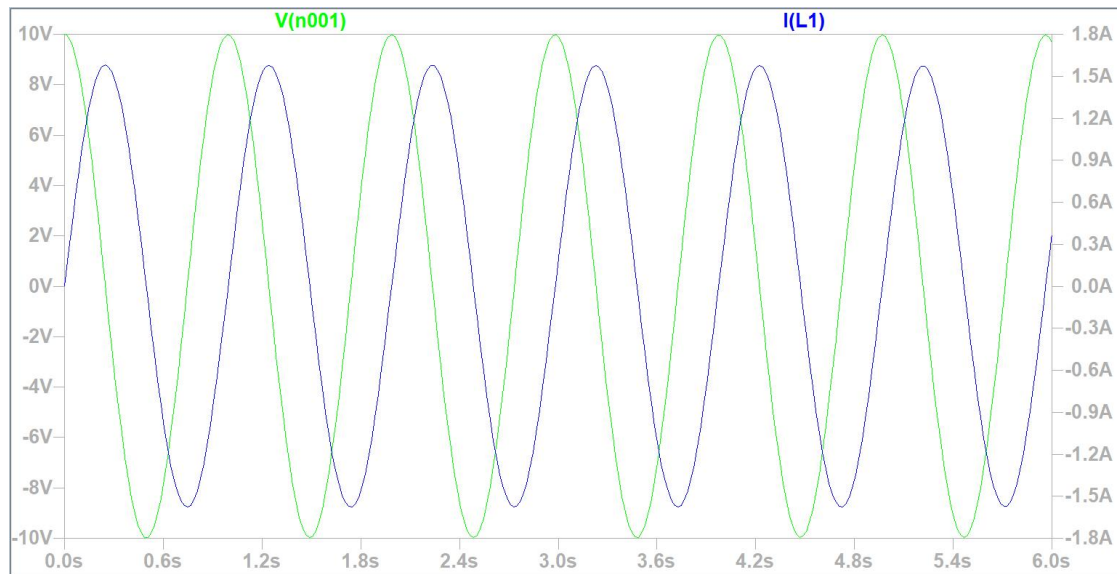The circuit schematic remains the same as in Fig. 5.

Fig.5 Task 2 Circuit construction

3.2 Results

3

The simulated waveforms are shown in Fig. 6.

Fig.6 Task 2 simulation waveform

The simulated waveforms are shown in Fig. 6. It can be observed that the voltage starts at +10 V and oscillates according to a cosine waveform, while the current starts from 0 A and follows a sine waveform. The two signals maintain a constant 90° phase difference and exhibit stable, undamped sinusoidal oscillations characteristic of an ideal LC resonant circuit.

To quantitatively extract key waveform parameters, the .meas commands listed in Table 2 are used.

Table 2 Task 2 Instructions

Instruction	Description
.meas tran Vpeak MAX V(N001)	Peak output voltage
.meas tran Ipeak MAX I(L1)	Peak output current
.meas tran tV1 WHEN V(N001)=0 RISE=1	Measure voltage first zero point
.meas tran tV2 WHEN V(N001)=0 RISE=2	Measure the voltage at the second zero point
.meas tran Tperiod PARAM='tV2 - tV1'	Computing cycle
.meas tran tI1 WHEN I(L1)=0 RISE=1	Measure the first zero point of the current
.meas tran PhaseDeg PARAM='360*(tI1 - tV1)/Tperiod'	Calculate the phase difference between current and voltage

The measurement results are summarized in Table 3.

Table 3 Simulation Results

Simulation object	Value
Peak voltage	10V
Peak current	1.58A
Period	0.99s
Phase difference	90°

To quantitatively extract key waveform parameters, the .meas commands listed in Table 2 are

4

used. The measurement results are summarized in Table 3. The voltage peak is approximately 10 V, the current peak is approximately 1.58 A, the oscillation period is about 0.99 s, and the phase difference between voltage and current is 90°.

3.3 Analysis and Comparison

Comparing the simulations with the pre-lab analytical results, under the initial conditions $V_0 = 10\,V, I_0 = 0$, equations(4) and (5) reduce to:

$$v(t) = 10\cos(\omega_0 t) \tag{6}$$

$$i(t) = -C\omega_0\, 10\sin(\omega_0 t) \tag{7}$$

From equation (7), the theoretical current amplitude is 1.581A, which matches the LTspice measurement. Since the voltage follows a cosine function and the current follows a sine function, their phase difference is expected to be 90°, consistent with the simulation.

Furthermore, the oscillation period calculated from $T = 2\pi/\omega_0 = 0.993s$, also agrees with the simulated result of approximately 0.99 s.

Overall, free oscillation occurs only when the LC circuit contains initial energy (or external excitation). The specific initial voltage and current determine the resulting oscillation amplitude and phase.

III. Composing a SPICE netlist and finding the MNA equations for a linear resistive circuit

For the second part of the experiment, the objective is to investigate how a circuit simulator computes numerical solutions from a SPICE netlist by constructing the MNA equations of a linear resistive network and generating its corresponding SPICE netlist. The overall procedure is illustrated in Fig.7. First, the MNA matrix for the given linear resistive circuit is formulated according to the principles of Modified Nodal Analysis. Then, the same circuit is implemented in LTspice to generate the SPICE netlist, and a simulation is performed to obtain node voltages, controlling currents, and other relevant numerical results. Finally, the matrix produced by the checking program is compared with the manually derived MNA matrix to verify the consistency between the SPICE netlist and the analytical MNA formulation.
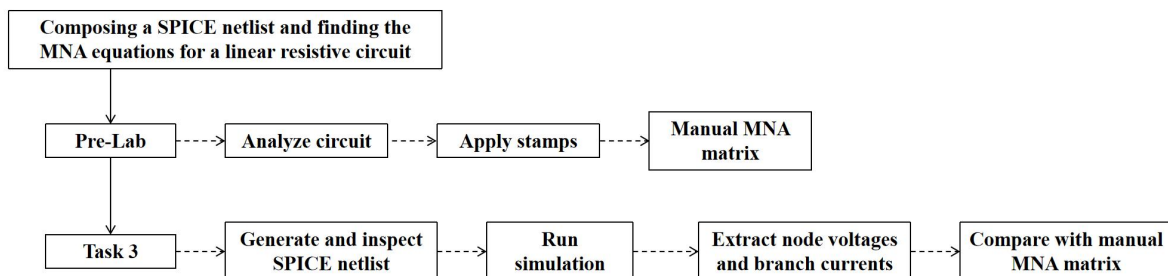

Fig.7 Workflow of Part II

2. Pre-lab (2 marks)

4.1 Procedure

5

Based on the circuit provided in the experiment (Fig. 8), the seven components are analyzed using their respective stamp representations. These individual stamps are then combined to construct the Modified Nodal Analysis (MNA) matrix in the form $\mathbf{A}\,\mathbf{x} = \mathbf{z}$。
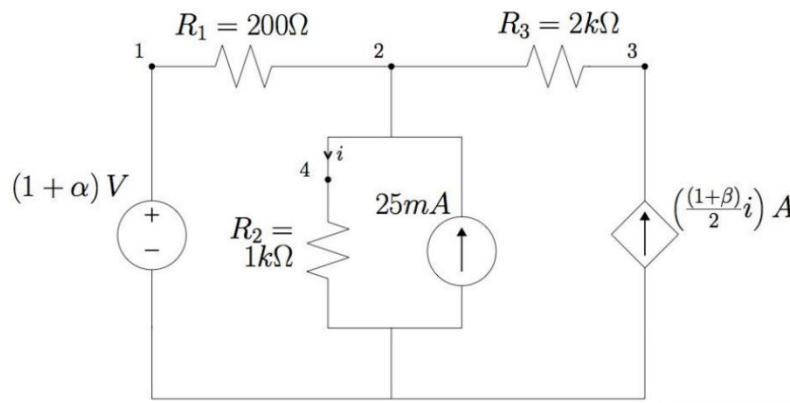

Fig.8 Circuit provided in the experiment

The stamps for the three resistors are shown in Fig.9.

Fig.9 Stamps for resistors

The stamps for the two independent voltage sources are shown in Fig.10.

Fig.10 Stamps for independent voltage sources

6

The stamp corresponding to the current-controlled current source (CCCS), together with the stamp for the independent current source, is shown in Fig.11.
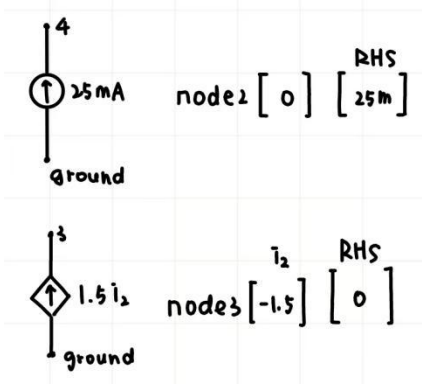

Fig.11 Stamp for current-controlled current source and independent current source

4.2 Results

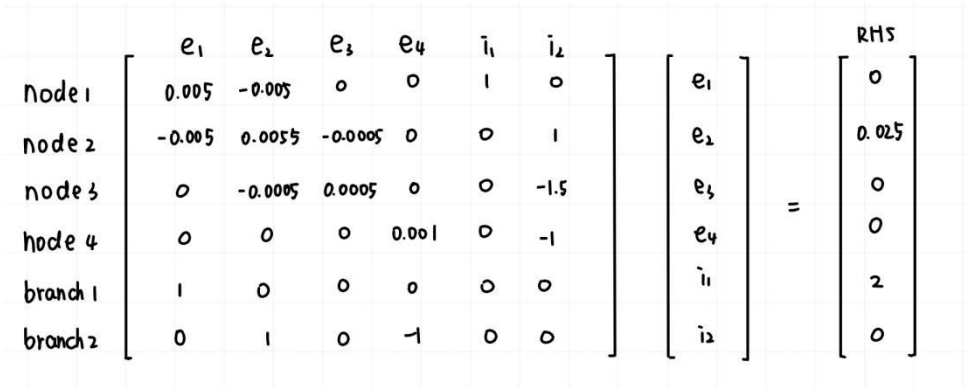The manually derived MNA matrix for the circuit is shown in Fig.12.

Fig.12 MNA matrix

3. Task 3 (2 marks)

5.1 Procedure

First, the circuit is constructed in LTspice, as shown in Fig.13.
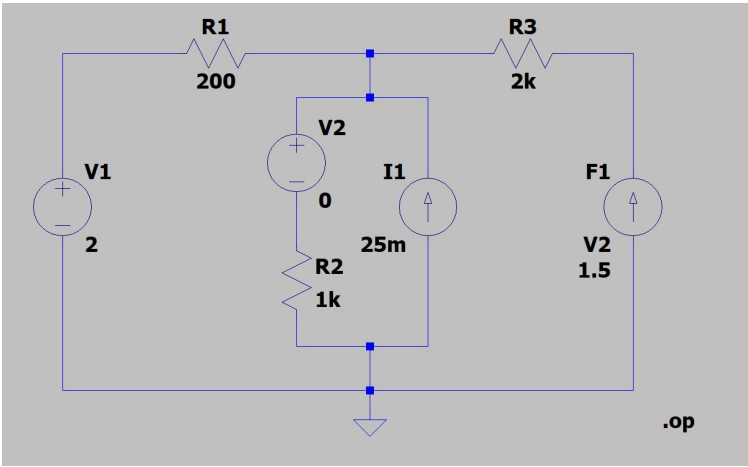

Fig.13 Circuit construction for Task 3

Next, the SPICE netlist is opened and examined (Fig. 14) to ensure that it accurately reflects the intended circuit.

```
* Generated by LTspice 24.1.10 for Windows.
V1 N001 0 2
R1 N002 N001 200
R3 N003 N002 2k
V2 N002 N004 0
R2 N004 0 1k
I1 0 N002 25m
F1 0 N003 V2 1.5
.op
.backanno
.end
```

Fig.14 LTspice netlist

A DC operating point analysis is then performed, and the results are presented in Fig. 15.

```
          --- Operating Point ---

    V(n001):        2               voltage
    V(n002):        7.77778         voltage
    V(n003):        31.1111         voltage
    V(n004):        7.77778         voltage
    I(F1):          0.0116667       device_current
    I(I1):          0.025           device_current
    I(R1):          0.0288889       device_current
    I(R2):          0.00777778      device_current
    I(R3):          0.0116667       device_current
    I(V1):          0.0288889       device_current
    I(V2):          0.00777778      device_current
```

Fig.15 Simulation results (DC operating point)

5.2 Results

The results of this task are summarized in Table 4. Notably, the value of I(V2) corresponds to the controlling current required by the CCCS.

Table 4. Simulation results

Object		Value
Node voltage	N001	2 V
	N002	7.78 V
	N003	31.11 V
	N004	7.78 V
Branch current	I(V2)	7.78 mA
	I(I1)	25 mA

5.3 Analysis and Comparison

Finally, the provided check_MNA.exe program is used to process the netlist created in this task. The resulting matrix and coefficients are shown in Fig. 16.

8

```
Default * - near end or something wrong
No stamp found!
This circuit Draft4.net contains 5 nodes including the reference (ground) node.

It contains:
3 Resistors
1 Current Sources
2 Voltage Sources
0 VCVS
0 VCCS
1 CCCS
0 CCVS

The Ymatrix is
|0.005000 -0.005000 0.000000 0.000000 1.000000 0.000000 |
|-0.005000 0.005500 -0.000500 0.000000 0.000000 1.000000 |
|0.000000 -0.000500 0.000500 0.000000 0.000000 -1.500000 |
|0.000000 0.000000 0.000000 0.001000 0.000000 -1.000000 |
|1.000000 0.000000 0.000000 0.000000 0.000000 0.000000 |
|0.000000 1.000000 0.000000 -1.000000 0.000000 0.000000 |

The e matrix is
|e_N001|
|e_N002|
|e_N003|
|e_N004|
|i_V1|
|i_V2|

And the RHSmatrix is
|0.000000|
|0.025000|
|0.000000|
|0.000000|
|2.000000|
|0.000000|


The MNA matricies have been written to out.txt also - Program finished.

Press the return key to close this window.
```

Fig.16 Matrix generated by the check_MNA.exe program

Comparing the program-generated matrix in Fig. 16 with the manually derived MNA matrix in Fig. 11 shows that the two are completely identical. This confirms that the SPICE netlist accurately represents the circuit and that the analytical MNA derivation is correct.