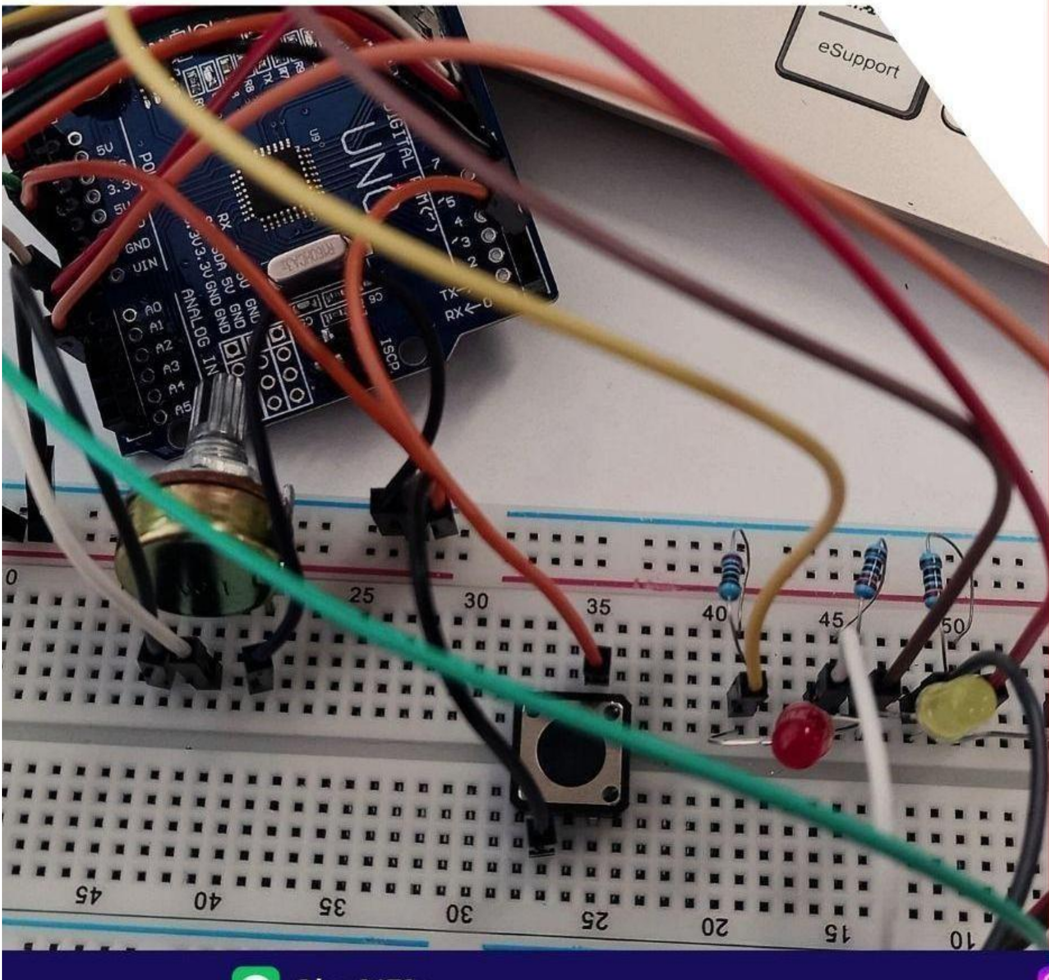
ELECTRICAL CIRCUIT PRACTICUM

MODULE

2025

ELECTRICAL CIRCUIT LABORATORY
ELECTRONIC SYSTEM EXPERTISE GROUP
FACULTY OF ELECTRICAL ENGINEERING
TELKOM UNIVERSITY

ONLY USED FOR
ELECTRICAL ENGINEERING FACULTY



ELECTRICAL CIRCUIT PRACTICAL MODULE PREPARATION TEAM

Head of Laboratory	: Dr. Rahmat Awaludin Salam	
Laboratory Builder	: Mohamad Ramdhani, S.T., M.T.	
Assistant Coordinator	: Alfian Ramadhan Munthe	1102223011
Deputy Assistant Coordinator	: Daniel Parulian	1102220192
Secretary	: Mardianah Tanza	1105223086
Secretary	: Virgin Virana Paradise	101022300077
Treasurer	: Hilaliyah Ayu Faoziyah	1102223025
Treasurer	: Mahdiya Huda	101022330330
Head of Practical Division	: Dinda Amalia Lestari	1101223188
Practical Division	: Rheira Nisrina Abiyah	1101223224
Practical Division	: Najwa Syafira Firdaus	1103223110
Practical Division	: Rayhan Imannasywan Akbar	1105220107
Practical Division	: Kaysa Adara Karim	101012330358
Practical Division	: Liony Syafitri	101012330035
Practical Division	: Sebastian Cahyaputra	101022300014
Practical Division	: Haifa Mohammad Adam	101012330060
Practical Division	: Zaed Al Musthofa	101022300219
Head of Hardware Division	: Raadhii Tsaqib Rabbanii	1105220081
Hardware Division	: Ahzami Muhammad Averous	1102223060
Hardware Division	: Iki Tayubi	1102223031
Hardware Division	: Keisha Mesmeralda Louis Silalahi	1102223081
Hardware Division	: Devin Marva Kusuma	1102220124
Hardware Division	: Daffa Aryaputra	101022330009
Hardware Division	: Haniyah Melati Utomo	101022300286
Hardware Division	: Muhammad Raffi Ibrahim	101032300020
Hardware Division	: Badar Zaki Baradja	101012300080
Hardware Division	: Rakha Tantra	101022330198
Hardware Division	: Arria Brata Sena Majid Budiyanto	101022300209
Head of Admin Division	: Ramzy Fawwaz	1102223113
Admin Division	: Mutia Azzahra Rahmadhani	1101223137
Admin Division	: Raissa Sadina Rendra	101022340183
Admin Division	: Myanda Piyay Nabila Putri	101052300045
Admin Division	: Muhammad Nur Hidayatullah	101012300360
Admin Division	: Nabilatul Inayah	101022300231
Admin Division	: Muhamad Naufal Jauhar Amjad	101022330222
Head of Research and Development Division	: Tasha Arafina Airyn	1102223008
Research and Development Division	: Nisrina Putri Nadhira	1101220081
Research and Development Division	: Abdurrasyid Ridho	1103223041
Research and Development Division	: Patar Idaon Situmorang	101012300016
Research and Development Division	: Mutia Maulida	101022330063
Research and Development Division	: Indah Natalia Nadeak	101022300167
Research and Development Division	: Zahra Ramadhina	101052300086
Research and Development Division	: Najwa Bilqis Al Khalidah	101032300186
Research and Development Division	: Agastya Pristyanto	101022300229

ELECTRICAL CIRCUITS LABORATORY PRACTICUM ASSISTANT 2025/2026

No	NIM	Assistant's Name	Position
1	1102223011	Alfian Ramadhan Munthe	Assistant Coordinator
2	1102220192	Daniel Parulian	Deputy Assistant Coordinator
3	1105223086	Mardianah Tanza	Secretary
4	101022300077	Virgin Virana Paradise	Secretary
5	1102223025	Hilaliyah Ayu Faoziyah	Treasurer
6	101022330330	Mahdiya Huda	Treasurer
7	1101223188	Dinda Amalia Lestari	Head of Practical Division
8	1101223224	Rheira Nisrina Abiyah	Practical Division
9	1103223110	Najwa Syafira Firdaus	Practical Division
10	1105220107	Rayhan Imannasywan Akbar	Practical Division
11	101012330358	Kaysa Adara Karim	Practical Division
12	101012330035	Liony Syafitri	Practical Division
13	101022300014	Sebastian Cahyaputra	Practical Division
14	101012330060	Haifa Mohammad Adam	Practical Division
15	101022300219	Zaed Al Musthofa	Practical Division
16	1105220081	Raadhii Tsaqib Rabbanii	Head of Hardware Division
17	1102223060	Ahzami Muhammad Averous	Hardware Division
18	1102223031	Iki Tayubi	Hardware Division
19	1102223081	Keisha Mesmeralda Louis Silalahi	Hardware Division
20	1102220124	Devin Marva Kusuma	Hardware Division
21	101022330009	Daffa Aryaputra	Hardware Division
22	101022300286	Haniyah Melati Utomo	Hardware Division
23	101032300020	Muhammad Raffi Ibrahim	Hardware Division
24	101012300080	Badar Zaki Baradja	Hardware Division
25	101022330198	Rakha Tantra	Hardware Division
26	101022300209	Arria Brata Sena Majid Budiyanto	Hardware Division
27	1102223113	Ramzy Fawwaz	Head of Admin Division
28	1101223137	Mutia Azzahra Rahmadhani	Admin Division
29	101022340183	Raissa Sadina Rendra	Admin Division
30	101052300045	Myanda Piyay Nabila Putri	Admin Division
31	101012300360	Muhammad Nur Hidayatullah	Admin Division
32	101022300231	Nabilatul Inayah	Admin Division
33	101022330222	Muhamad Naufal Jauhar Amjad	Admin Division
34	1102223008	Tasha Arafina Airyn	Head of Research and Development Division
35	1101220081	Nisrina Putri Nadhira	Research and Development Division
36	1103223041	Abdurrasyid Ridho	Research and Development Division
37	101012300016	Patar Idaon Situmorang	Research and Development Division
38	101022330063	Mutia Maulida	Research and Development Division
39	101022300167	Indah Natalia Nadeak	Research and Development Division
40	101052300086	Zahra Ramadhina	Research and Development Division
41	101032300186	Najwa Bilqis Al Khalidah	Research and Development Division
42	101022300229	Agastya Pristyanto	Research and Development Division

Preface

We give thanks to the presence of the Almighty God, the Most Gracious and Merciful, because thanks to His grace and guidance, the Electric Circuit Practical Module can be completed.
On this occasion we would like to thank all parties who have contributed, so that the Electric Circuit Practicum Module can be published.
We really hope for criticism and suggestions for future improvements.
Hopefully this book can be useful for readers. Enjoy reading and carrying out the practicum

Bandung, 5 August 2025

Drafting Team

REVISION SHEET

The undersigned below::

Name : Dr. Rahmat Awaludin Salam

NIP 14890058

Position : Head of the electrical circuit laboratory

Hereby declare that the implementation of the Revision of the Electrical Circuit Practicum Module for the Undergraduate Electrical Engineering study program has been carried out with the following explanation:

NO	Modul	Keterangan Revisi	Tanggal Revisi Terakhir
1		Penyesuaian *template* modul	5 Agustus 2025

STATEMENT SHEET

The undersigned below:

Nama : Dr. Rahmat Awaludin Salam

NIP 14890058

Jabatan : Head of the electrical circuit laboratory

Explaining truly that this practicum module has been reviewed and will be used for practicum implementation in the Odd Semester of the 2024/2025 Academic Year at the Electrical Circuit Laboratory, Faculty of Electrical Engineering, Telkom University

Bandung, 5 August 2025

Accepted by,
Head of the electrical circuit laboratory

Dr. Rahmat Awaludin Salam
NIP 14890058

Acknowledged by,
Koordinator Asisten Laboratorium Rangkaian Listrik

Alfian Ramadhan Munthe
NIM 1102223011

Vision and Mission of the Faculty of Electrical Engineering

VISION:

To become a globally recognized faculty that actively contributes to the development of education, research, and entrepreneurship in electrical engineering and physics, with a foundation in information technology.

MISSION:

1. Organizing an international standard education system in the engineering field electrical and physical engineering based on information technology.
2. Organize, disseminate and utilize standard research results Internationally in electrical engineering and physics.
3. Organizing a technology-based entrepreneurship program in electrical engineering and physics engineering among the academic community to support national economic development.
4. Develop networks with leading universities and industry in Indonesia abroad in the context of educational, research and entrepreneurship collaboration.
5. Develop resources to achieve excellence in education, research and entrepreneurship

RULES FOR IMPLEMENTING PRACTICUM ACTIVITIES ODD SEMESTER ACADEMIC YEAR 2025/2026 FACULTY OF ELECTRICAL ENGINEERING

Following up on Circular Letter Number: 358/AKD 6/AKD-BAA/2022 concerning Implementation of the Lecture Model. The following are the Rules for Implementing Practicum Activities for the Even Semester Academic Year 2024/2025.

A. **Requirements for Participating in Practicum**

The requirements for students taking part in the practicum consist of:

1. Students can take part in practicum if they take credits in practicum courses and have taken or are currently taking credits in related courses
2. Students who continue to take credits for practicum courses without fulfilling the requirements for point 1, the practicum grades are invalid and cannot be saved.
3. For students who do not pass certain practicum courses, they are not required to take all the practicum courses, students can repeat only the practicum courses that are declared not to have passed.

B. Practical Implementation Rules for Laboratory Assistants

Every laboratory assistant who carries out practical work must comply with the following rules:

1. Laboratory assistants are required to wear official Telkom University uniforms and carry a valid Student Identity Card (KTM).
2. All laboratory assistants with long hair are required to tie their hair
3. Laboratory assistants are prohibited from eating and drinking indoors during practicum
4. Laboratory assistants who do not carry out practical assistance according to the predetermined schedule for any reason are required to report to OA Line seelabs

C. Practicum Implementation Rules for practitioners

Every practitioner who takes part in the practicum must comply with the following rules:

1. All Practitioners MUST wear the official Telkom University uniform
2. For safety during the practicum, hair must be neat and practitioners with long hair must tie their hair
3. The practicum is carried out for 2.5 hours according to a predetermined schedule
4. Practitioners must be present 10 minutes before the practicum begins

5. If the practitioner is late for over 20 minutes, they will not be allowed to take part in the practical activities in the module being implemented.
6. Practitioners can carry out practicum after receiving instructions from the Practicum Assistant
7. During the practicum, Practitioners are prohibited from:

- Eating, drinking and smoking
- Making noise in the room
- Changing software or hardware configuration
- Leaving the room without the permission of the Practicum Assistant
- Using a smart phone without the permission of the Practicum Assistant
- Any inappropriate actions taken during the practicum

8. Implementing "special rules"/flow of practical activities in accordance with the procedures established by each laboratory
9. Practitioners who do not do practicum due to illness, incidents or accidents, religious services or campus activities that have been permitted by the Central Student Affairs Department (BK) must report to the OA line seelabs a maximum of three days after their absence, or three days beforehand.
10. If the intern violates rules 1 to 9 above, the intern will be subject to sanctions in accordance with the applicable rules

D. Practical Assessment System

1. One practicum course at the Faculty of Electrical Engineering consists of 12 practicum modules with an assessment percentage according to the number of modules for each practicum course given.
2. In particular, the practicum assessment system for students taking practicum courses at the Faculty of Electrical Engineering follows the following rules:

a). First Condition

Practical courses consist of:

a. 1 practicum course, totaling 12 practicum modules
b. 2 practicum courses, each with 6 practicum modules
c. 3 practicum courses, each with 4 practicum modules

- Students are given tolerance for not participating in the Practicum, only 1 (one) meeting for each practicum subject.

- If the absence exceeds the points above, the grade for the practicum course is worth E

b). Second Condition

The practicum course consists of 4 practicum subjects, each with 3 practicum modules

- Students are required to attend all practical meetings
- If you do not follow the provisions above then the value of the practicum course is worth E

3. For students who are repeating the practicum, they are required to confirm the grades of the practicum subjects they passed in the previous year to the laboratory assistant during the grade validation period

Bandung, August 2025
Wakil Dekan 1 Fakultas Teknik Elektro,

Dr. Mamat Rokhmat, S.Si., M.T.

RULES FOR PERMITTING PRACTICUM FOR ADDITIONAL ODD SEMESTER ACADEMIC YEAR 2025/2026 FACULTY OF ELECTRICAL ENGINEERING

A. Students who take part in practicums at the Faculty of Electrical Engineering are required to follow the rules of the Faculty of Electrical Engineering and each related laboratory and comply with the sanctions that have been determined.

B. The requirements for applying for a follow-up practicum permit only apply to those who meet the following criteria:

Sick

a. Hospitalized

- Practitioners are required to attach a certificate of hospitalization from the hospital/puskesmas/relevant agency
- Practitioners are required to process applications for follow-up practicum a maximum of seven days after the inpatient period is over

b. Chronic Disease

Practitioners are required to attach a sick letter from the hospital/puskesmas/relevant agency and provide proof of medical records showing that the illness has been suffered for a long period of time.

c. Outpatient

- Practitioners who are in the Bandung and surrounding areas are required to attach a certificate from the **"Telkomedika Kampus"** doctor stating that they are sick and need rest.
- Practitioners who are outside the Bandung area are required to provide a letter stating that they are sick and need rest from the doctor who handles them, as well as ***attaching photo evidence of the prescription and administration.***
- Practitioners are ***not permitted*** to attach a sick certificate from an online clinic/doctor
- The date on the sick note **must match** the absence's practicum schedule
- Practitioners are only given the opportunity for one follow-up practicum

Academic Dispensation

Practitioners must attach a copy of proof of dispensation approval from i-Gracias which has been approved by the Student Body

Disaster

a. Death of the Nuclear Family

- Practitioners are required to attach a death certificate a maximum of seven days after the sad news occurs and proof of Family Card. The nuclear family consists of parents, older brothers and sisters.
- A follow-up permit is permitted if the time span between the incident and the practical schedule is a maximum of h+2 after the death of the immediate family.

Accident

- Practitioners are advised to document the crime scene/other related evidence of accidents as evidence for submitting follow-up practicums.
- Practitioners who do not have time to document the scene of the crime/other evidence of the accident are required to make a stamped statement accompanied by a statement of validation from the elders around the scene of the crime.

Worship

Practitioners must attach a copy of proof of dispensation approval from i-Gracias which has been approved by the Student Body

C. The flow of applications for follow-up practicum is as follows:

a. Practitioners submit follow-up practicums online to OA Line seelabs
b. The laboratory assistant checks the reasons for applying for follow-up practicum
c. The laboratory assistant determines whether the follow-up practicum application will be approved or rejected
d. Only practitioners who have been approved by the laboratory are permitted to take part in follow-up practicums
e. The laboratory assistant records follow-up practical data
f. Announcement of the list of practitioners taking part in the follow-up practicum will be made via OA Line seelabs and FTE Practicum Info on Telegram
g. The relevant laboratory assistant determines and informs the follow-up practicum schedule
h. Implementation of follow-up practicum

D. The decision regarding the application for follow-up practicum in point C by the FTE Laboratory Assistant as a representative of the Faculty of Electrical Engineering is absolute and cannot be contested.

Bandung, August 2025
Wakil Dekan 1 Fakultas Teknik Elektro,

Dr. Mamat Rokhmat, S.Si., M.T.

ELECTRIC CIRCUITS LABORATORY RULES

Objective

After reading the Electrical Circuit Laboratory Rules, students are expected to be able to:

1. Understand the regulations for practical activities.
2. Understand the rights and obligations of practitioners in practicum activities.
3. Understand the components of practicum activity assessment.

Offline Practicum Regulations

1. Practicum under the guidance of the Practicum Lecturer and assisted by Laboratory Assistants and Electrical Circuit Practicum Assistants.
2. The practicum is carried out in the Electrical Circuits Laboratory according to the specified schedule.
3. Practitioners are required to bring practicum modules, practicum cards and stationery.
4. Practitioners are required to fill in the attendance list and practicum BAP (Inspection Minutes).
5. Duration of practical activities = 2.5 hours (120 minutes).
   a. 15 minutes for the Initial Test
   b. 105 minutes for practical work followed by journal work
6. Practitioners can be absent at least 1 time from all practical lab meetings. If the total attendance is less than 75% then the Electrical Circuit Practicum value = 0.
7. Practitioners who arrive late:
   - ≤ 10 minutes: allowed to take part in practicum, but there is no additional time for completing Preliminary Assignments.
   - 10 – 20 minutes: allowed to take part in practicum with a TA=0 score
   - 20 minutes: not allowed to take part in practicals.
8. During the practicum, the practicum assistant and practicum:
   - Must wear a uniform according to Institution regulations.
   - For practice, hair must be neat (not long for students and hair tied back for students).
   - Must turn off or silence all communication devices (smartphone, tab, iPad, etc.).
   - It is prohibited to open applications that are not related to practicum.
   - It is prohibited to change computer software or hardware settings without permission.

- It is prohibited to bring food or drinks into the practicum room
- It is prohibited to give answers to other practitioners (pre-test, TP, journal, and post-test).
- It is prohibited to distribute pre-test, journal and post-test questions.
- It is prohibited to throw rubbish/anything in the practicum room.

9. Each practicum can take part in a follow-up practicum with a maximum of 1 module for one practicum.
   a. Practitioners who can take part in the follow-up practicum are only those who meet the requirements according to the Institution's provisions, namely inpatient treatment at a hospital (showing proof of hospitalization and a prescription for medication from the hospital), assignments from the Institution (showing an official letter from the Institution), or having an accident (showing a letter information from the student's parent/guardian).
   b. Requirements for follow-up practicum are submitted as soon as possible to the Laboratory Assistant of the Faculty of Electrical Engineering for administrative purposes.
10. Violations of these practicum regulations will be dealt with firmly in stages within the classroom, laboratory, study program, faculty and institution.
11. For practitioners who do not take part in the practicum for certain reasons, they can take part in a follow-up practicum if they receive approval from the Laboratory Assistant.

Practical Assessment

1. One practicum course at the Faculty of Electrical Engineering consists of 12 practicum modules with an assessment percentage according to the number of modules for each practicum course given.
2. In particular, the practicum assessment system for students taking practicum courses at the Faculty of Electrical Engineering follows the following rules:

   a). First Condition

   Practical courses consist of:

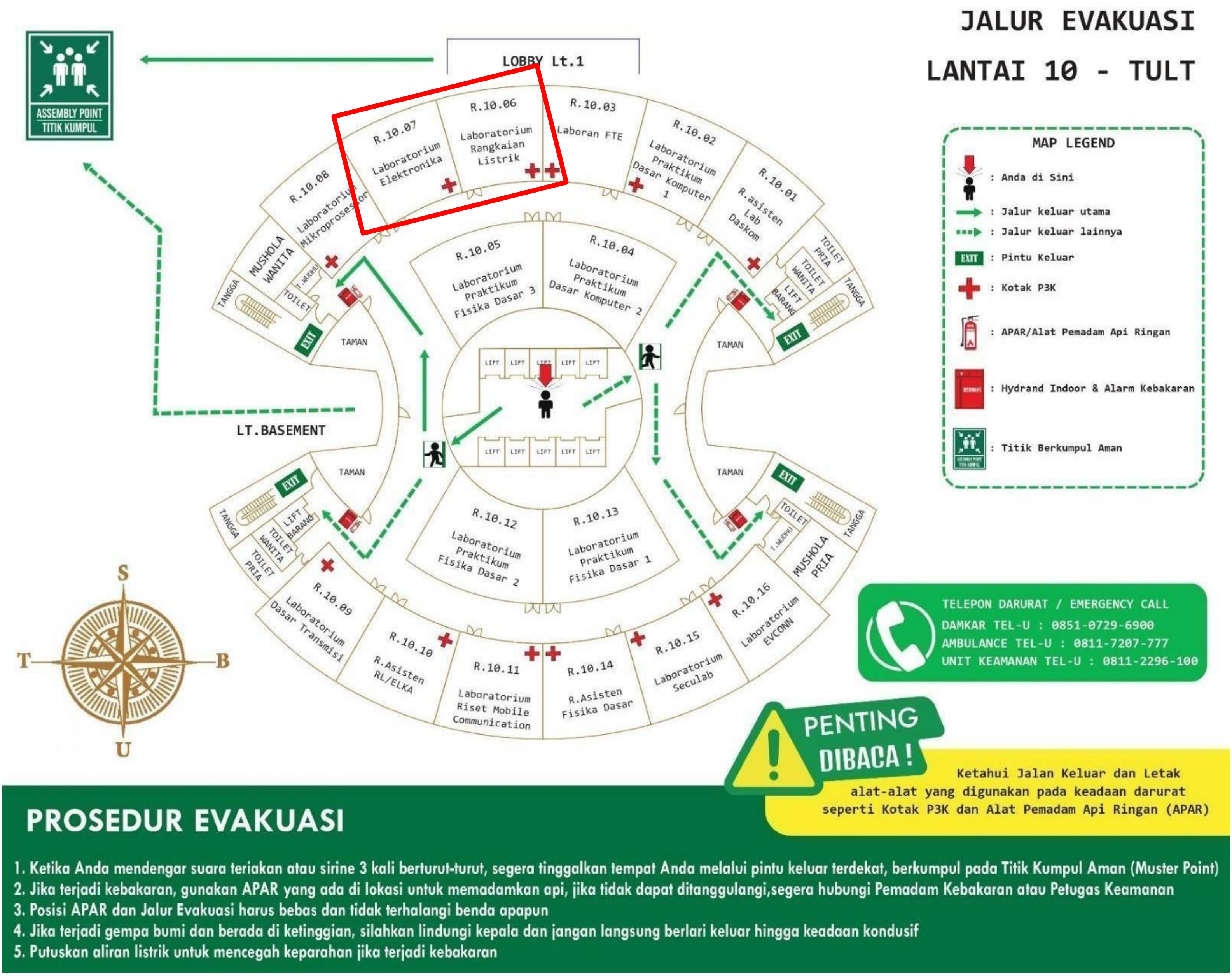
   a. 1 practicum course, totaling 12 practicum modules
   b. 2 practicum courses, each with 6 practicum modules
   c. 3 practicum courses, each with 4 practicum modules
      - Students are given tolerance for not participating in the Practicum, only 1 (one) meeting in the entire practicum
      - If the absence exceeds the points above, the grade for the practicum course is worth

b). Second Condition

The practicum course consists of 4 practicum subjects, each with 3 practicum modules

- Students are required to attend all Practicum meetings
- If you do not follow the provisions above then the value of the practicum course is worth

Lab Location Plan



GENERAL SAFETY AND GUIDELINES

Electrical Hazards

1. **Introduction and Supervision of Electricity Sources**
   - Before starting the practicum, the practitioner must identify the location of the sockets and circuit breakers in the laboratory. This is important so that practitioners know where the power source is and how to operate or turn it off if necessary.
   - Practitioners must learn how to turn on and off power sources correctly to avoid the risk of operational errors that can be dangerous.
   - If there is damage to electrical installations or equipment that has the potential to cause danger, immediately report it to the assistant. Do not try to repair it yourself as this may increase the risk of an accident.

2. **Electric Shock Prevention**
   - Avoid contact with areas or objects that have the potential to cause electric shock, such as frayed cables or damaged equipment.
   - Do not perform actions that could endanger yourself or others, such as handling electrical equipment with wet hands.
   - Make sure that wet body parts, such as from sweat or leftover ablution water, are dry before touching electrical equipment.
   - Always be alert to potential electrical hazards in every practical activity. Pay attention to the surrounding environment and ensure all tools are in good condition before use.

3. **Emergency Actions When Electric Shocked**
   - Do not panic. Panic can worsen the situation and make rescue actions more difficult.
   - Immediately turn off all electronic equipment and power sources around the table affected by the shock to further reduce the risk.
   - Help the practitioner who has been shocked to remove himself from the electrical source safely. Use insulating material, such as wood or plastic, to cut off the victim's contact with the electrical source.
   - Immediately report the incident to an assistant or someone nearby to get medical assistance and further action.

Danger of Fire or Excessive Heat

1. **Fire Prevention**
   - Do not bring flammable objects (matches, gas, etc.) into the practical room unless absolutely necessary.
   - Avoid actions that could cause fire, sparks or overheating, such as using tools without supervision or placing flammable objects near hot equipment.
   - Always be aware of the danger of fire in every practical activity. Watch for early signs of a potential fire, such as the smell of burning or smoke.

2. **Emergency Actions When Fire Danger Occurs**
   - Do not panic. Remain calm and focus on rescue steps.
   - Immediately report the incident to an assistant or bystander for help.
   - Turn off all electronic equipment and electrical sources in the practical area to avoid the spread of fire.
   - Evacuate yourself from the practicum room in an orderly manner, following the designated evacuation route. Do not use the elevator in a fire emergency.

Use of Practical Equipment

1. **Instructions for Using the Tool**
   - Before using practical tools, understand the instructions for use provided. Read the manual or instructions carefully.
   - Pay attention to and obey the warnings which are usually printed on the body of the tool. This warning is important to avoid usage errors that could be dangerous.
   - Understand the function or purpose of practical tools and use these tools only for activities that are appropriate to their function. Using a tool beyond its intended purpose can cause damage and be a safety hazard.

2. **Equipment Maintenance**
   - Understand the rating and working range of practical tools and use these tools according to their specifications. Using a tool outside its rating may cause damage and danger.
   - Make sure that all practical equipment used is safe from sharp objects or metal, fire, excessive heat, or other conditions that could damage the equipment.

- Do not carry out activities that can cause dirt, streaks, scratches, or the like on the body of the practical equipment

3. **Damage Liability**
   - Damage to equipment is the collective responsibility of the practicum group concerned. Damaged equipment must be replaced by the group.
   - If damage occurs, immediately report it to the assistant and coordinate the replacement of the damaged equipment to ensure the smooth running of the next practicum.

<u>Steps if an Earthquake Occurs</u>

1. **Avoid Panic and Try to Be Calm**
   - If an earthquake occurs, it is important to stay calm. A calm attitude will allow us to think clearly about what actions to take.

2. **Use the Emergency Staircase**

   If you use the emergency stairs, there are several things you need to pay attention to.
   - First, hold on to the side of the ladder. Second, don't run. Running can increase the risk of falling when going down stairs.
   - Remind women such as friends or lecturers to remove high-heeled shoes because they can be dangerous.

3. **Don't Use the Elevator**
   - Never use the elevator in the event of an earthquake. Earthquakes can trap us in elevators.
   - If you feel an earthquake while you are in the elevator, immediately press all the buttons and go to any floor.
   - Once the door opens, immediately find a place to take cover. However, if the door cannot be opened, press the emergency button and contact the building staff via the interphone in the elevator provided.

4. **Take refuge from the ruins in the building**
   - If you are in a tall building and cannot get out of the building immediately, immediately protect your body from the rubble.

- Friends must take cover under a table, or in a strong corner of the room such as a wall.
- Avoid objects that can fall such as windows, cupboards, or other items.

<u>Steps if a Fire Occurs</u>

1. **Prevent fires**
   - Do not light matches and the like without an assistant's command. When you want to light it, don't be close to flammable materials such as books, curtains, etc
   - Avoid installing too many electronic devices in one extension cable.

2. **How to deal with fire**
   - Try to stay calm (don't panic) when a fire suddenly breaks out around your friends.
   - If the fire can still be controlled using an APAR, you should immediately use this tool or you can also use a burlap sack/cloth that has been moistened with water. Small-scale fires can usually be overcome using these tools, but make sure you do it immediately so that the fire doesn't spread further.
   - If it turns out that the fire was caused by an electrical short circuit, you should immediately turn off the main electricity switch.
   - Immediately close the room that is experiencing fire so that it does not spread to other rooms. It is best not to lock the burning room so that it is easier for officers to extinguish the fire.
   - If the fire that occurs is a large-scale fire, you should immediately save yourself and immediately try to leave the building. Avoid being busy saving valuables because this can actually endanger your friends' lives.
   - When a fire occurs, try not to inhale the thick smoke that emerges from the fire. Friends can walk on all fours and breathe by lowering their heads to the floor. Apart from that, use a wet cloth to cover your nose so it will be easier to breathe.
   - Look for a way through the emergency stairs and also pay attention to pressing the back of your palm against the door to know the temperature. Apart from that, hold the door handle. If it feels hot, you should go to another emergency stairwell that is safe from the flames.
   - Immediately contact the fire brigade by calling 112 if the fire starts to become difficult to control.

Table of Contents

LIST OF IMAGES

LIST OF TABLES

INTRODUCTION TO CIRCUIT SIMULATOR (CS) APPLICATIONS

Circuit Simulator is a software released by Java that has a function to simulate the circuit that will be made hardware. Circuit Simulator at the time of release was in the form of version 2.8.1js. This software was created by Paul Falstad and serves to simulate various kinds of electrical and electronic circuits. This application is run based on JavaScript for free and can be run based on a web browser. This software is available in online and offline forms that can be installed standalone. Circuit Simulator uses a mathematical model approach to represent the behavior of electronic devices or circuits that actually occur.

TUTORIAL ON INSTALLATION AND OFFLINE ACCESS OF CIRCUIT SIMULATOR PROGRAM FOR LAPTOP / PC

1. Open the https://falstad.com/circuit/offline/ link
2. Select the file to be downloaded according to the *Operating System.*

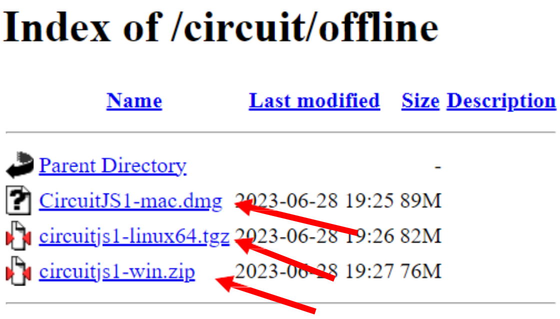


figure 0. 1: Choose the file according to the operating system

3. Right-click on the downloaded file > *Extract here.*

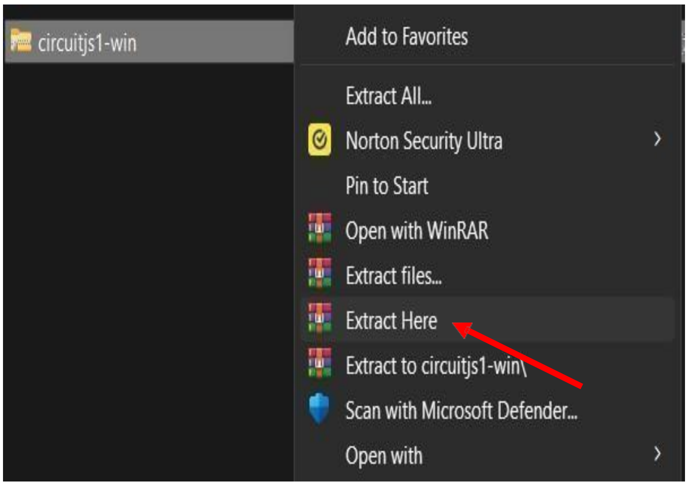


figure 0. 2"Extract Here" Locations

4. Open the *extracted file* > open *Software circuitjs1.*

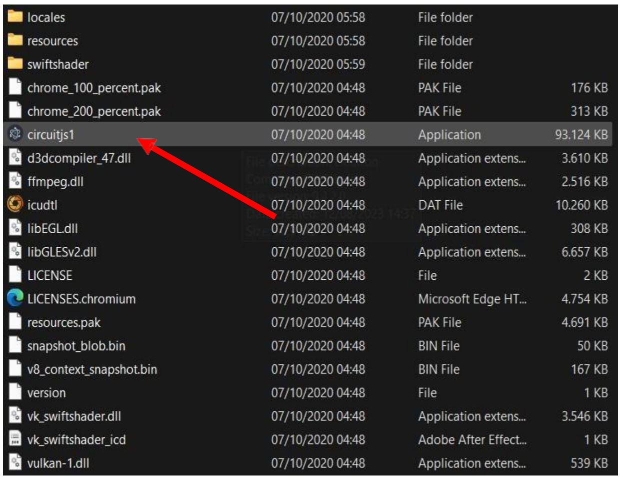


*figure 0. 3****: Software circuitjs1***

5. The *default* view can be seen in the image below.

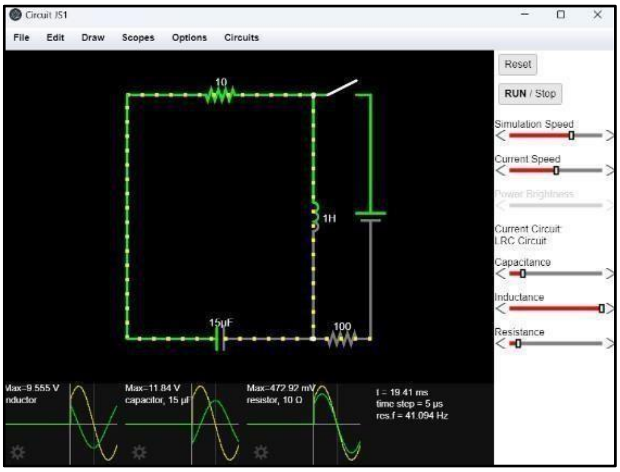


*figure 0. 4: **Default Circuit Simulator***

6. To open a new worksheet, select *File > New Blank Circuit.*

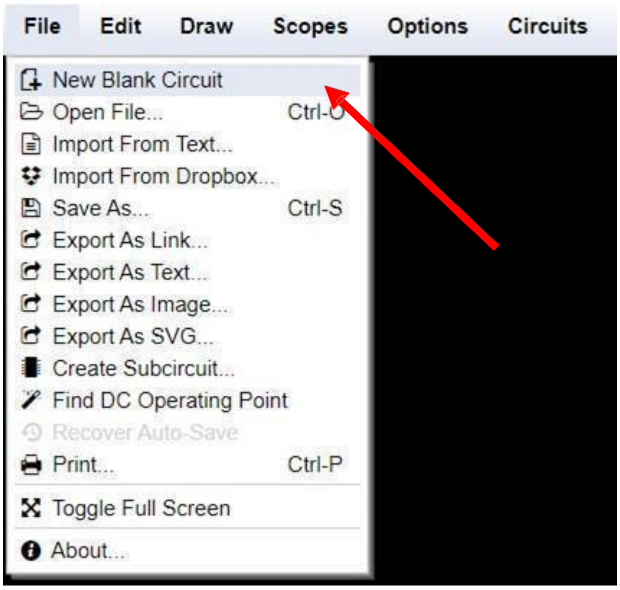


figure 0. 5: New Blank Circuit

7. Then the display will be like this.

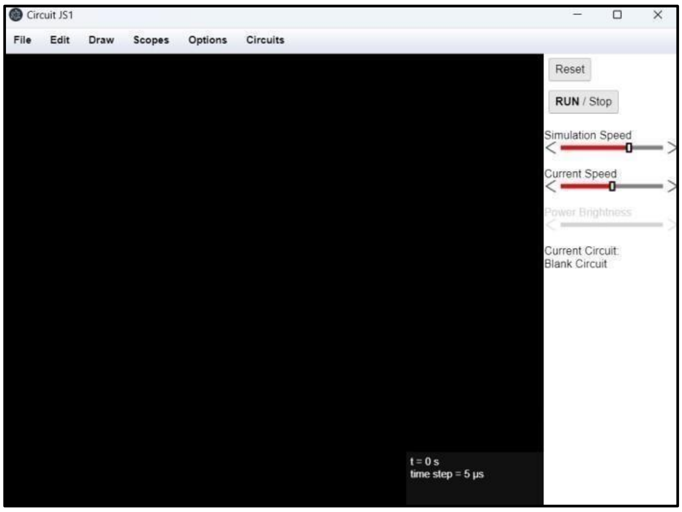


figure 0. 6: Display after "New Blank Circuit"

TUTORIAL *ON ONLINE* ACCESS ***TO CIRCUIT SIMULATOR* PROGRAM ON SMARTPHONES**

1. Smartphone settings in the form of landscape/auto rotate
2. Visit [the https://www.falstad.com/circuit/circuitjs.html](https://www.falstad.com/circuit/circuitjs.html) link via the internet browser on the smartphone or scan the following QR code:

3. Make the Circuit Simulator screen full screen by means of File > Toggle Full Screen.

figure 0. 7: Full screen display

4. To create a blank worksheet, click File > New Blank Circuit.

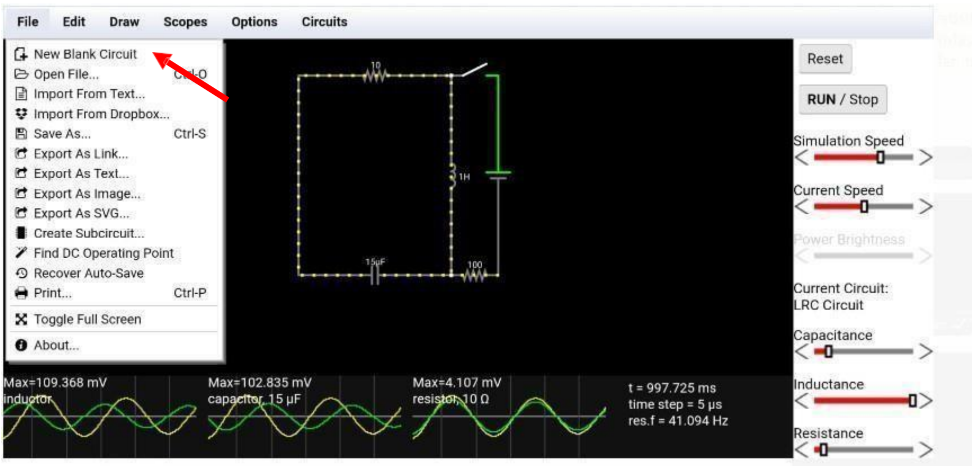


figure 0. 8: "New Blank Circuit" Menu

5. Start creating a series by selecting the Draw menu. All components in CS are contained in the Draw menu. Make sure every time you finish dragging the component on the schematic to release the component, it must return to the Draw menu --> Select / Drag Cell. Use the Select/Drag Cells menu to move, extend/shorten components and rotate components

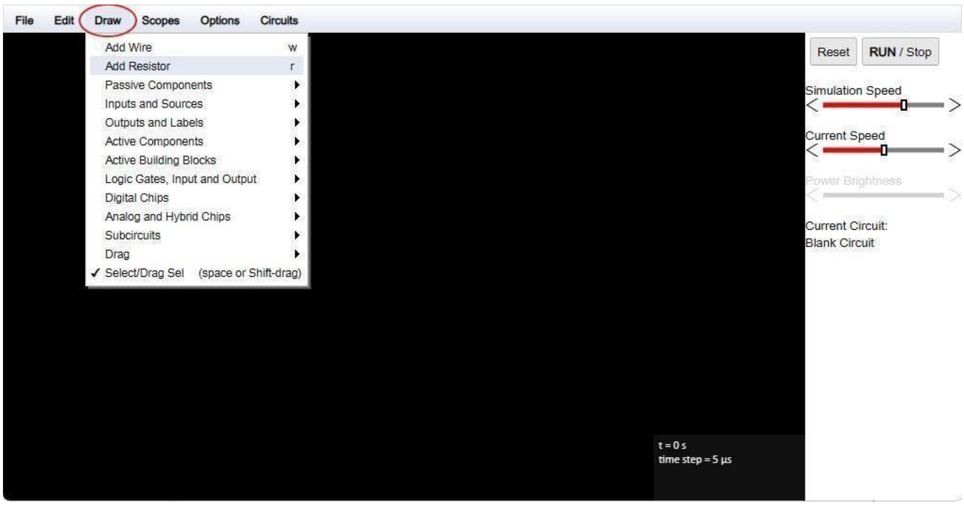


figure 0. 9: "Draw" menu

6. To change the size of a component by double-tapping on it

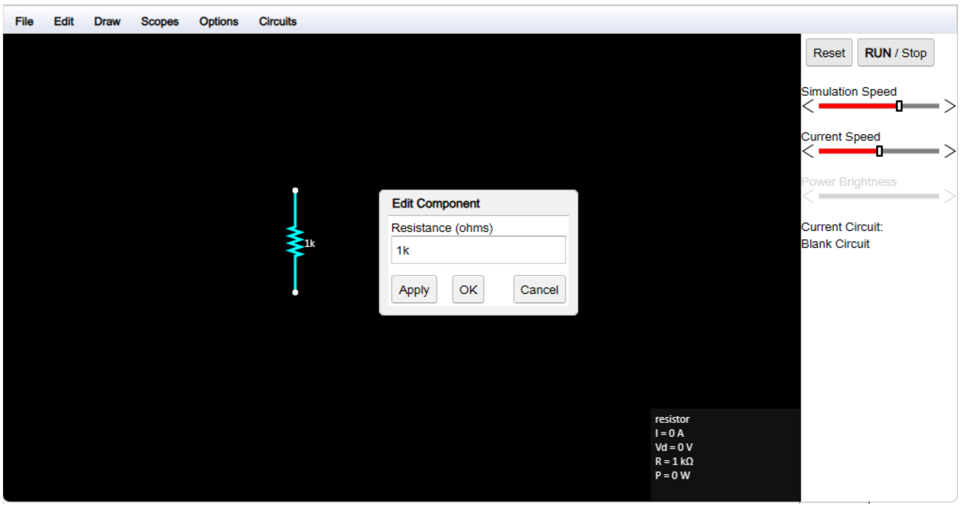


figure 0. 10: Changing the Component Magnitude

or hold for some time on the component so that it appears as shown below then select Edit

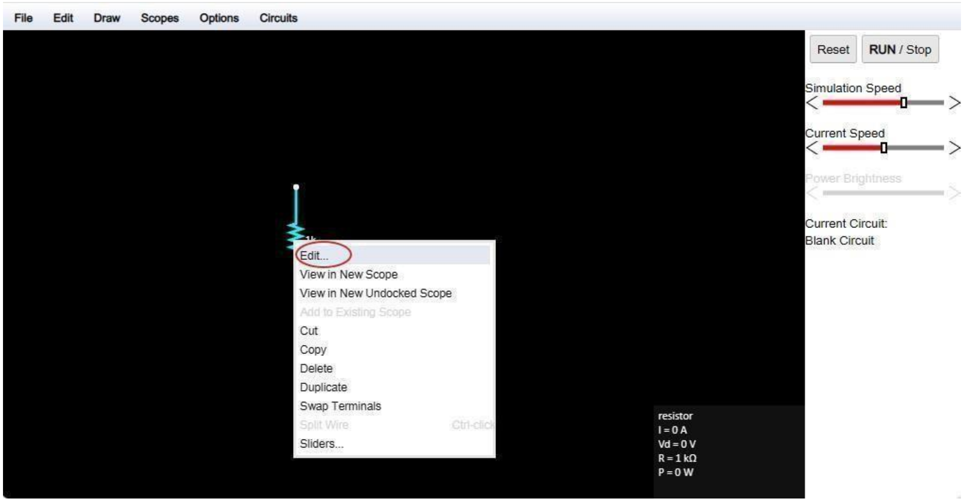


figure 0. 11: "Edit" menu

7. To connect between components by clicking Draw --> Add Wire, make sure to connect the end of the component with the other end of the component (it is not allowed to pull the wire directly at once, resulting in bad connection / red dot, for that the wire must be one by one end to end).

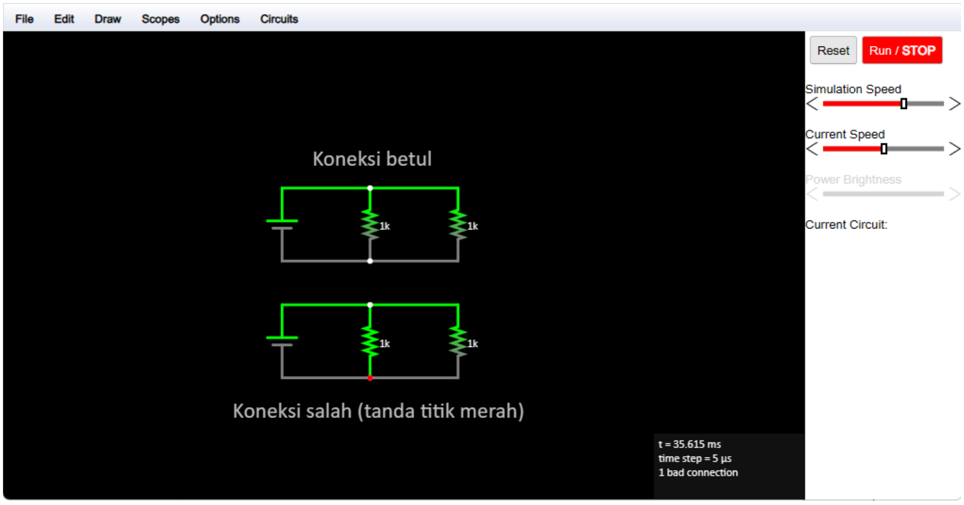


figure 0. 12: Connecting components

8. To save the series, click File > Export As... (desired form of document). Example: If the file is saved in text form. Click File > Export As Text.

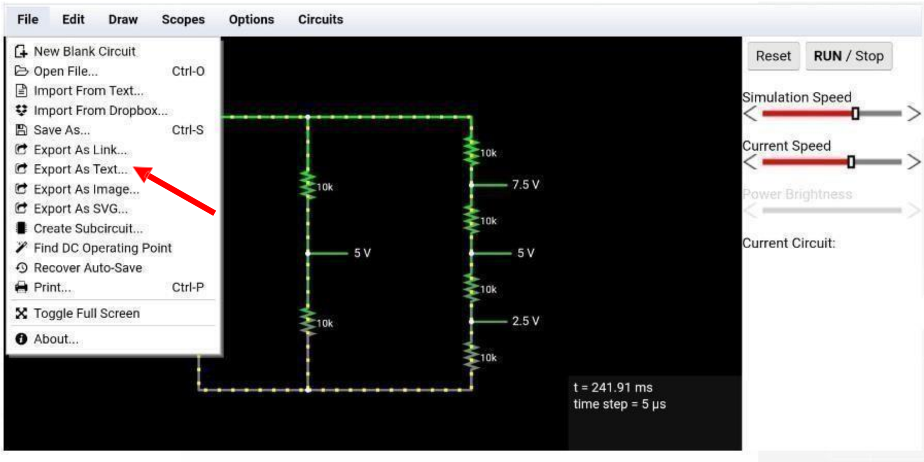


figure 0. 13: "Export As Text" Menu

Then click Copy to Clipboard > OK, then save the file in the form of .txt on the smartphone.

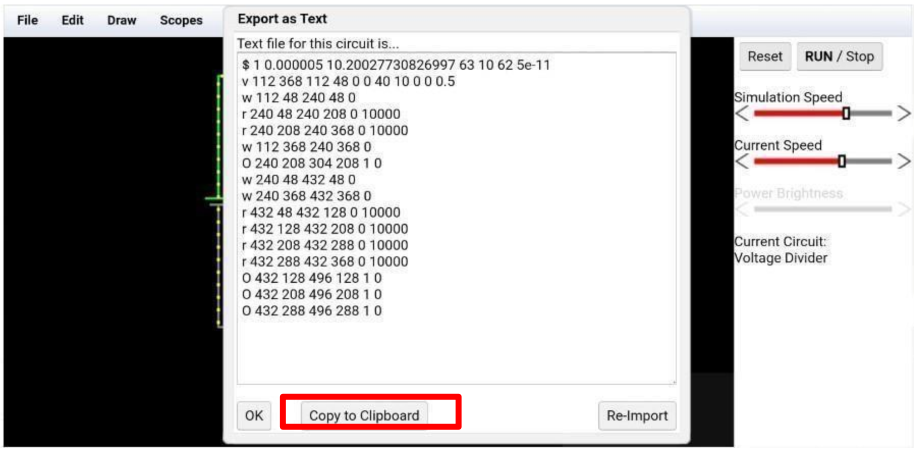


figure 0. 14: Save the ".txt" file

9. To import a text file, click File > Import From Text.

figure 0. 15: "Import from Text" menu

Enter text from the copied file > OK.

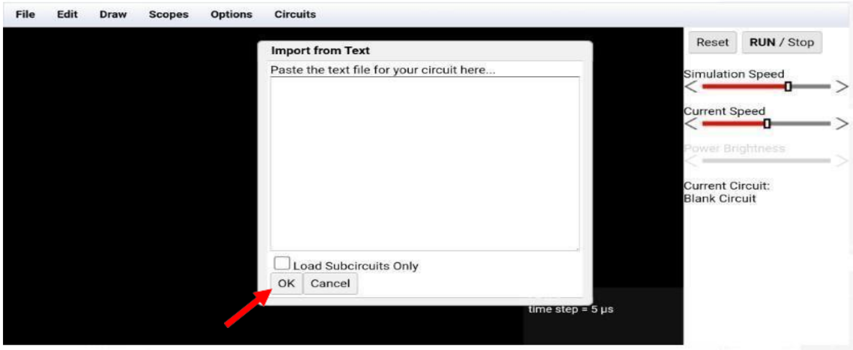


figure 0. 16: Insert a file from copied text

10. The next way to open a saved .txt file, click File --> Open file then find the form file .txt

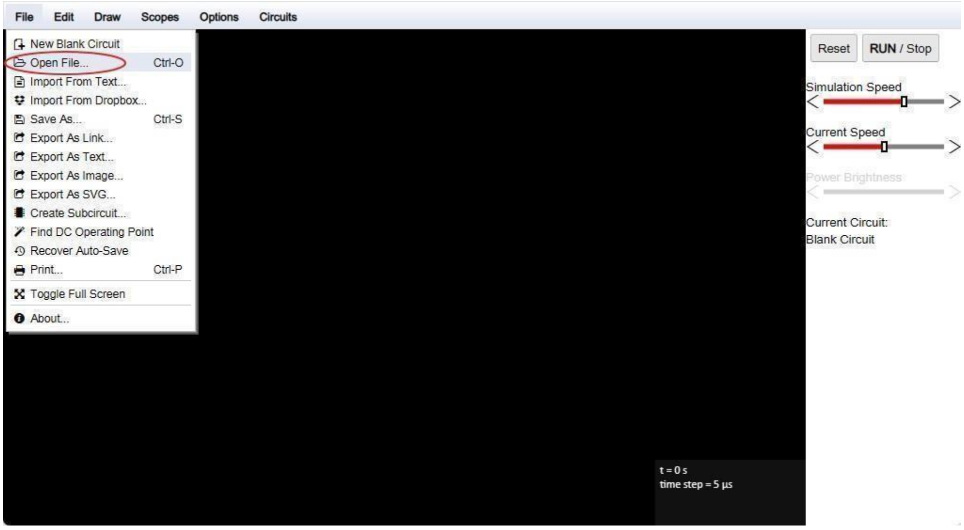
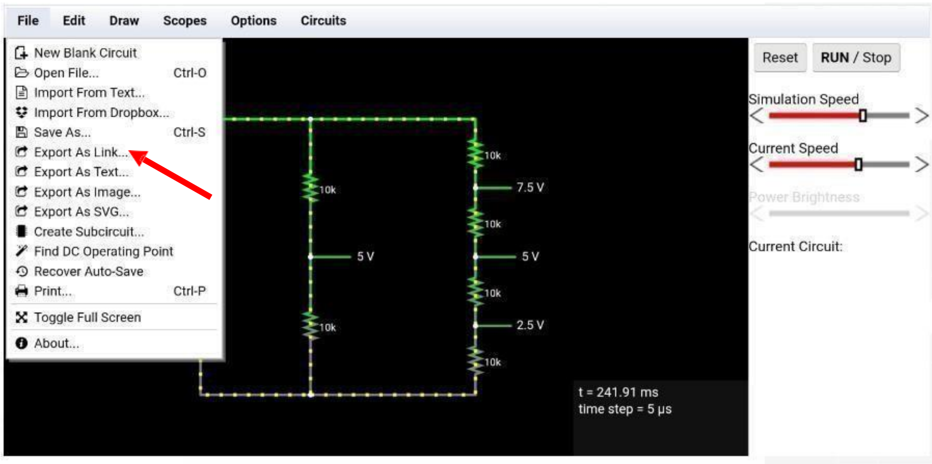


figure 0. 17: Open old files

11. Files can be saved in the form of links by clicking File > Export As Link. Links can be shortened. Links can be saved in notepads or other document applications.



.

figure 0. 18: Open a file as a link

figure 0. 19: Text form concatenation file.

Then a series file will appear that has been saved in the form of teks. To open a file in the form of a url link, double-tap or copy-paste it into a web browser.

CIRCUIT SIMULATOR MENU GUIDE

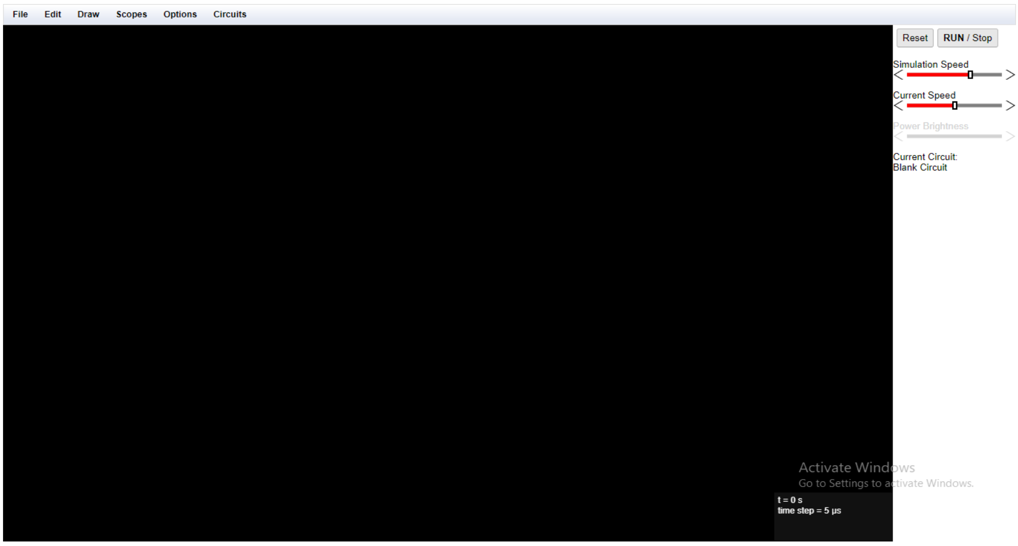


figure 0. 20: Main view of Circuit Simulator

File Edit Draw Scopes Options Circuits	
New Blank Circuit	Create a new *file*
Open File... Ctrl-O	Open an existing *file*
Import From Text...	Import *files* from .txt

Import From Dropbox...	Import *files* from Dropbox
Save As... Ctrl-S	Save *a file*
Export As Link... Export As Text... Export As Image... Export As SVG...	Export *files* in the form of Links, Text, Photos, and SVG
Create Subcircuit...	Create a subcircuit model
Find DC Operating Point	For circuits that take a long time to use. Usually used to charge capacitors
Print... Ctrl-P	Print *a file*
Toggle Full Screen	Make the circuit simulator view full *screen*
About...	Program description

Table 0. 1 Circuit Simulator "File" Menu

File Edit Draw Scopes Options Circuits	
Undo Ctrl-Z	Undo a previous action
Redo Ctrl-Y	Reverts back a previously undone action.
Cut Ctrl-X	Cut/move selected elements
Copy Ctrl-C	Copy selected elements
Paste Ctrl-V	Paste cut/copied elements
Duplicate Ctrl-D	Duplicate selected elements
Select All Ctrl-A	Mark all elements on a worksheet
Find Component... /	Search for components
Centre Circuit	Place all components in the center of the worksheet
Zoom 100% 0	Return worksheet size to *default*
Zoom In +	Zoom in on objects
Zoom Out -	Zoom out of objects

Table 0. 2 Circuit Simulator "Edit" Menu

File Edit **Draw** Scopes Options Circuits	
Add Wire w	Add cables to connect between components
Add Resistor r	Adding resistor components
Passive Components	Add passive components
Inputs and Sources	Add *input* and source components
Outputs and Labels	Add *output* components and assign labels
Active Components	Add active components
Logic Gates, Input and Output	Add logic gates, *inputs*, and outputs.
Digital Chips	Digital chip menu
Analog and Hybrid Chips	Menu of analog and hybrid chips
Subcircuits	Add *ground*
Drag	Shift components
Select/Drag Sel (space or Shift-drag)	Selecting components

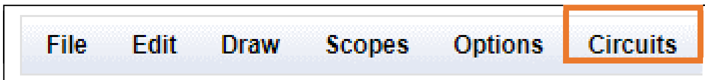
Table 0. 3 Menu "Draw" Circuit Simulator

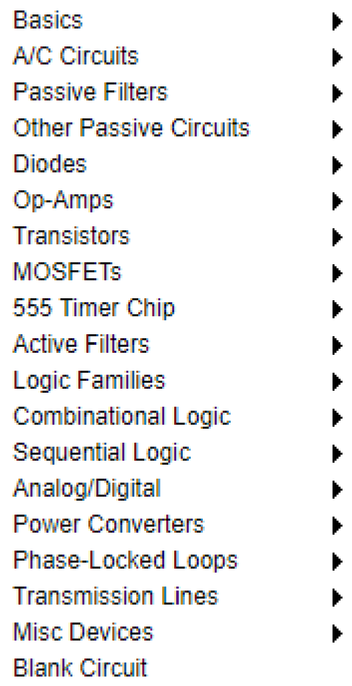
File Edit Draw **Scopes** Options Circuits	
Stack All	Unify simulation results in one horizontal graph (inductors, capacitors, resistors)
Unstack All	Separate simulation results one by one (inductors, capacitors, resistors)
Combine All	Consolidate simulation results into one graph
Separate All	Separating simulation results into three parts (inductors, capacitors, resistors)

Table 0. 4 Circuit Simulator "Scopes" Menu

File Edit Draw Scopes Options Circuits	
Show Current	Displays a flow illustration
Show Voltage	Display voltage
Show Power	Display power
Show Values	Display values on components
Small Grid	Close grid lines to make them denser
Show Cursor Cross Hairs	Display horizontal and vertical line crossover
European Resistors	European resistor symbol mode
IEC Gates	Display IEC standard logic gate symbols
White Background	Change the background to white
Conventional Current Motion	Displays conventional current movement
Disable Editing	Turn off editing
Edit Values With Mouse Wheel	Features editing component values by scrolling mouse
Shortcuts...	Display shortcuts and can add shortcuts on the Shorcuts menu
Other Options...	Customize settings

Table 0. 5 Circuit Simulator "Options" Menu





The Circuits menu can be used to view some interesting pre-defined circuits. Once the circuit is selected, you can later modify it as you wish.

For example, if you select the *Basics* menu there will be options:

a) Resistors: these denote multiple resistors of various sizes in series and parallel.
b) Capacitor: this indicates a capacitor that can be charged and discharged by clicking a switch.
c) Inductor: it shows an inductor that can be charged and emptied by clicking a switch.
d) LRC circuit: this shows the circuit oscillating with inductors, resistors, and capacitors. It can close the switch to drive the current in the inductor, and then open the switch to see the oscillations.
e) Voltage Divider: it indicates a voltage divider, which produces 7.5V, 5V, and 2.5V reference voltages from a 10V power supply.
f) Thevenin's theorem states that the circuit above is equal to the circuit below.
g) Norton's theorem states that the circuit above is equal to the circuit below.

	More details can be seen at the following link. https://www.falstad.com/circuit/directions.html

TUTORIAL ON CREATING A SIMPLE CIRCUIT

Make a simple electrical circuit as shown in the following picture. The resistor model used in this simulation is a European or IEC model, do the settings on the *Option menu* --> *checklist* on *European Resistors.*

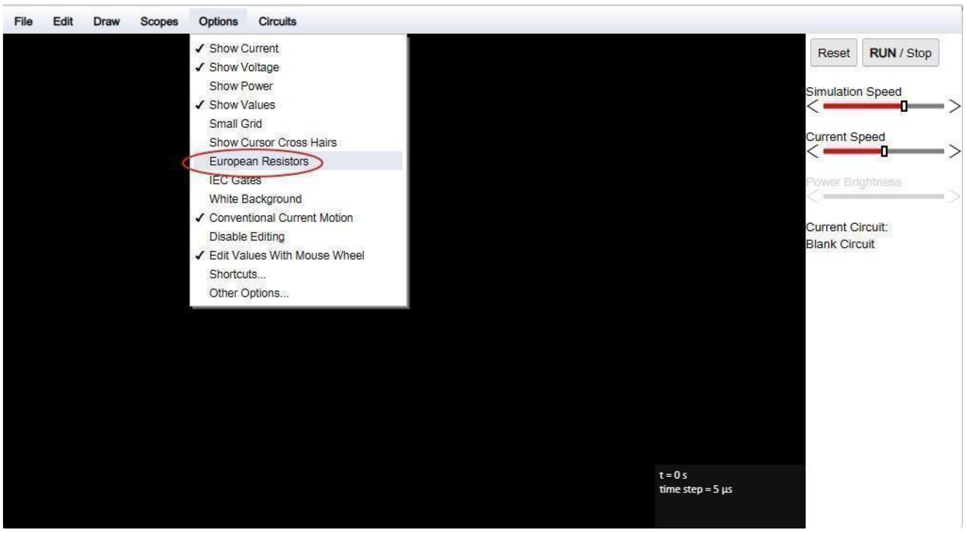


figure 0. 21: "European Resistor" menu.

Then the schematic will be simulated as follows:

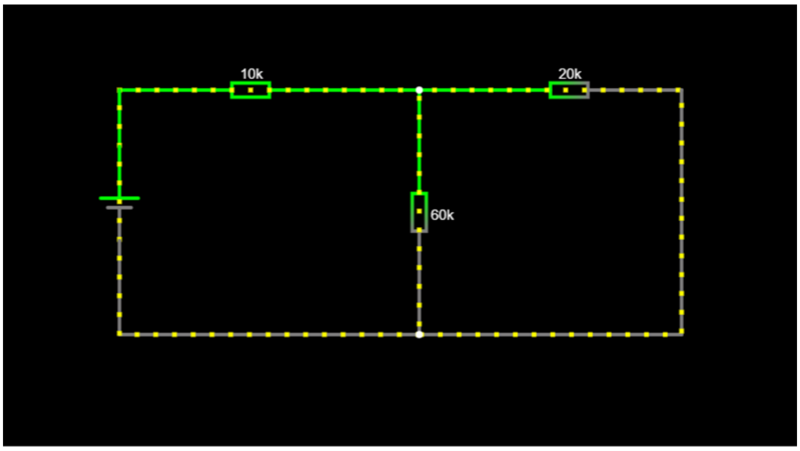


figure 0. 22: Schematic example of a circuit

1. Create a blank worksheet by clicking *File --> New Blank Circuit*, add resistors by clicking Draw > *Add Resistor.*

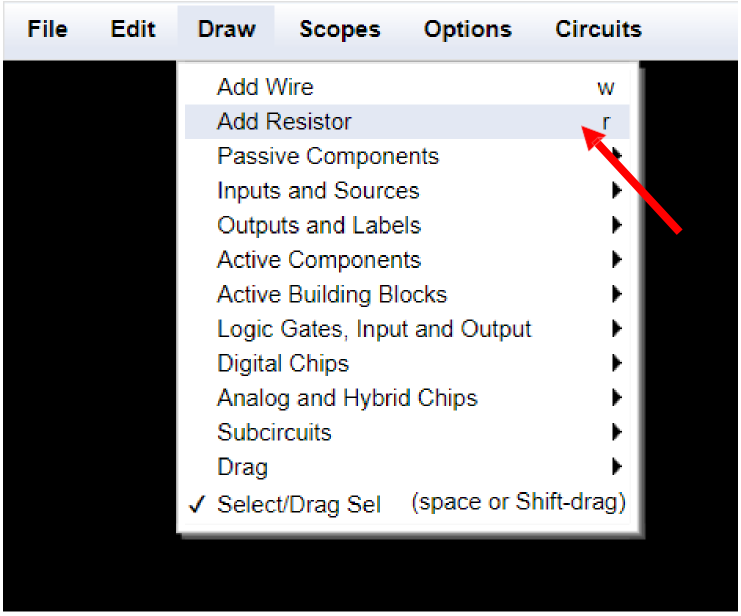


figure 0. 23: Adding resistors

2. Drag the *mouse* cursor in the vertical and horizontal directions.

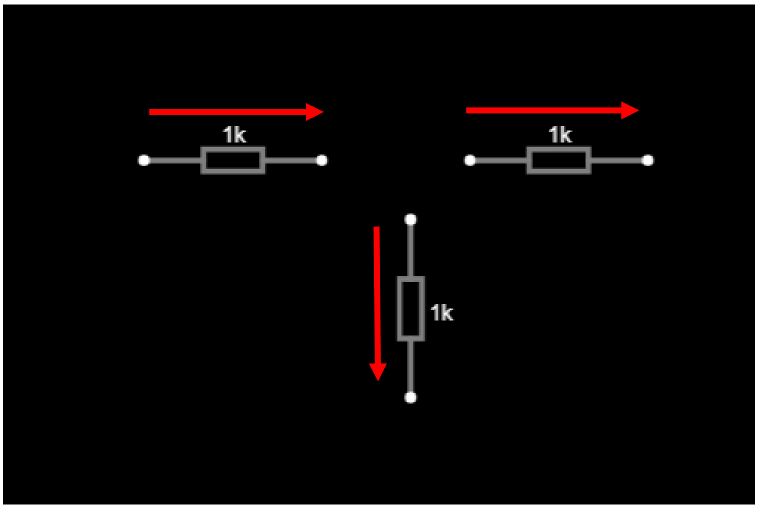


figure 0. 24: Resistor position.

3. Add a voltage source of 2 terminals by clicking *Draw > Input and Sources > Add Voltage Source (2 Terminals).*

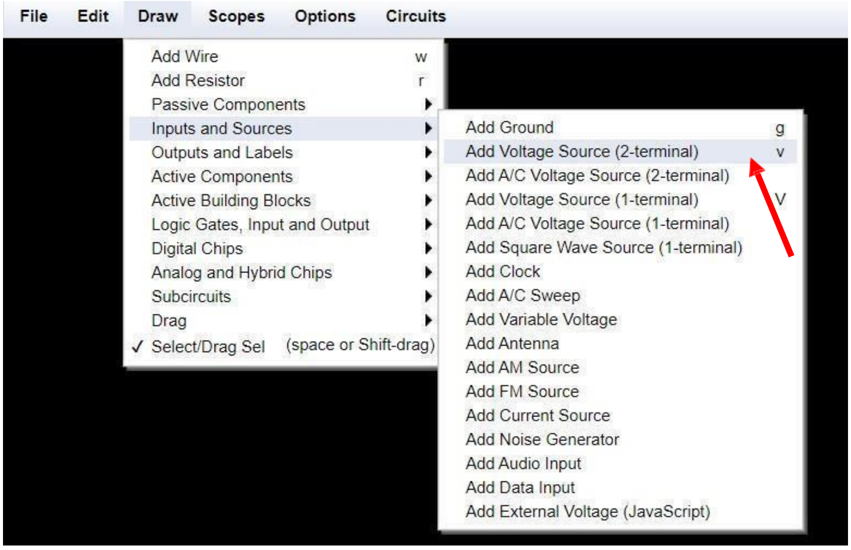


figure 0. 25: Add a voltage source.

4. Drag the cursor vertically.

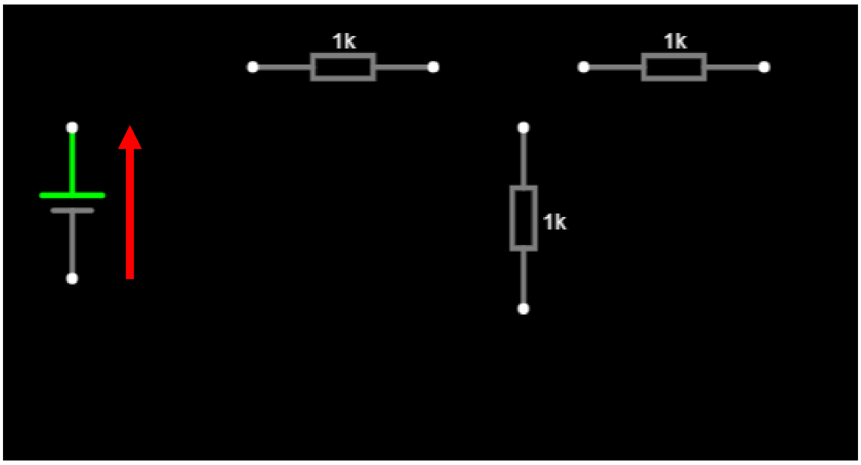


figure 0. 26: Position of voltage source

5. Select the "*Draw*" menu then click "*Add Wire*".

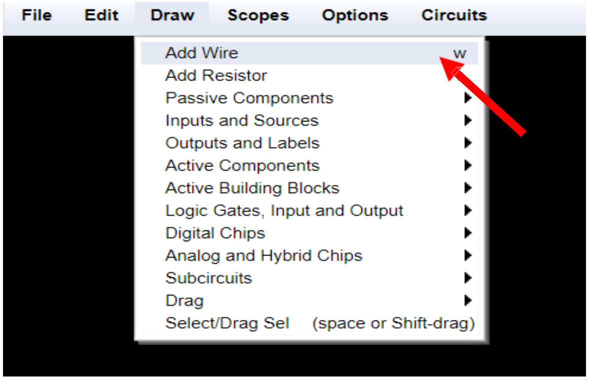


figure 0. 27: "Add Wire" Menu

6. Connect between components at the ends of the components.

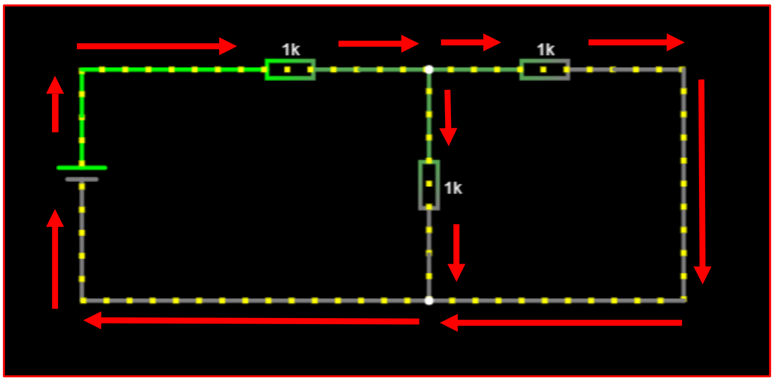


figure 0. 28: Connecting components

7. Change the value of the resistor and voltage source according to the example, by *double* clicking on the component or right-clicking the component > *edit*.

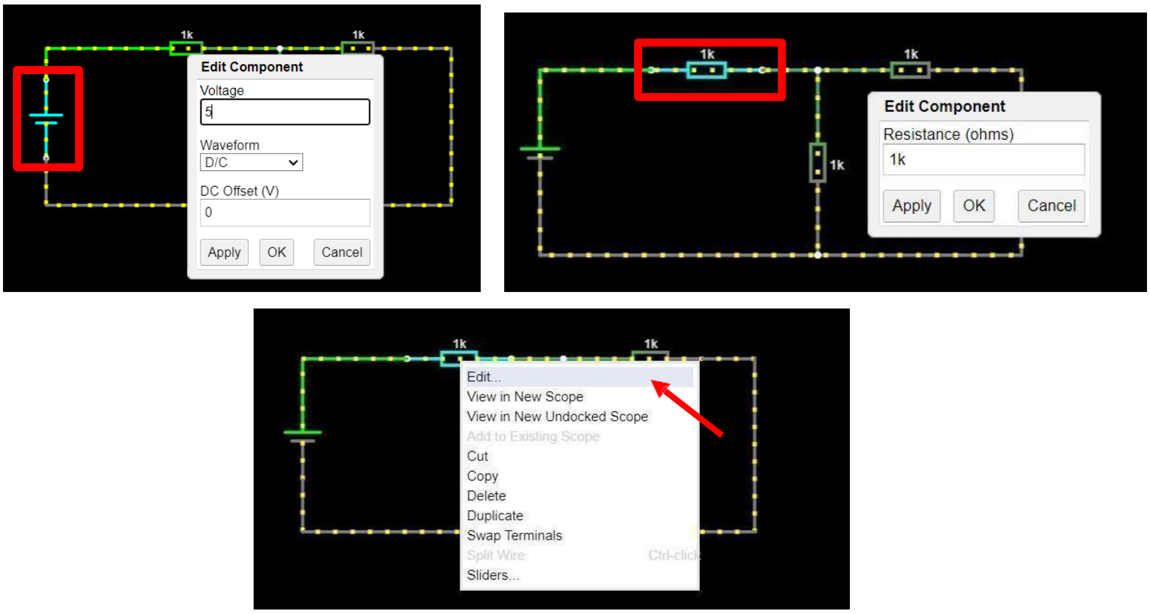


figure 0. 29: Set of tutorial images changing component values

Information:

*R1 = 10k Ω, R2 = 20k Ω, R3 = 60k Ω

*Source voltage = 10 V

8. Then the display will be like this.

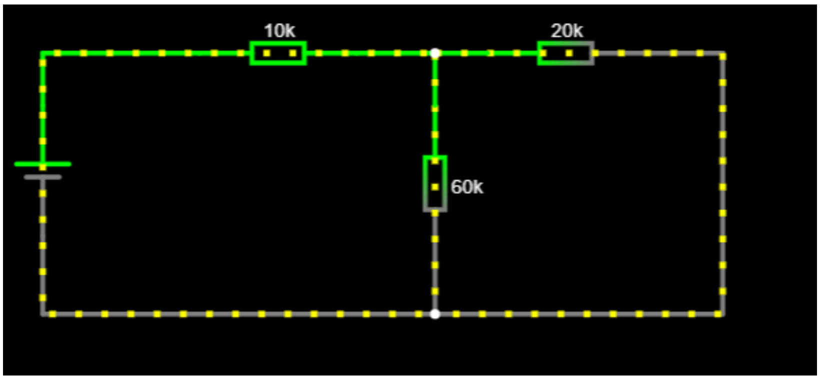


figure 0. 30: End of series view

VOLTAGE MEASURING TUTORIAL

Make a simple electrical circuit to measure voltage as shown in the following figure.

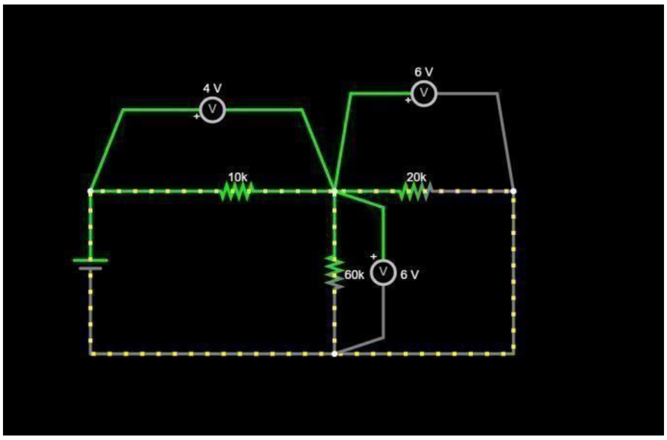


figure 0. 31: ***Voltmeter circuit***

1. Select *Draw > Outputs and Labels > Add Voltmeter/Scope Probe*.

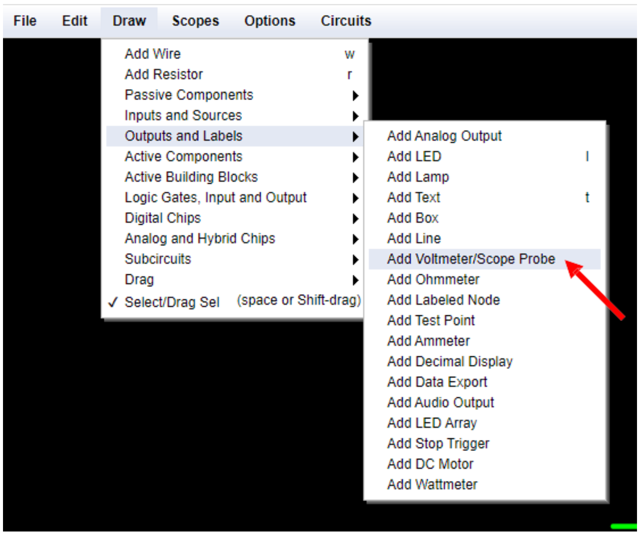


figure 0. 32: Menu to display the voltmeter

2. Add *a voltmeter* to the circuit.

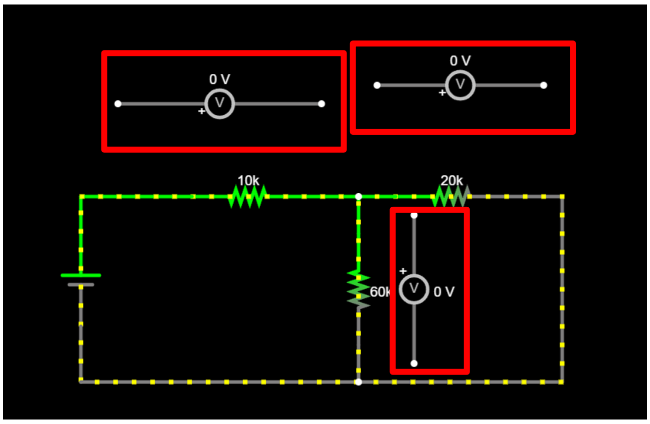


figure 0. 33: Adding a voltmeter to the circuit

3. *Add wire* to the *voltmeter*, so that the display looks like the following image.

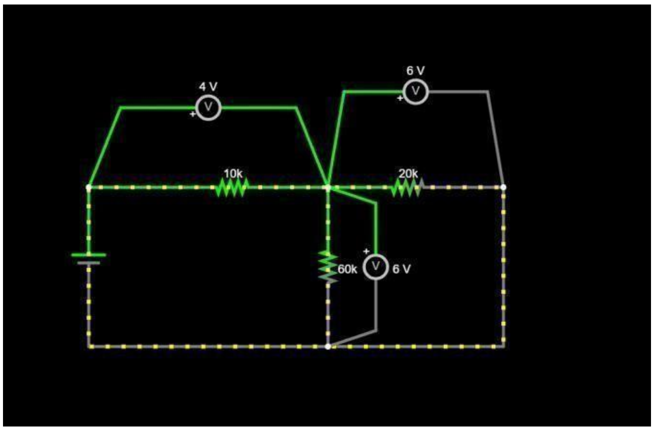


figure 0. 34: Final view of voltmeter circuit

CURRENT MEASURING TUTORIAL

1. Add an Ammeter component, by clicking *Draw > Outputs and Labels > Add Ammeter.*

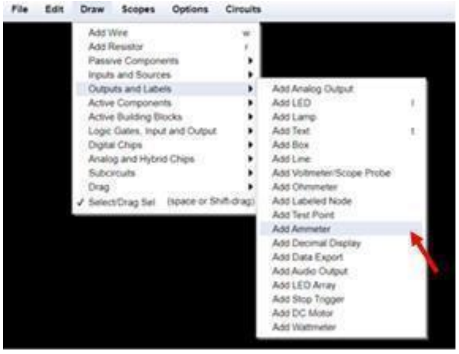


figure 0. 35: Ammeter component menu

2. Place *the Amperemeter* in series with the circuit.

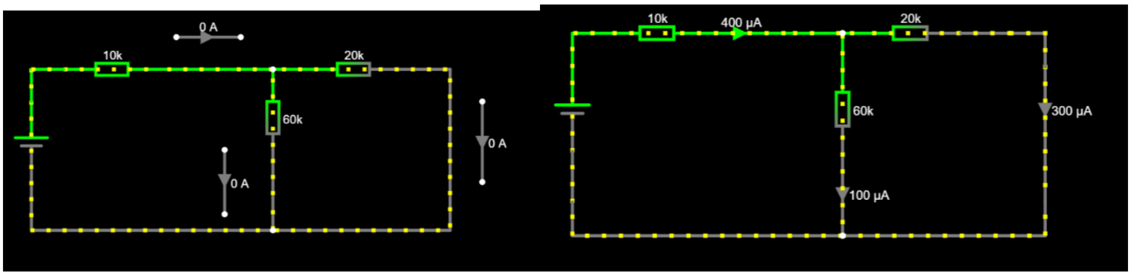


figure 0. 36: Ammeter position

TUTORIAL ON MEASURING TOTAL RESISTANCE

Make a simple electrical circuit to measure total resistance as shown in the following figure.

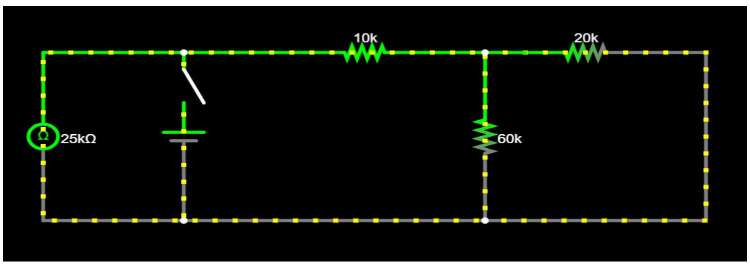


figure 0. 37: Ohmmeter circuit

1. Select *Draw* > *Outputs and Labels* > *Add Ohmmeter*.

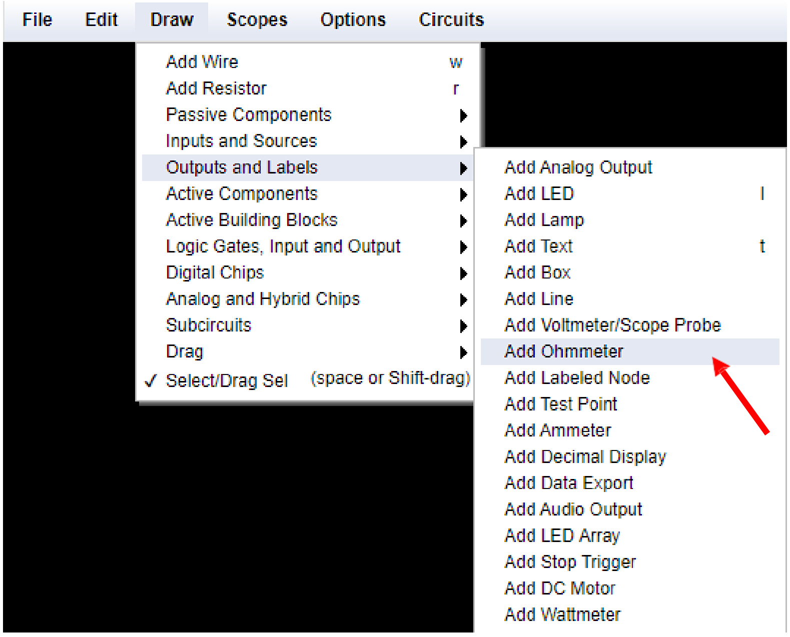


figure 0. 38: "Add Ohmmeter" Menu

2. Add *Ohmmeter* to the circuit.

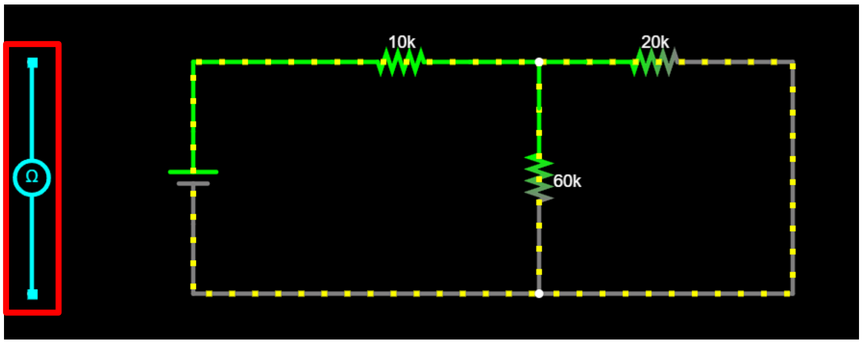


figure 0. 39: Adding an ohmmeter to the circuit

3. *Add wire* to *ohmmeter*.

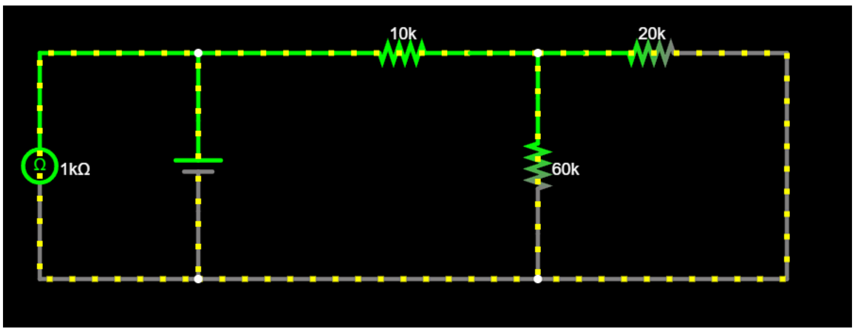


Figure 0.40 Wiring Ohmmeter

4. Add *a switch* and disconnect the *wire*, so that it looks like the following image.

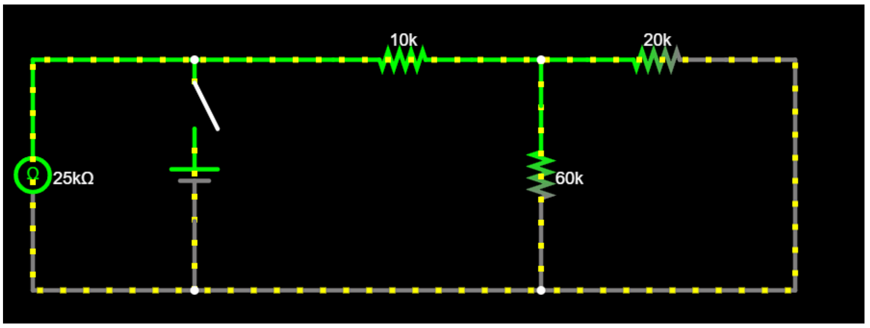


figure 0. 41: Addition of switches

Modul 1: Component Recognition, Reading, and Measurement

1.1 Purpose

After participating in this practicum, students are expected to:

1. Recognize and analyze the components of electrical circuits.
2. Understand how to take readings and measurements on electrical circuit components.

1.2 Tools and Materials

1. Resistor
2. Potentiometer
3. Trimmer Potentiometer (TRIMPOT)
4. Thermistor (PTC-NTC)
5. LDR
6. Project Board
7. Multimeter
8. Falstad Circuit Simulator

1.3 Theoretical Basis

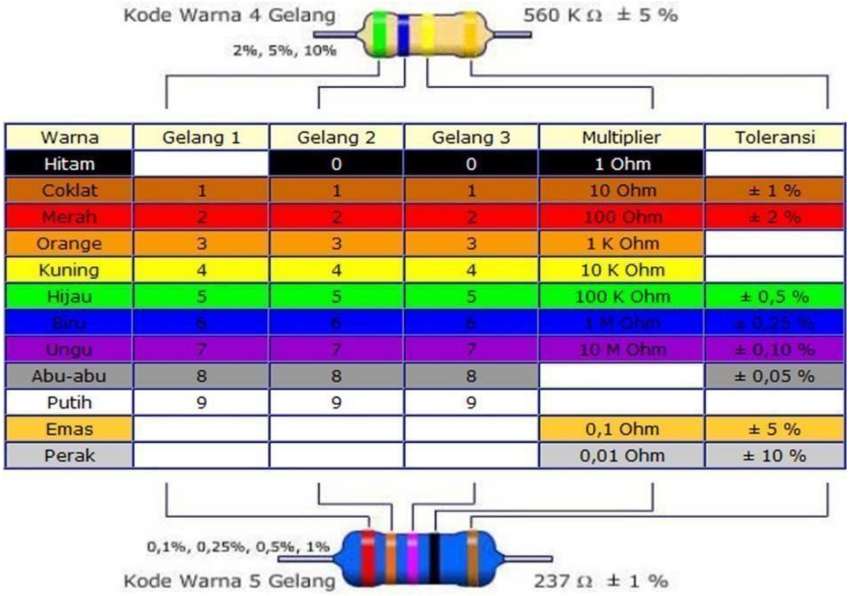
1.3.1 Component

A. Resistors

Resistors are passive components that function primarily to divide voltage and divide current in an electrical circuit. Types of resistors based on their value, can be divided into 3, namely:

1. *Fixed Resistors* are resistors whose resistance value is fixed.
2. *Variable Resistor* is a resistor whose resistance value can be changed.
3. Nonlinear resistors are resistors whose resistance values are linear due to environmental influences, such as temperature and weather.

The value or amount of resistance on the resistor varies, the value can be known through the color-coded rings on the resistor. One way to find out the resistance price of a resistor is to read its color code.



Warna	Gelang 1	Gelang 2	Gelang 3	Multiplier	Toleransi
Hitam		0	0	1 Ohm	
Coklat	1	1	1	10 Ohm	± 1 %
Merah	2	2	2	100 Ohm	± 2 %
Orange	3	3	3	1 K Ohm	
Kuning	4	4	4	10 K Ohm	
Hijau	5	5	5	100 K Ohm	± 0,5 %
Biru	6	6	6	1 M Ohm	± 0,25 %
Ungu	7	7	7	10 M Ohm	± 0,10 %
Abu-abu	8	8	8		± 0,05 %
Putih	9	9	9		
Emas				0,1 Ohm	± 5 %
Perak				0,01 Ohm	± 10 %

figure 1. 1: Resistor color band table

B. Thermistor (PTC-NTC)

Thermistor is one type of resistor whose resistance value or resistance value is influenced by *temperature*. Thermistor stands for "t*hermal* resistor" which means *resistor* which is related to *thermal*. Thermistor consists of 2 types, namely:

1. Thermistor NTC (*Negative Temperature Coefficient*). The resistance value of the NTC thermistor will drop if the temperature around the NTC Thermistor is high (inversely proportional / Negative).
2. Thermistor PTC (Positive Temperature Coefficient). The resistance value thermistor PTC, the higher the surrounding temperature, the higher the value of the resistance (directly proportional / positive).

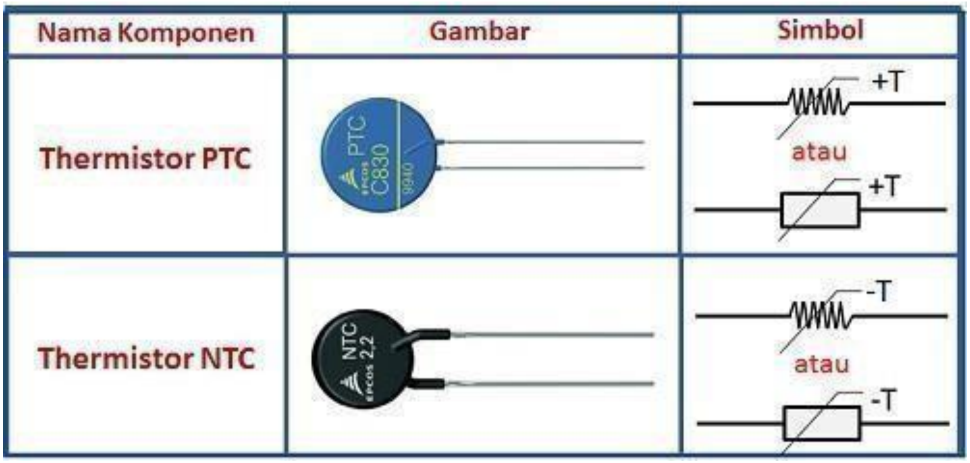


figure 1. 2: PTC and NTC thermistors

C. LDR

LDR or *Light Dependent Resistor* is a type of resistor whose resistance value or resistance value depends on the intensity of light it receives. The lower the intensity of the light received, the greater the LDR resistance value. Conversely, the higher the intensity of light received by LDR, the LDR resistance value will be smaller.

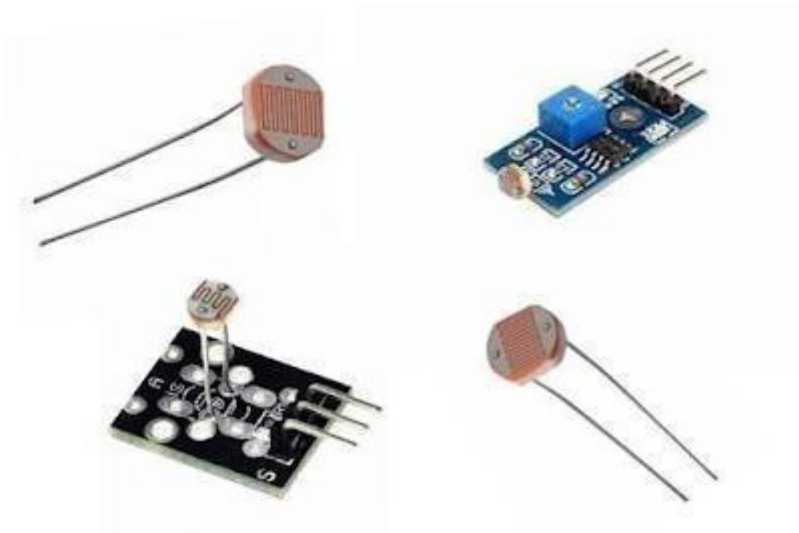


figure 1. 3: Light Dependent Resistor

The function of LDR is as a light sensor that can be used in various types of devices or electronic circuits at low cost, such as automatic switches using light which if the sensor is exposed to light then electric current will flow (ON) and vice versa if the sensor is in dark light conditions then the flow of electricity will be blocked (OFF).

D. Potentiometer

Potentiometer (POT) is one type of resistor whose value can be adjusted according to the needs of the number or the needs of the wearer. Potentiometer is a family of resistor which belongs to the category variable resistor. Structurally, the potentiometer consists of 3 terminal legs with a *shaft* or lever that functions as a regulator.

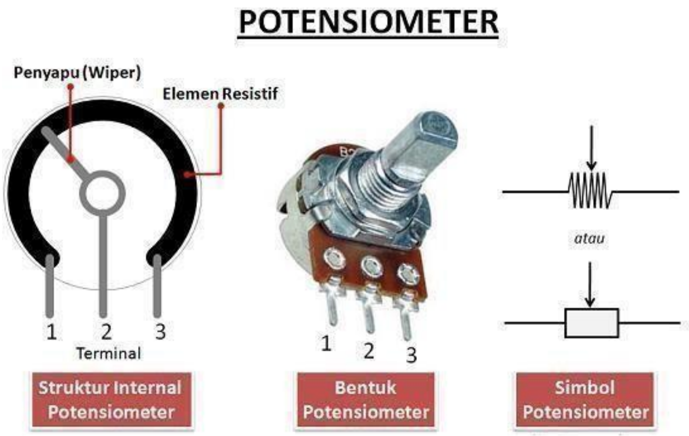


figure 1. 4: Potentiometer

Based on its shape, Potentiometers can be divided into 3 types, namely:

1. *Potentiometer Slider*, which is a Potentiometer whose resistance value can be adjusted by sliding the Wiper from left to right or from bottom to top according to its installation. Usually use the thumb to slide the wiper.
2. *Rotary Potentiometer*, which is a Potentiometer whose resistance value can be adjusted by rotating the Wiper along a circular trajectory. Usually use the thumb to rotate the wiper. Therefore, the Rotary Potentiometer is often referred to as the Thumbwheel Potentiometer.

The relationship between the legs on a rotary potentiometer:

- Foot 1 is called terminal 1
- Leg 2 is called wiper
- Leg 3 is called terminal 3

In addition, there are several things that are considered, including the following.

- **If terminal 1 and wipers are connected** to a component, it applies: turning the potentio counterclockwise reduces its resistance value, while turning the potentio clockwise increases its resistance value.
- **If terminal 3 and wipers are connected** to a component, it applies: turning the potentio counterclockwise increases its resistance value, while turning the potentio clockwise decreases its resistance value.
- **If terminal 1 and terminal 3** are connected to a component, the resistance value will remain at the maximum value of that resistance.

3. Trimmer Potentiometer, which is a potentiometer that is small in shape and must use a special tool such as a screwdriver (screwdriver) to rotate it. This Trimmer Potentiometer is usually mounted on the PCB and rarely is set up.

E. Trimmer Potentiometer

Trimpot stands for trimmer potentiometer. These are adjustable resistors commonly found in electronic design projects. Basically, the function of the trimpot on the circuit is to provide good control over the voltage level of the output signal, and basically, the trimpot adjusts the total resistance value in it.

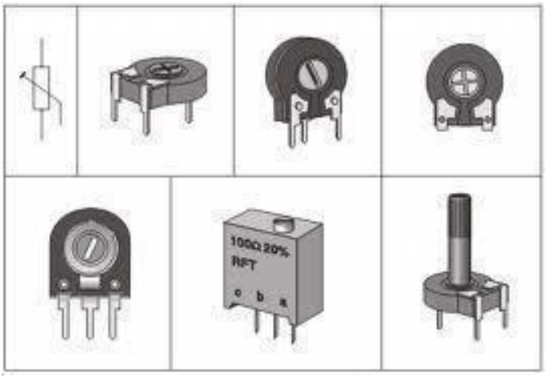


figure 1. 5: Potentiometer Trimmer

There are two basic types of trimpots used, such as:

1. Trimpot Single turn = Trimpot single turn is the most common trimpot and is used where the resolution of one turn is sufficient. In terms of price, It is a variable resistor that is quite cheap.
2. Trimpot Multi-turn = Trimpot multi turn or round is a more versatile option. By having a different number of spins available.

F. Capacitors

A capacitor is an electronic component that can store electrical charge. The structure of a capacitor is made of 2 metal plates separated by a dielectric material. The ability of a capacitor to store electric charge is called capacitance, which is denoted by the Farad (F) unit. Based on polarity, capacitors can be divided into two types as follows.

1. Polar Capacitors

   A polar capacitor is a type of capacitor that has positive and negative poles. This causes the installation of this type of capacitor in reverse can cause short circuit conditions in the circuit. Polar capacitors are usually made of electrolytes that tend to have a greater capacitance value than capacitors with paper or mica dielectric materials. The positive leg of a polar capacitor is longer than the negative leg.

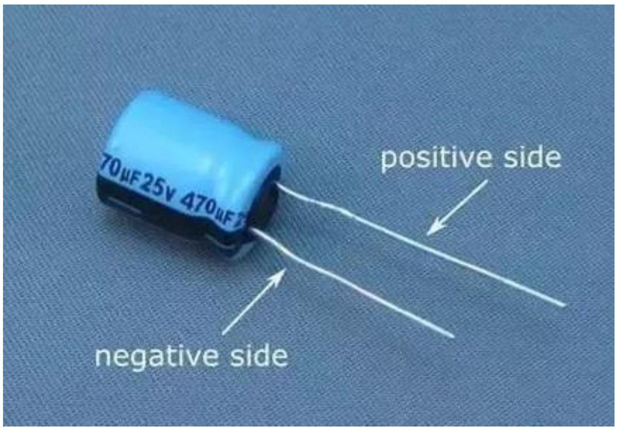


figure 1. 6: Polar Capacitors

<u>Reading Polar Capacitor values</u>

- Electrolytic Capacitors (ELCO)

  The value on the ELCO capacitor is usually already listed on the capacitor body so we just must look at the capacitor body to find out the value.

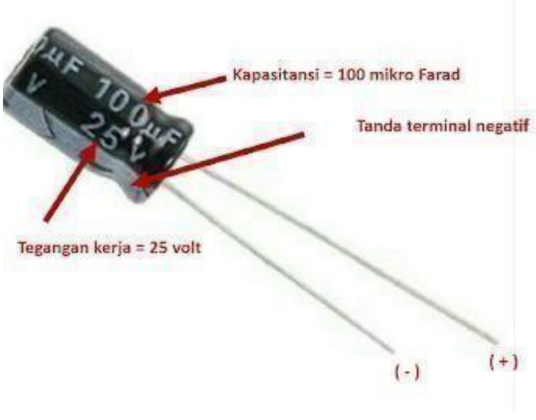


figure 1. 7: ELCO capacitor with quantity description

2. Non-Polar Capacitors

A nonpolar capacitor is a type of capacitor that has no poles, so a reverse installation will not affect the circuit at all. Nonpolar capacitors are characterized by the absence of a difference in length between their legs. Usually, nonpolar capacitors have a small capacitance value.

figure 1. 8: Non polar capacitors

<u>Reads the value of a non-polar capacitor.</u>

- **Ceramic Capacitors**

  In ceramic capacitors, the capacitance value is marked with a number located on the capacitor body. There are three sequences of numbers, each of which shows a different description. The first and second numbers are numerical values, while the third number is the multiplier value on the capacitor. The unit used in ceramic capacitors is pF (picoFarad), where 1

  pF = 1 × 10-12

  Example:

figure 1. 9: Ceramic capacitors

In the figure, the ceramic capacitor shows the number 104. This indicates that:

- ➢ The numerical value of the capacitor is 10.
- ➢ The value of the capacitor multiplier is$^{10.4}$

So, the value of the capacitor is as large as pF or about F.$10 \times 10^4 10^{-7}$

- **Mylar Capacitors**

Kode	Toleransi
F	1%
G	2%
H	3%
J	5%
K	10%
M	20%

figure 1. 10: Mylar capacitors

The way to calculate Mylar capacitors is to look directly at the code on the capacitor body. Its unit is pF (picoFarad). In the picture looks 105J400V. This shows the value on the capacitor, namely:

- ➢ The number 1 is the first digit
- ➢ The number 0 is the second digit
- ➢ The number 5 is a multiplier, namely10^5
- ➢ The letter J represents the tolerance value of 5%
- ➢ 400 V is the working voltage/voltage limit

So, the value of such capacitors is pF or F with a tolerance of 5% and 400 V working voltage.$10 \times 10^5\ 10^{-6}$

- **Polyester Capacitors**

Kode	Tegangan
1H	50V
2A	100V
2C	160V
2D	200V
2P	220V
2E	250V

figure 1. 11: Polyester capacitors

For polyester capacitors, the calculation is the same as for mylar capacitors, but the working voltage / voltage limit is at the beginning of the code. Its unit is pF (picoFarad). In the picture is seen 2A563J. This shows the value on the capacitor, namely:

- ➢ The value of 2A is the working voltage/voltage limit of 100V.
- ➢ The number 5 is the first digit.
- ➢ The number 6 is the second digit.
- ➢ The number 3 is a multiplier that is10^3.
- ➢ The letter J is a tolerance of 5%.

So, the value of such capacitors is pF or Farad with a tolerance of $5\% 56 \times 10^3 5.6 \times 10^{-5}$

G. Inductors

Inductors are electrical components formed from wire or copper windings that can store electrical energy in the form of magnetic fields. The measure of how much energy an inductor can store is called inductance, which is denoted by Henry units (H).

1. Wristband Inductor

   Reading a wristband inductor is the same as reading a resistor color band with 4 bands (three numeric bands and one tolerance value band), with microhenry (uH) reading units. An example is as follows.

figure 1. 12: Bracelet inductor

- First bracelet brown color: 1
- Second bracelet black color: 0
- Third bracelet orange color: 10^3
- Silver fourth bracelet: ±10%

Then the inductance value of the inductor is uH or 10 mH with a tolerance of ±10%.10×10^3

1.3.2 Project Board

Project boards are used to prototype electronic circuits, with component board projects easily replaced and can be reused for other projects because the components do not need to be soldered.

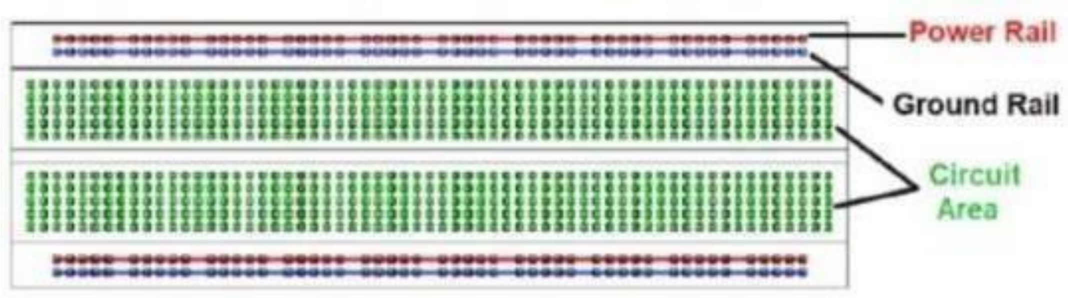
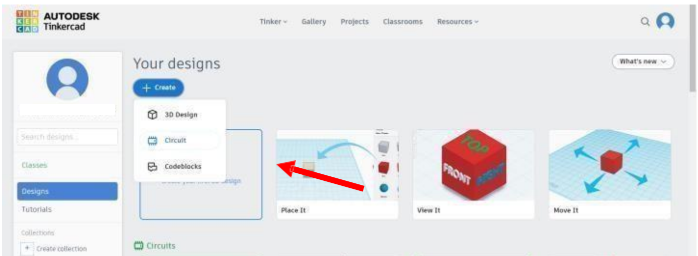


figure 1. 13: Project board

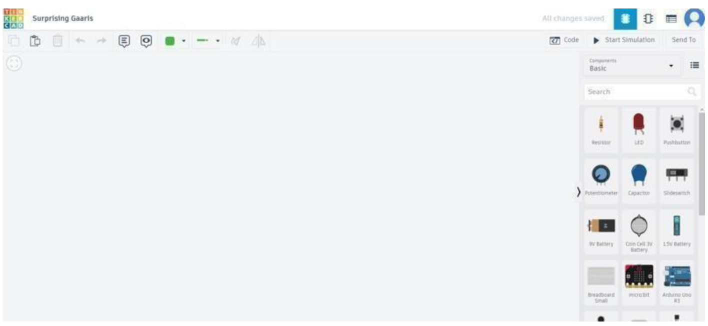
On the project board, the holes at the top are connected in a row to the side marked with red and blue lines used for the power supply line, red for the positive line of the power supply and blue for the negative line of the power supply. The central hole is connected in a row down and is used to place components.

<u>Protoboard Tutorial on Tinkercad</u>

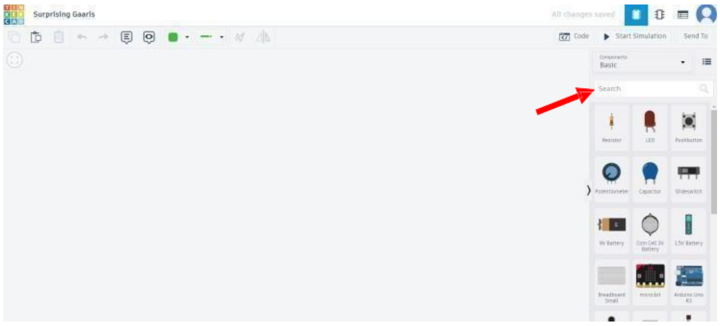
1. Go to https://www.tinkercad.com/dashboard website.
2. Select the *Create* > *Circuit* menu



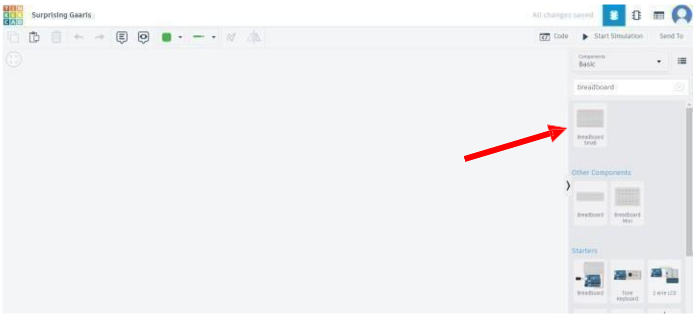
3. Then the display will be like this.

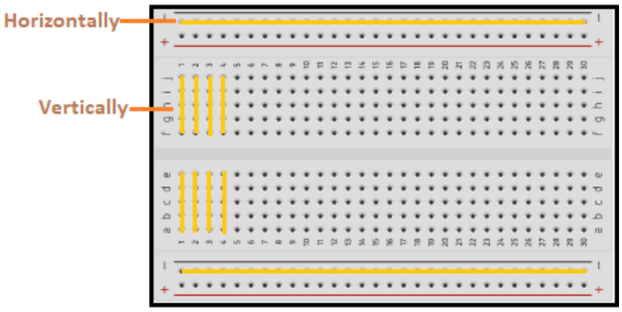


4. All components needed to assemble are available in the *Search* field.



5. Add a protoboard component by typing "*Breadboard*" in the *Search* field > select *Mini breadboard* > place it on the worksheet.





6. The horizontally connected yellow line serves as a link between the power supply and the circuit, while the vertically connected yellow line serves to assemble the components.

1.3.3 Introduction to Measuring Instruments

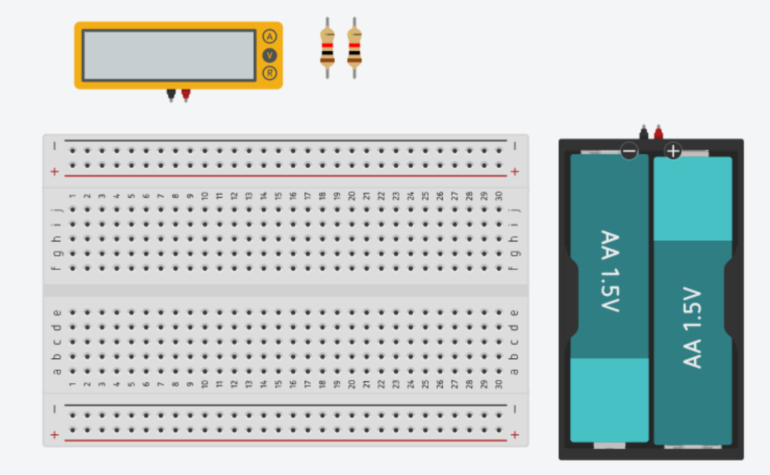
In the Electrical Circuit practicum we use a measuring instrument in the form of a multimeter. A multimeter is a measuring instrument that can measure electrical quantities. There are generally three amounts of electricity measured, namely current, voltage, and resistance. To measure the three quantities of electricity, there are three modes that we can choose on the multimeter, namely:

1. Voltmeter (measures voltage).
2. Ammeter (measuring current)
3. Ohmmeter (measures resistance).

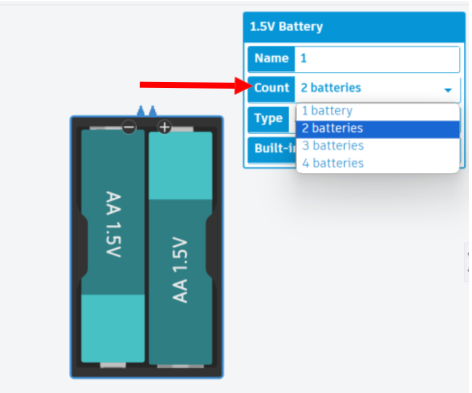
Tutorial on Using Ammeters, Voltmeters, and Ohmmeters.

A. Ammeter

1. In the *Search* field, type Breadboard, Multimeter, Resistor and 1.5v Battery > place them on the worksheet.

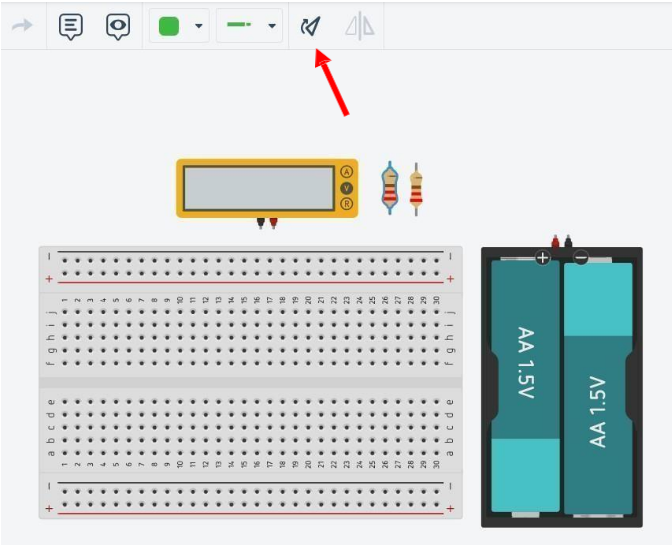


2. To change the battery value can click battery > change *count* to 2

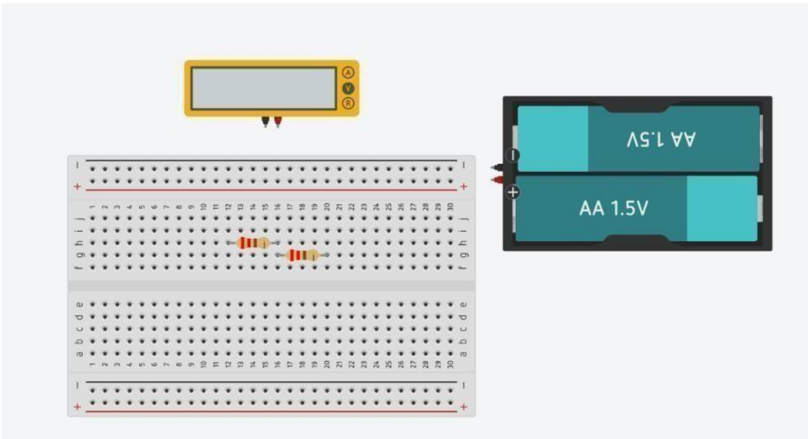


3. To change the resistor value, press on the resistor > change it to 220 Ohms

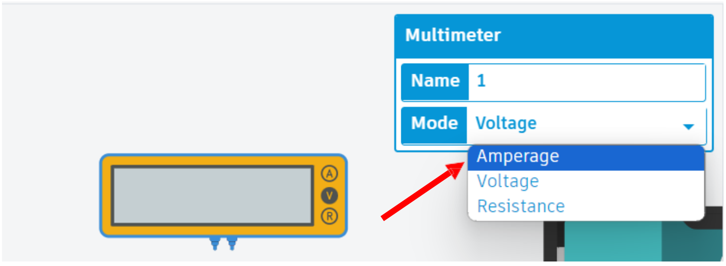
4. To rotate a component, press the selected component and then click this icon.



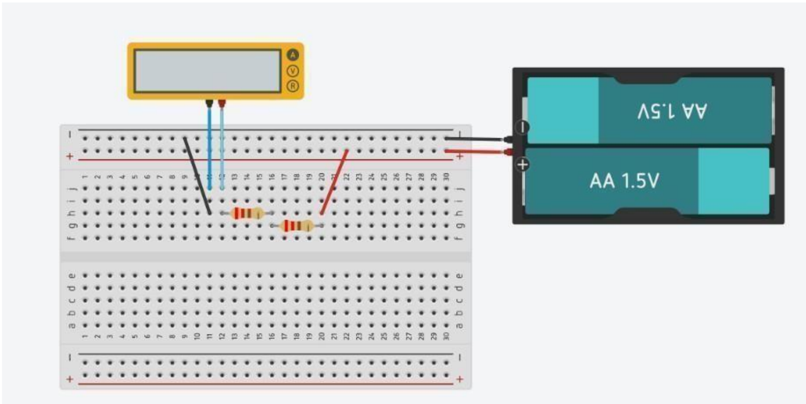
5. Place the resistor on the breadboard in a Horizontal position.



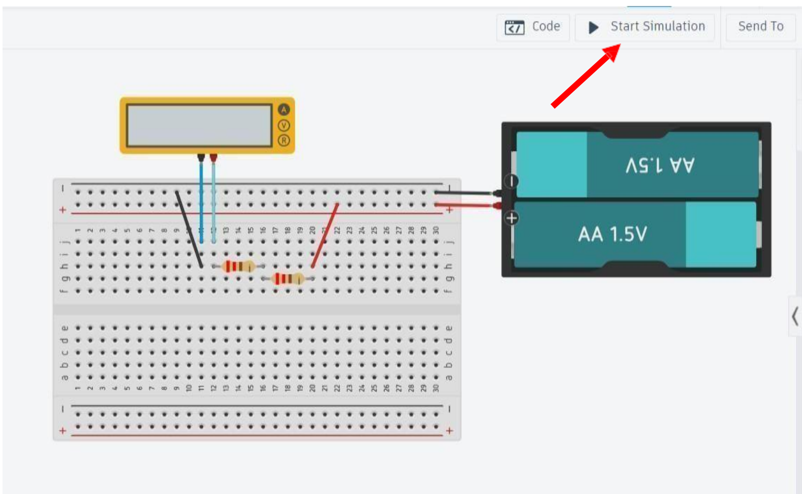
6. On the multimeter change to mode A*mperemeter* by pressing the multimeter > mode = *Ampereage*



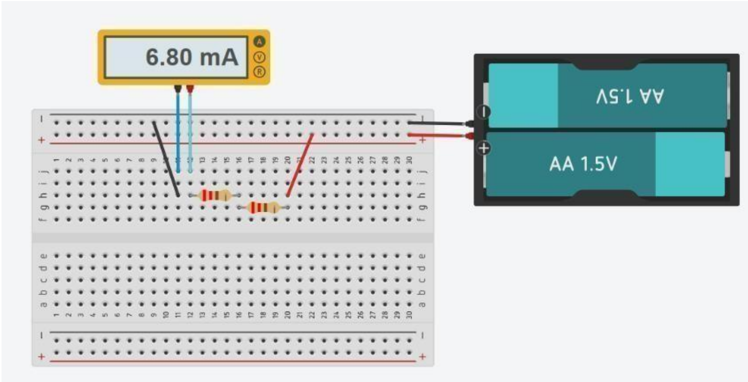
7. Connect the positive pole of the battery to power (+) and the negative pole to ground (-)
8. Circuit resistors in series with a multimeter



9. Click *Start Simulation*

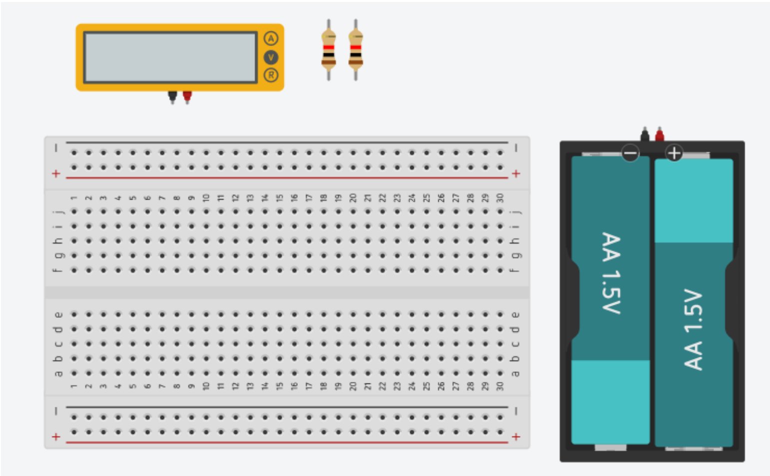


10. The multimeter will display the current value.

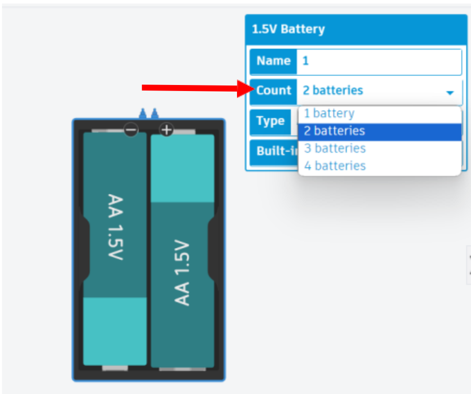


B. Voltmeter

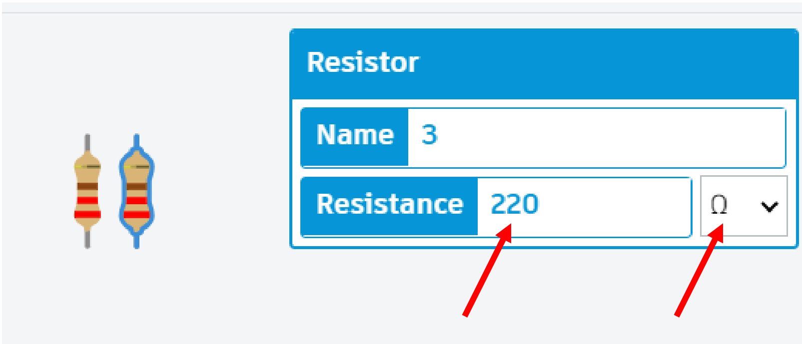
1. In the *Search* field, type Breadboard, Multimeter, Resistor and 1.5v Battery > place them on the worksheet.



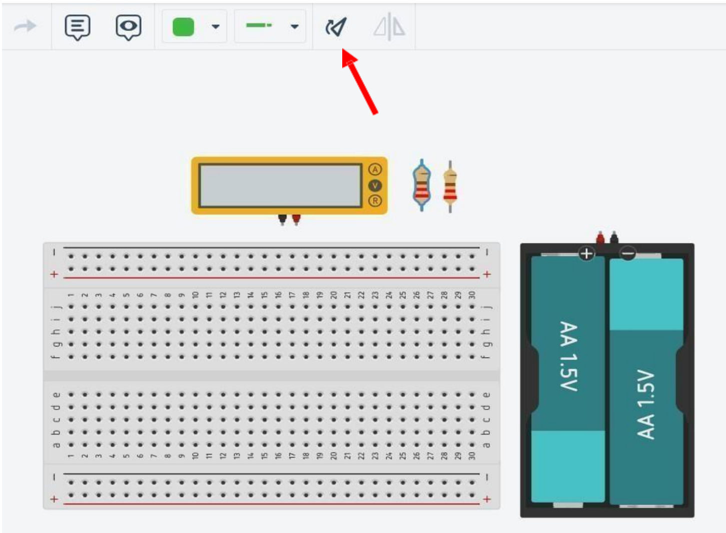
2. To change the battery value can click battery > change *count* to 2



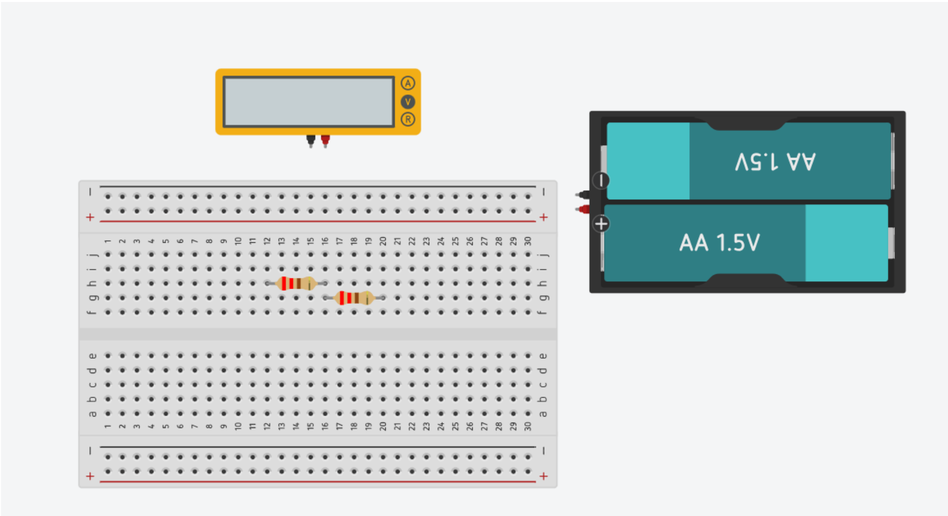
3. To change the resistor value, press on the resistor > change it to 220 Ohms



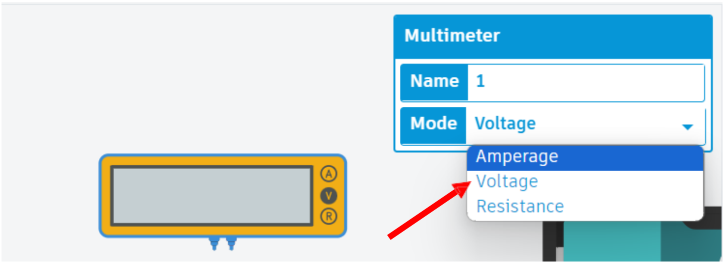
4. To rotate a component, press the selected component and then click this icon.



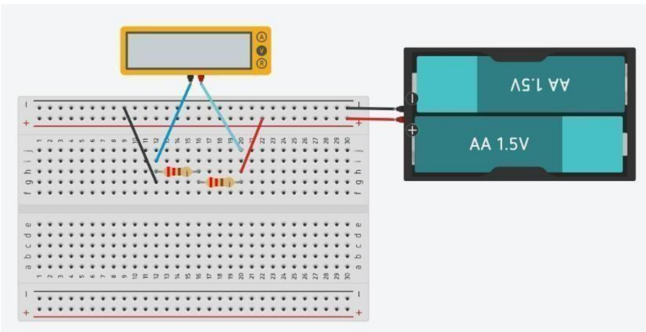
5. Place the resistor on the breadboard in a Horizontal position.



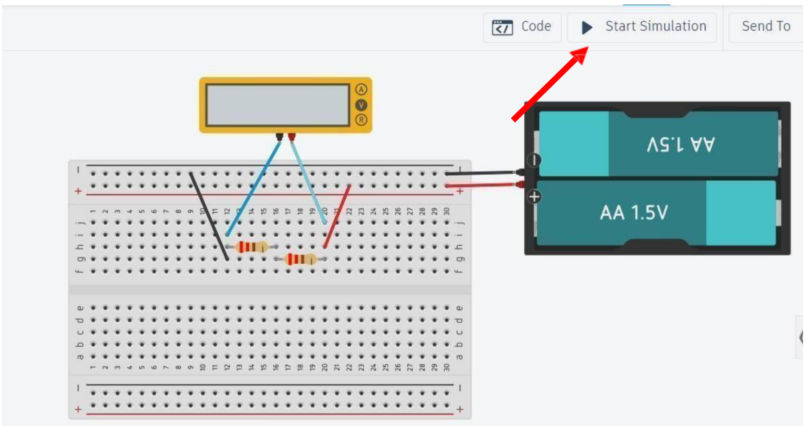
6. On the multimeter change it to Voltmeter mode by pressing the multimeter > mode = *Voltmeter*.



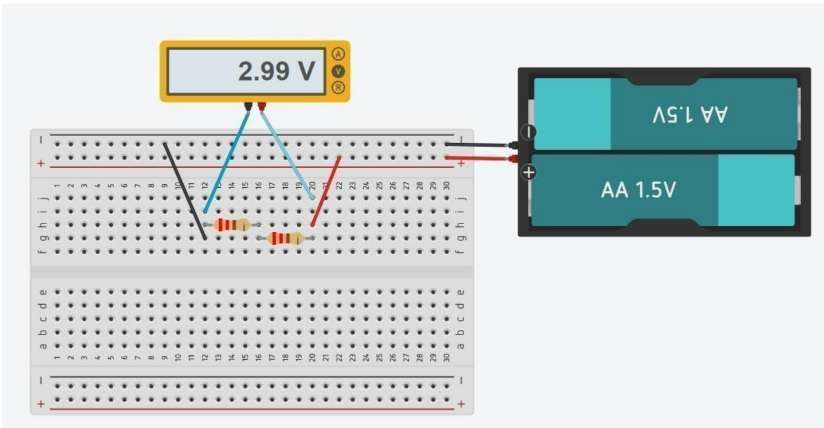
7. Connect the positive pole of the battery to power (+) and the negative pole to ground (-).
8. Assemble resistors in series. Place the positive and negative sides of the multimeter on the end of each resistor to calculate the total voltage.



9. Click *Start Simulation*

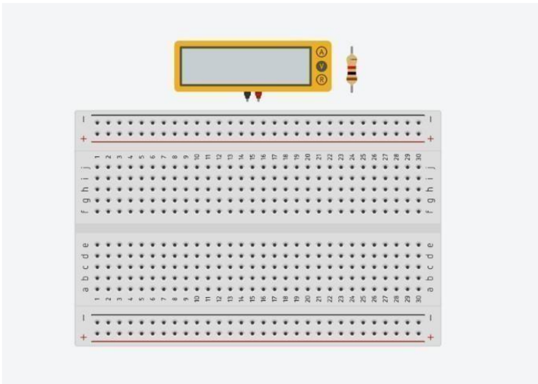


10. The multimeter will display the voltage value.

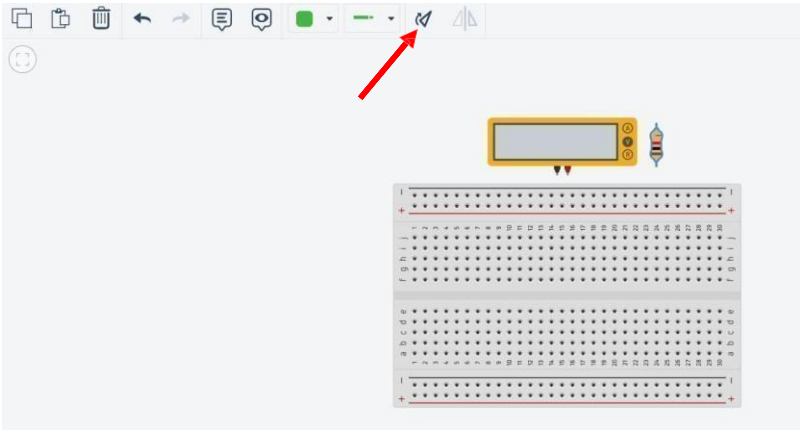


C. Ohmmeter

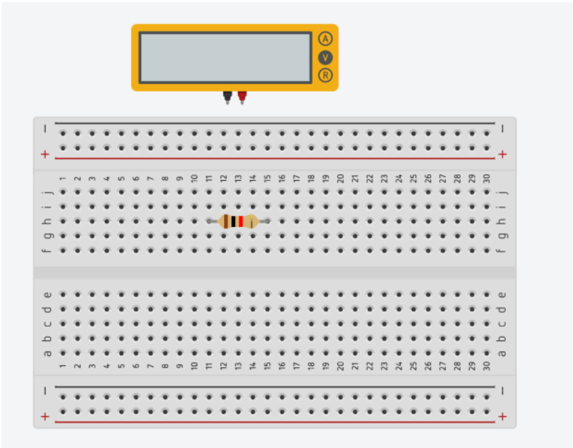
1. In the *Search* column, type Breadboard, Multimeter and Resistor > place them on the worksheet.



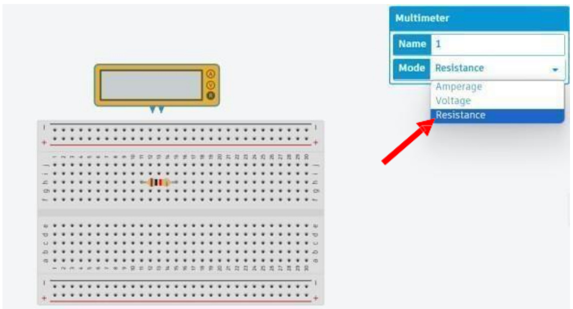
2. To rotate a component, press the selected component and then click this icon.



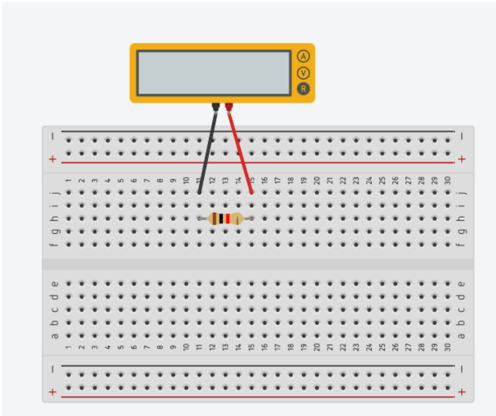
3. Place the resistor on the breadboard in a Horizontal position.



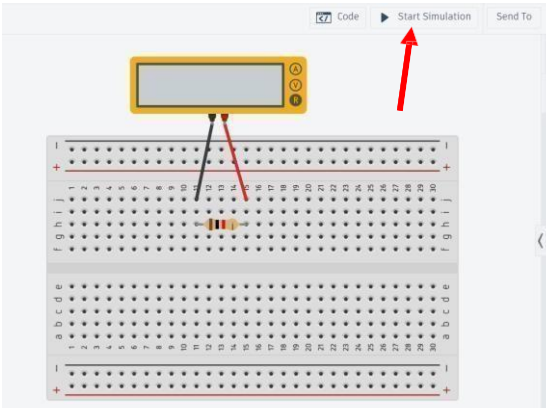
4. On the multimeter change it to Resistance mode by pressing the multimeter > mode = *Resistance.*



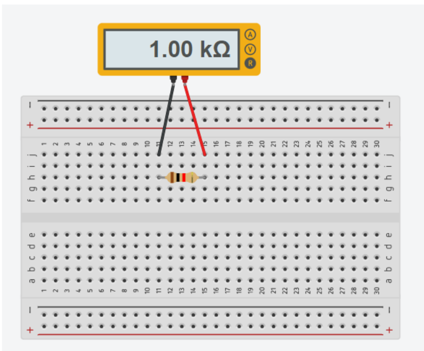
5. Connect the pole end of the multimeter to the breadboard hole connected to each resistor leg.



6. Click *Start Simulation*

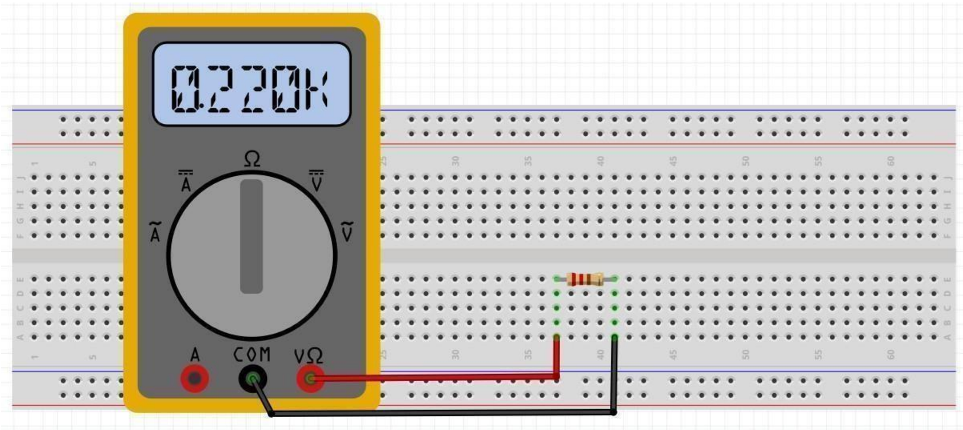


7. The multimeter will show the value of the resistor.



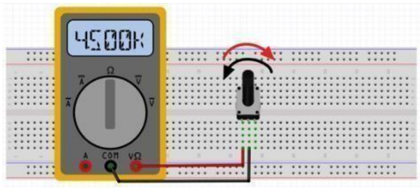
1.4 Practicum Procedure

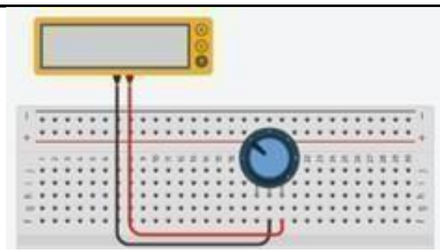
1.4.1 Resistor Readings and Measures



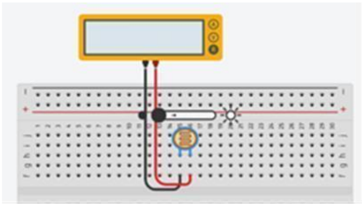
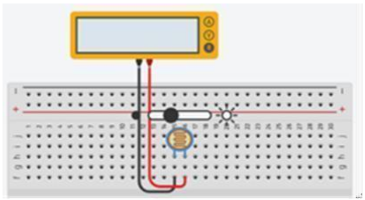
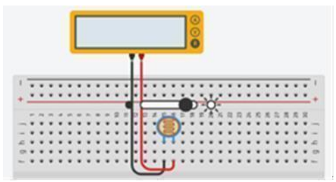
<table>
<tr><th>Component</th><th>Color code reading</th><th>Ohmmeter measurement</th></tr>
<tr><td>Example:
</td><td><table><tr><td>Red</td><td>Red</td><td>Brown</td><td>Gold</td></tr><tr><td>2</td><td>2</td><td>x10</td><td>5%</td></tr></table>Value = 220 ± 5%
Range = 219.95 – 220.05</td><td>0.220k</td></tr>
</table>

1.4.2 Variable Resistor readings and measurements

Component	Reading Number code	Measurement Instructions with Ohmmeter
 	**B10K** Linear potentiometer with maximum large resistance of 10k	- Arrange measurements as shown below for potentiometer feet in legs 1 and 2, - Position the full button to the left - Turn the potentio 45 degrees to the right and measure the size of the resistance 
 	**103** Trimpot with max large 10K resistance	- Set a multimeter on resistance measurement. - Variable resistor at leftmost rotating position - Measure **foot 1 and foot 2** with a multimeter - Turn the variable resistor approximately **45 degrees** - Write down the value indicated on the multimeter

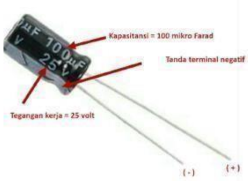
1.4.3 LDR Readings and Measurements

Component	Light Conditions	Ohmmeter measurement
	**Dark** 	- Sequence measurements like the image on the side - Slide the position of the light regulator, the more left is shifted, the darker the light will be - The results of these measurements will be visible on the multimeter
	Little light 	- Sequence measurements like the image on the side - Shift the position of the light adjuster, for low light conditions slide to the center/neutral position - The results of these measurements will be visible on the multimeter
	Bright 	- Sequence measurements like the image on the side - Shift the position of the light regulator. The more right you slide, the brighter the light will be - The results of these measurements will be visible on the multimeter

1.4.4 Capacitor Readings and measurements

First digit	Second digit	Multiplier	Tolerance
1	0	$\times 10^4$	-

Capacitance	Work mid-range
100 μF	25 V

Component	Number code reading
	Value = pF or F$10 \times 10^4 10^{-7}$
	Value = 100 μF or 100 x 10-6 ^{F} with a working voltage of 25 V

1.4.5 Inductor Readings and P measurements

Brown	Black	Orange	Silver
1	0	$\times 10^3$	5%

Component	Color code reading
	10×10^3 Value = uH or 10 mH with a tolerance of 5%

1.4.6 Number Inductor Readings and Measurem

<table>
<tr><th>Component</th><th>Number code reading</th></tr>
<tr><td></td><td>
<table>
<tr><th>First digit</th><th>Second digit</th><th>Multiplier</th></tr>
<tr><td>1</td><td>5</td><td>×10^1</td></tr>
</table>
Value = 150 uH
</td></tr>
</table>

Modul 2
Direct Current Circuit

2.1 Purpose

After attending this practicum, students are expected to:

1. Know how to measure current, voltage and resistance using a multimeter.
2. Can use oscilloscopes to measure voltage, frequency and phase differences of various waveforms.
3. Can measure and calculate current and voltage on a load in a linear circuit by applying the superposition and substitution theorems.

2.2 Tools and Materials

1. Multimeter
2. DC voltage source
3. *Project Board*
4. Jumper Cable
5. Probe

2.3 Theoretical Basis

2.3.1 Measuring Instruments

A. Multimeter

Multimeter is a measuring instrument used to determine the value of electrical quantities such as voltage, current, resistance, frequency, and others. There are two types of multimeters, namely analog and digital. Measurement mode on the multimeter:

1. Voltmeter

Voltmeters are used to measure the voltage from the terminals or ends of a circuit and are placed in parallel to the load of the circuit to which the voltage is to be known. According to the nature of parallel circuits, stringing the circuit in parallel will make the voltage in the voltmeter equal to the voltage of the components to be measured.

A good voltmeter is a voltmeter that has a large internal resistance (Rv), the greater the better, because the resistance in the voltmeter must be as large as possible to avoid the current being divided into circuits that cause the measured voltage is not the actual voltage.

Voltage Measurement Circuit

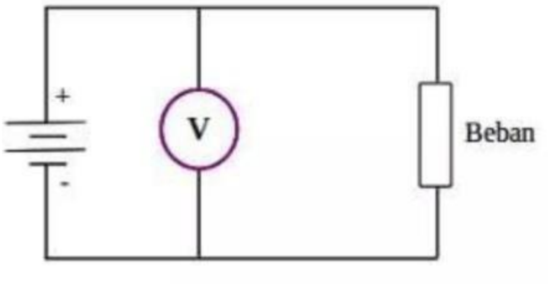


figure 2. 1: Voltage measurement scheme

V voltmeter is connected in parallel with the load, How to measure voltage, namely components or circuits between 2 points to be measured voltage, must be connected in parallel with the voltmeter measuring instrument.

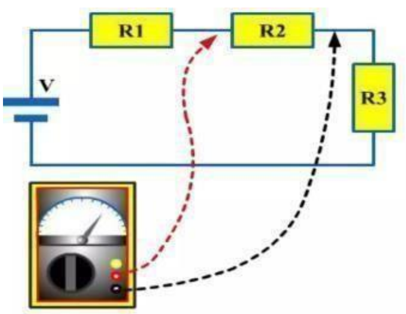


figure 2. 2: Another example of a voltage measurement scheme.

1. **Ammeter**

Ammeters are used to measure the current of a circuit and are assembled in series with the circuit to be measured current. Stringing in series will make the current through the ammeter equal to the measured current (the nature of the series circuit).

A good ammeter is an ammeter that has a small inner resistance (Ra), the smaller the better, because (Ra) must be as small as possible to avoid *voltage drops* in the circuit (there is a divided voltage) so that the measured current is not the current to be measured actually.

Current Measurement Circuit

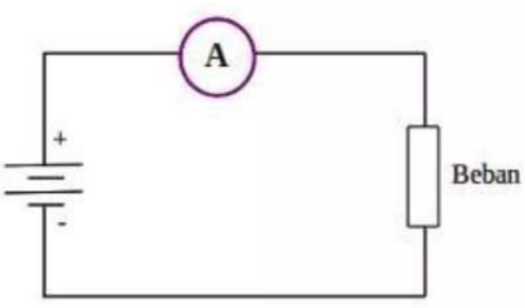


figure 2. 3: Current measurement circuit

Amperemeter A is connected in series with the load. How to measure current is with components or branches of the circuit that will be measured current, must be opened / disconnected and then connected in series with an ammeter measuring device.

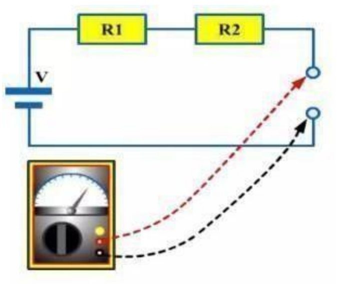


*figure 2. 4****: Another example of a current measurement scheme.***

1. **Ohmmeter**

Ohmmeter is an electrical resistance measuring instrument. The magnitude of the unit of resistance measured by this device is expressed in ohms. This ohmmeter uses a galvanometer to measure the amount of electric current passing through an electrical resistance (R), which is then calibrated to ohm units.

The working principle is that the object is electrified and measured electrical resistance / resistance. Electrical resistance/resistance is the ratio between the electrical voltage of an electronic component (e.g. resistor) and the electric current passing through it.

2.3.2 Direct Current Circuit

A. Linear Circuit

A linear circuit is a circuit formed by linear sources and the current or and voltage equations of the circuit satisfy the linear equation. An equation is said to be linear if it satisfies the property of superposition, i.e. $f(kx) = k.F(x)$, where k is a constant.

B. Superposition Series

The superposition theorem states that the current and voltage for each element in a linear circuit with multiple sources is the sum of the current and voltage produced by each source acting independently. Here's an illustration.

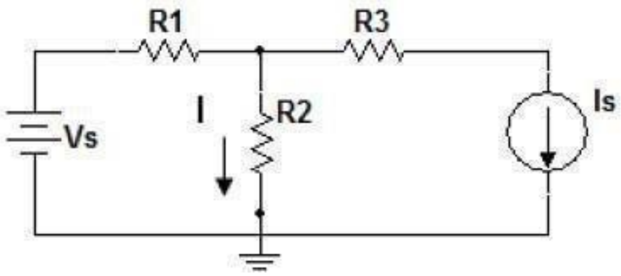


figure 2. 5: Two-source free circuit

From the above circuit, the value of current I is asked, the step to work on it if using the Superposition Circuit:

1. When the voltage source is active and the current source is off *(Open Circuit).*

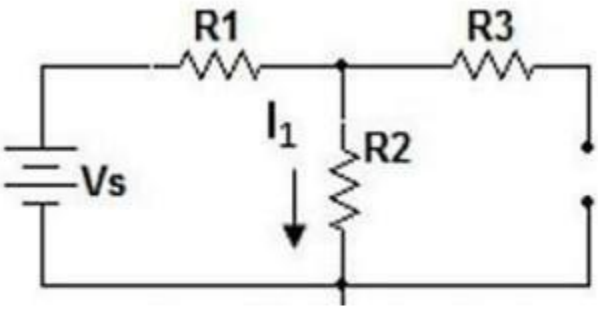


figure 2. 6: The current source is made into an open circuit

2. When the current source is active and the voltage source is off *(Short Circuit).*

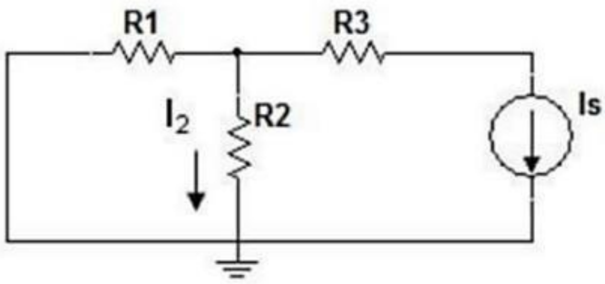


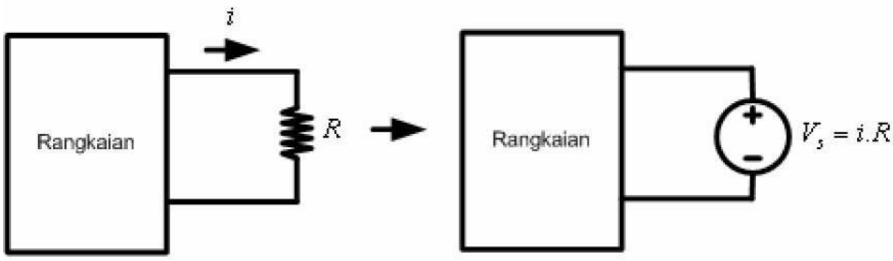
figure 2. 7: The voltage source is made into a short circuit.

3. With the I_1 and I_2 values obtained, sum the two values to get the result of the I value.

C. Substitution Circuit

In this substitution theorem applies that a passive component or element through which a current flows (of i) then the passive component can be replaced with a voltage source Vs which has the same value as the current through the passive component.

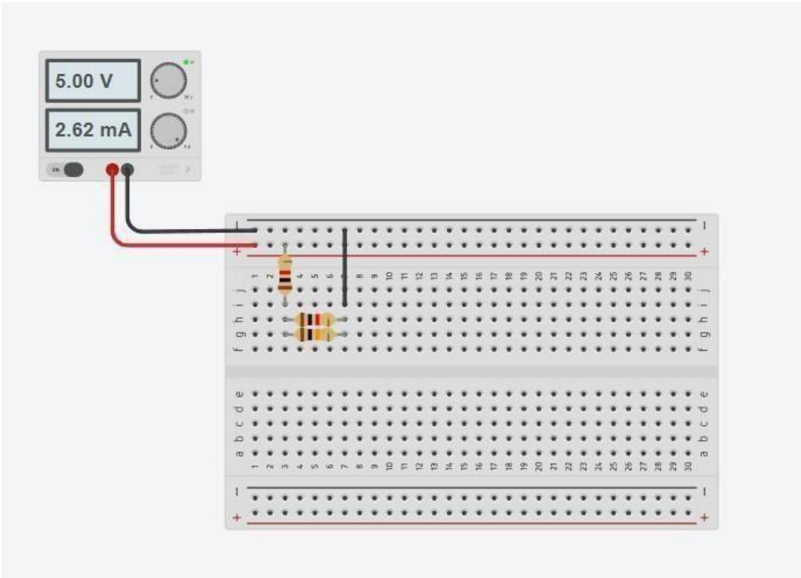
If the passive component is a resistor of R, then the replacement voltage source is $V_s = i.R$ with the inner resistance of the voltage source equal to zero.



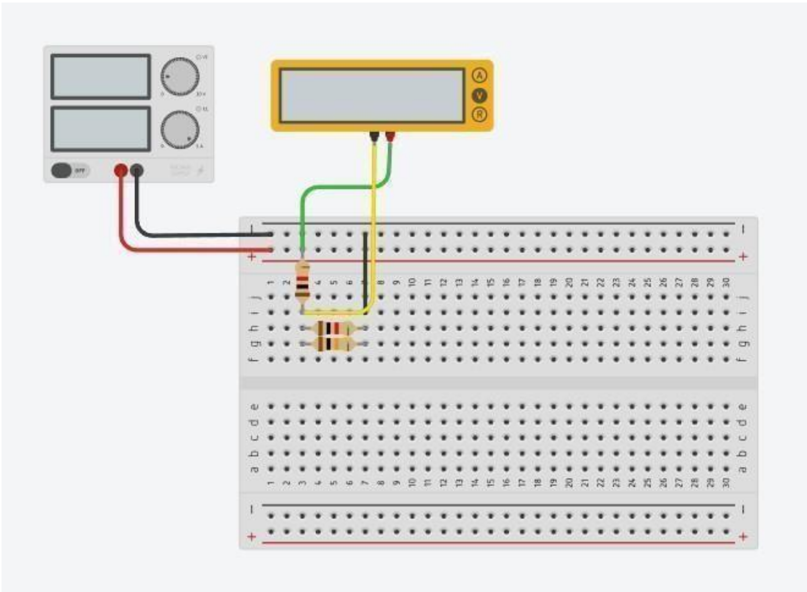
2.4 Practicum Procedure

2.4.1 DC Measurement

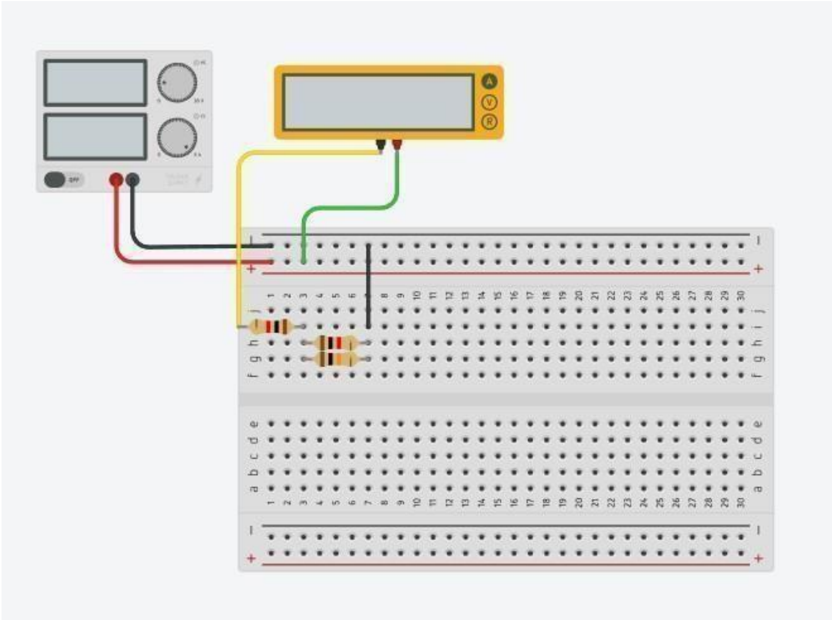
1. Make a series as follows.



2. Measure the voltage value on the 1k resistorΩ. An example of the circuit is as follows



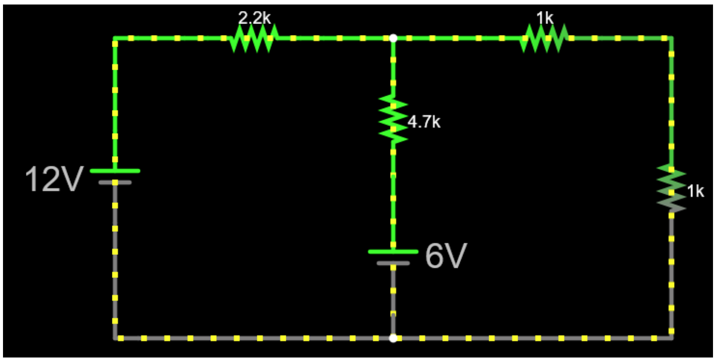
3. Next, measure the current on the same resistor. An example of the circuit is as follows.



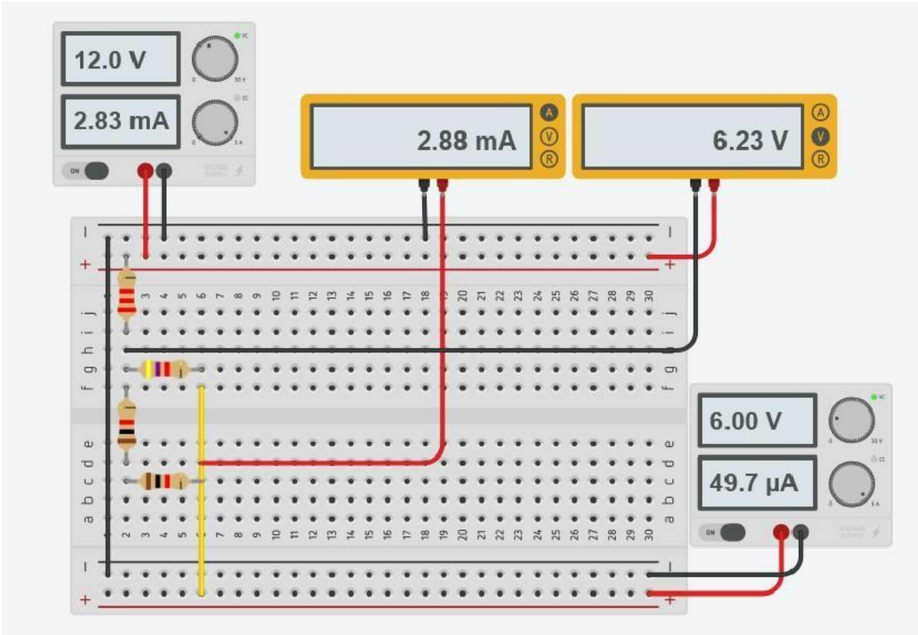
4. Change the 1kresistor Ω to 2k2Ω.
5. Repeat steps number 2 and number 3 on the replaced resistor.
6. Change the 2k2Ω resistor to 4k7Ω.
7. Repeat steps number 2 and number 3 on the replaced resistor.

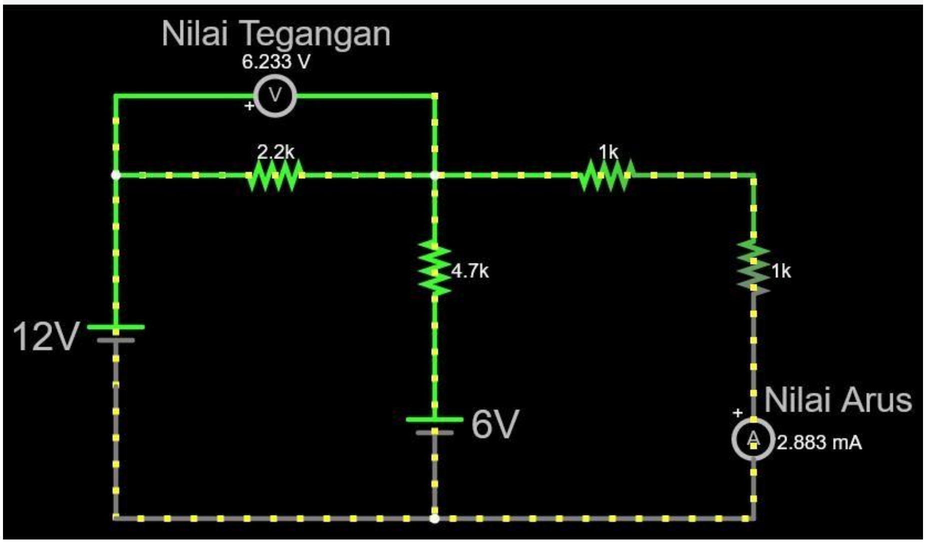
2.4.2 Superposition Series

1. Make a series as follows.

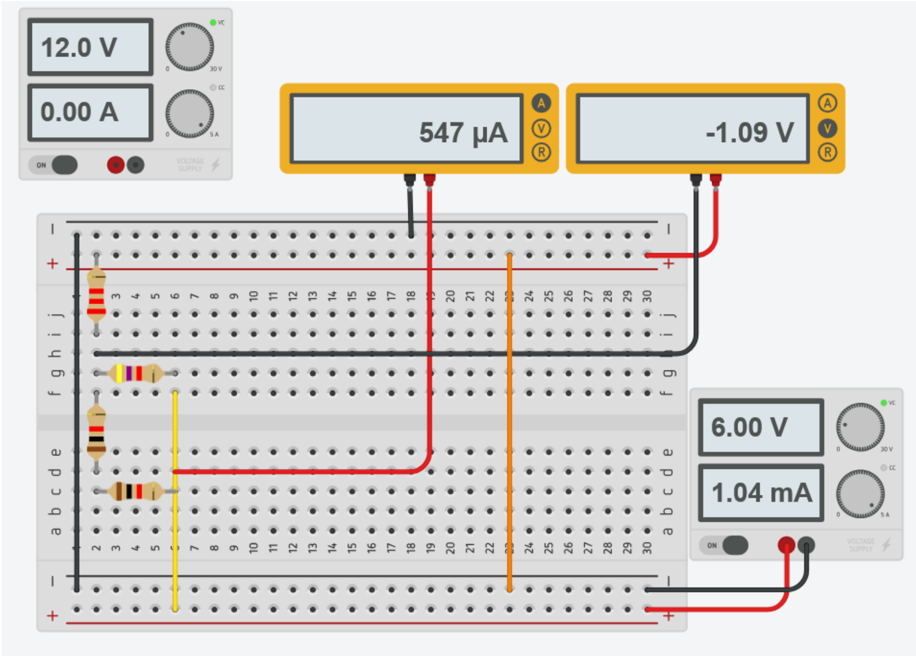


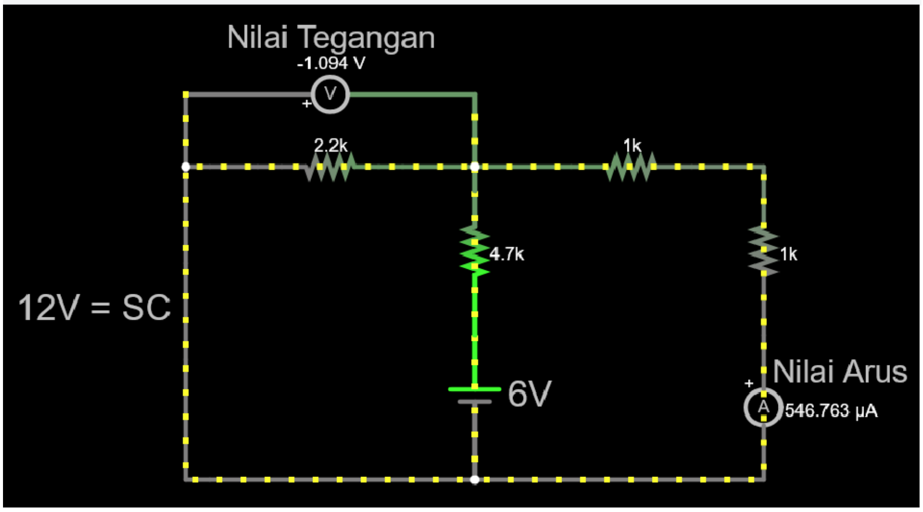
2. With 12V and 6V active, measure and record the current value (i) in the 1k resistor Ω in the journal.



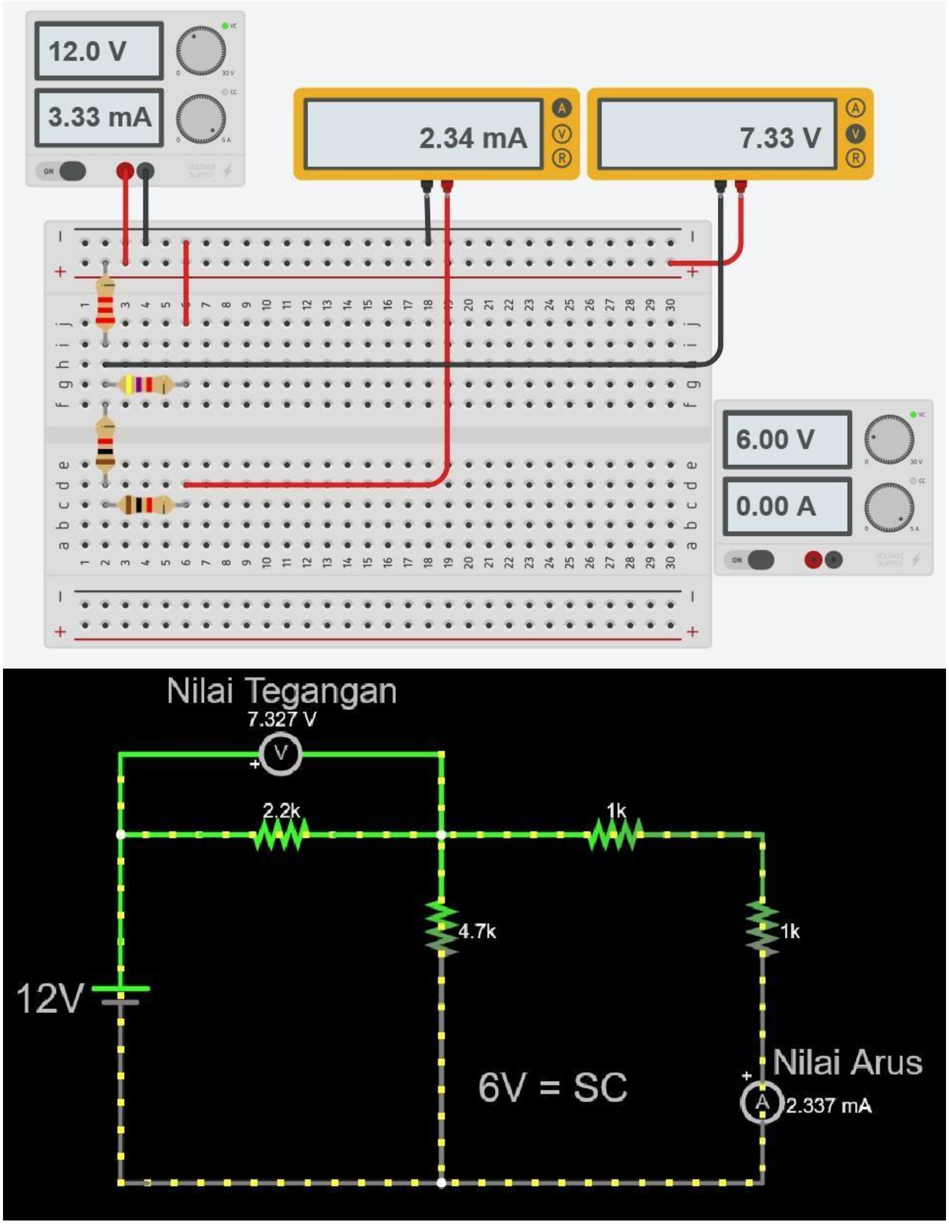


3. Unplug and increase the voltage of 12V into *a short circuit*, measure the current at the same point, then record the measurement results in a journal.





4. Reconnect the 12V voltage to the circuit, unplug and convert the 6V voltage to *a short* circuit, measure and record the current value at the same point on the journal.



5. Compare the three current measurement results at that point.

Module 3
Thevenin and Norton's theorem

3.1 Purpose

After attending the practicum, practicum is expected to:

1. Proving Thevenin's theorem and its successor circuits
2. Proved Norton's theorem and its substitute circuits.
3. Determine the voltage and current in the Thevenin and Norton replacement circuits.
4. Proving maximum power transfer

3.2 Tools and Materials

1. *DC Power Supply*
2. Multimeter
3. Jumper
4. 100 Ω, 220 Ω, 4k7 Ω, 1k Ω, 2k2 Ω, and 470 Ω Resistors
5. Project Board

3.3 Theoretical Basis

3.3.1 Thevenin's theorem

Thevenin's theorem states that a two-terminal linear circuit can be simplified to one equivalent voltage source (Vth) connected in series with an equivalent resistance (Rth) at the two observed terminals. The replacement circuit scheme is as follows.

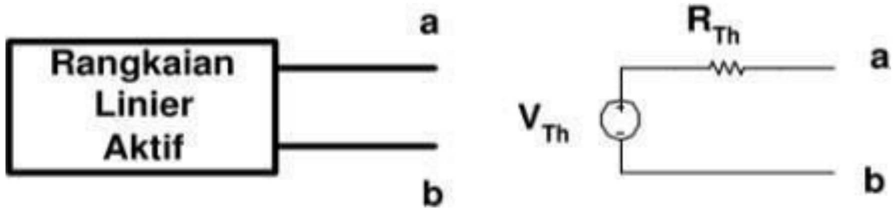


figure 3. 1: Thevenin replacement circuit

The purpose of this theorem is to simplify the analysis of complex circuits into simpler substitute circuits, with the same calculation results as the previous circuits.

The steps for simplifying circuits using Thevenin's theorem are as follows.

1. Determine the two points that will be measured for electricity (current / voltage).
2. Think of these points as points a and point b, and the connection between them as parameters a-b.
3. Remove the resistor connecting point a to point b (**R_{ab}**) so that the connection between the two points is broken (*open circuit*), then calculate the voltage flowing between point a and point b (voltage this we refer to as **V_{th}**).
4. Turn off all free sources (short circuit for voltage sources and *open* circuit for current sources), then calculate the equivalent resistance of the circuit (equivalent resistance value = **R_{th}**).
5. Create a replacement circuit for Thevenin by connecting **V** th, R_{ab}, and **R_{th}** in series.
6. Calculate the magnitude (voltage/current) at **R_{ab}.**

3.3.2 Norton's theorem

Norton's theorem states that any electrical circuit consisting of multiple current sources, voltage sources, and resistances can be replaced by a substitute circuit consisting of a single current source (**I**N) known as a Norton current and a single resistance (**R_N**) known as Norton resistance. Here is a picture of his replacement circuit.

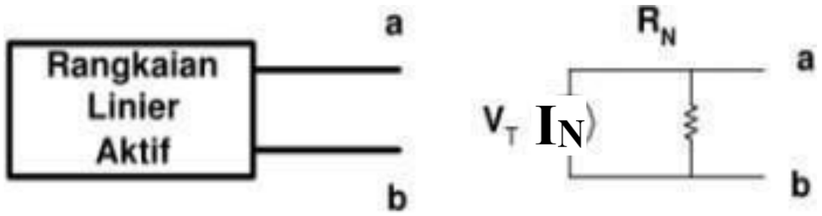


figure 3. 2: Norton's suite of replacements

Just like Thevenin's theorem, Norton's theorem aims to simplify circuit analysis. However, Norton's theorem has a different substitute circuit than Thevenin's theorem.

The steps for simplifying the circuit using Norton's theorem are as follows:

1. Determine the two points that will be measured for electricity (current / voltage).
2. Think of these points as points a and point b, and the connection between them as parameters a-b.
3. Remove the resistor connecting point a to point b (**R_{ab}**) so that the connection between the two points is broken (*open circuit*), then reconnect the two points using *a jumper* So that parameters A-B become short *circuits*.
4. Calculate the value of the current (**I_N**) on parameters a-b.
5. Turn off all free sources (short circuit for voltage sources and *open* circuit for current sources), then calculate the equivalent resistance of the circuit (equivalent resistance value = **R_N**).
6. Create a Thevenin replacement circuit by connecting **I** N, R **ab**, and **R N** in parallel.
7. Calculate the magnitude (voltage/current) at **R_{ab}.**

3.3.3 Maximum Power Transfer

Maximum power transfer occurs when the load resistance value is equal to the source resistance value ($R_S = R_L$), either installed in series with the voltage source or installed in parallel with the current source. When we use Thevenin's theorem to analyze the circuit the source resistance value is the equivalent resistance value (R_S = Rth) so the maximum power transfer occurs when the load resistance value must be equal to the equivalent resistance value, similarly when we use Norton's theorem.

The maximum power transfer theorem can be proven if the greatest power occurs when the equivalent resistance of the circuit is equal with Rth or R_N, with the same current and voltage values. The derivation of the formula is as follows

- $P_L = V_L \times I = (I \times R_L) \cdot I = I^2 \cdot R_L$

where,

- $I = \frac{V_S}{R_S + R_L}$

thus,

- $P_L = \left(\frac{V_S}{R_S + R_L}\right)^2 \cdot R_L$

Assuming V_S and R_S are fixed, and P_L is a function of R_L, then to find the maximum value of P_L is:

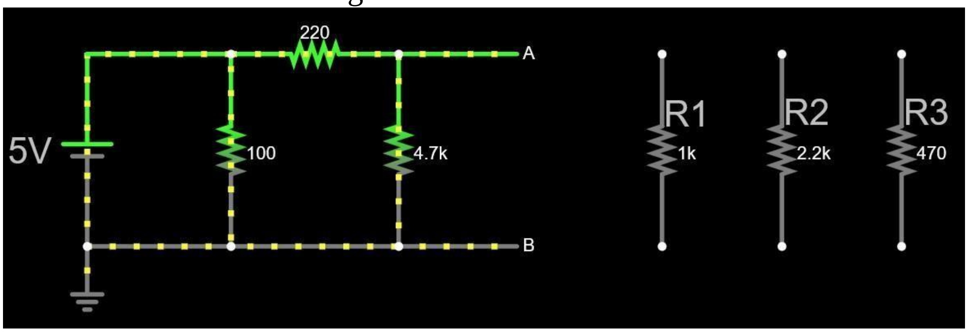
- $P_L = \left(\frac{V_S}{R_S + R_L}\right)^2 \cdot R_L = \frac{V_S^2}{(R_S + R_L)^2} \cdot R_L$
- $P_L = V_S^2 (R_S + R_L)^{-2} R_L$
- $\frac{dP_L}{dR_L} = V_S^2 [(R_S + R_L)^{-2} - 2(R_S + R_L)^{-3} R_L]$
- $0 = V_S^2 \left[\frac{1}{(R_S + R_L)^2} - \frac{2R_L}{(R_S + R_L)^3}\right]$
- $\frac{1}{(R_S + R_L)^2} = \frac{2R_L}{(R_S + R_L)^3}$
- $1 = \frac{2R_L}{(R_S + R_L)}$
- $R_S + R_L = 2R_L$
- $R_s = R_L$

Thus $R_L = R_S$

3.4 Practicum Procedure

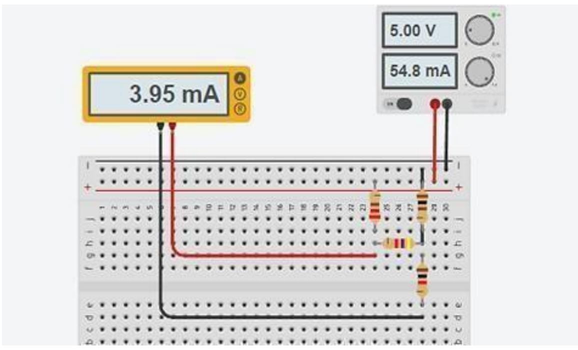
3.4.1 Direct Current and Voltage Measurement

1. Create a circuit with the following scheme.

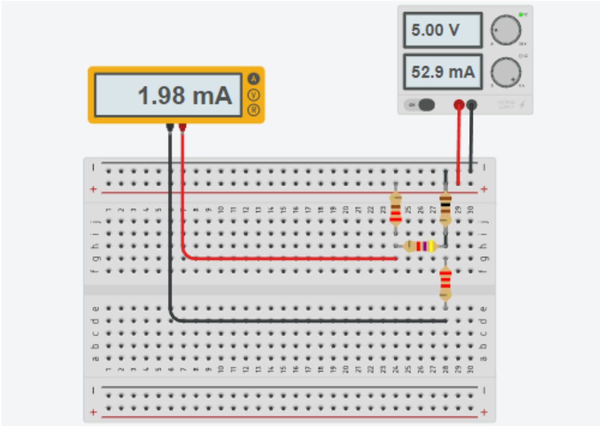


2. Measure the current from point a to point b with different resistor values! (1KΩ, 2k2 Ω, and 470Ω). Here is an example of the series.

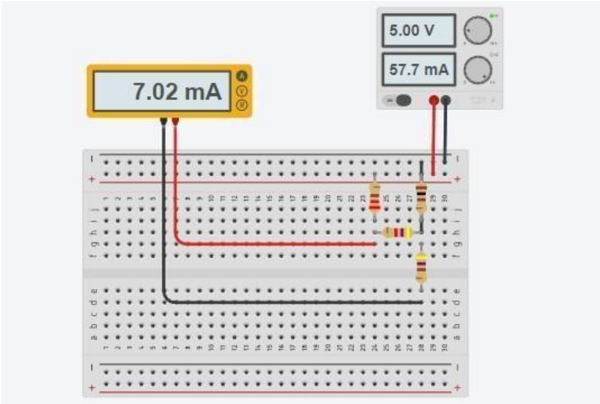
   a. 1k resistors Ω:



b. 2k2 resistors Ω:



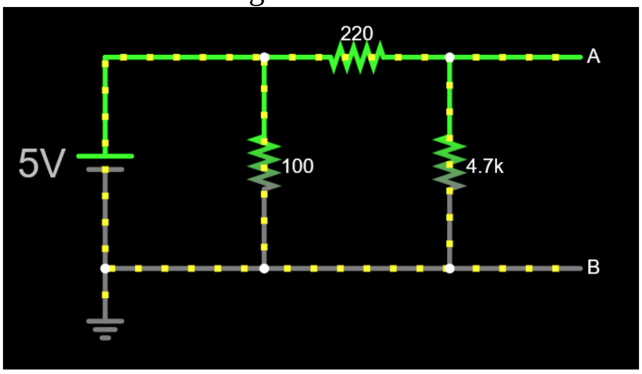
c. Resistor 470Ω:



3. Record the current measurement results in a journal

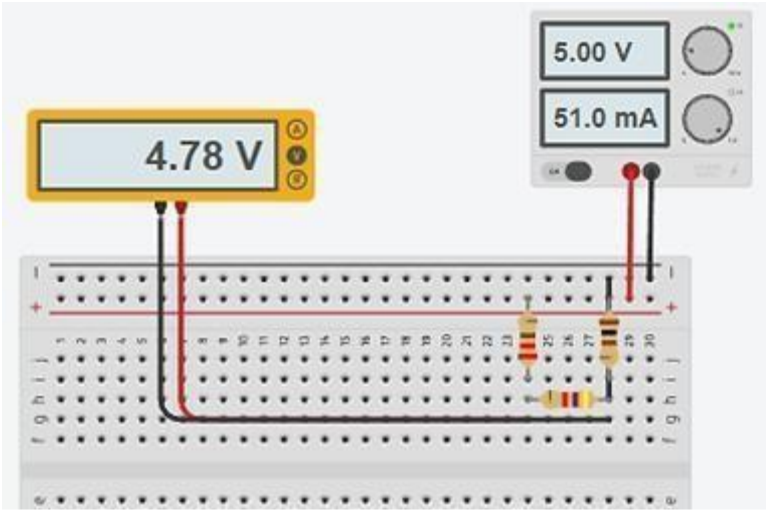
3.4.2 Thevenin's theorem

1. Make a circuit with the following scheme.

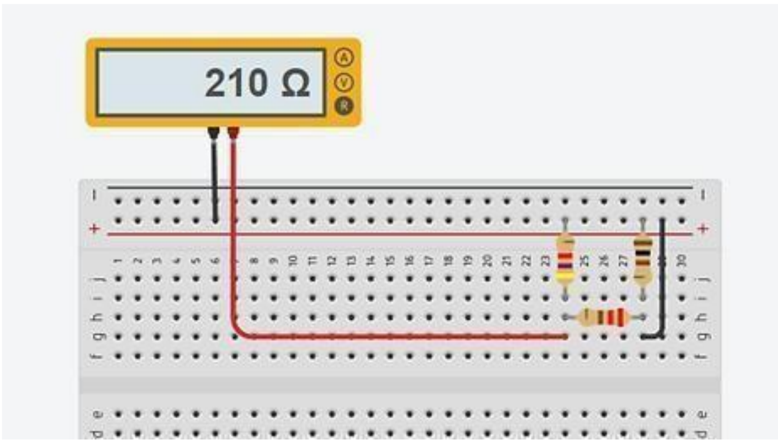


2. Remove the resistor at point a-b so that the connection of point a to point b is broken (*open circuit*).

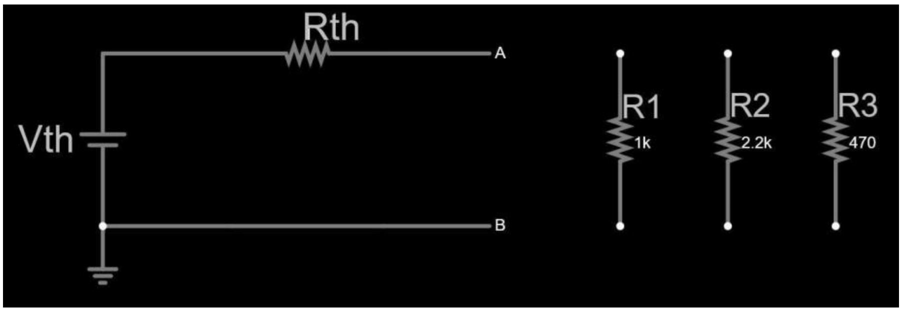
3. Calculate the value of V_{th} by measuring the voltage value at point a to point b using a multimeter. An example of the circuit is as follows.



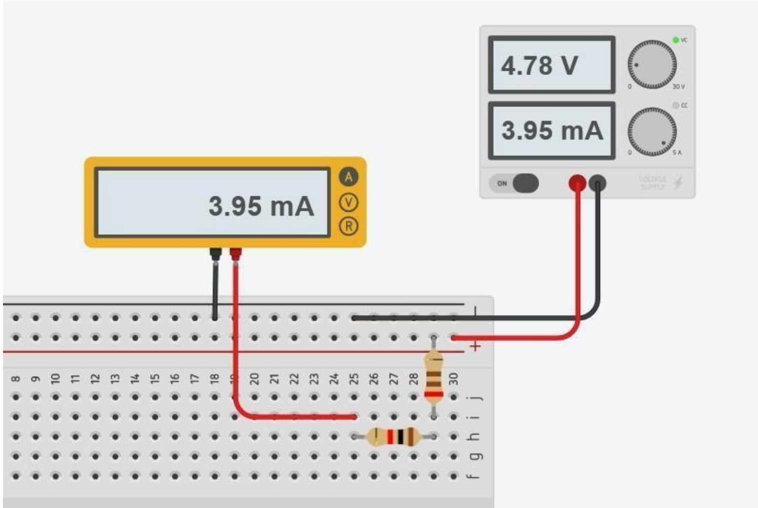
4. Disconnect the voltage source and make a short circuit using a jumper, then calculate the value of Rth with a multimeter. The example circuit is as follows.



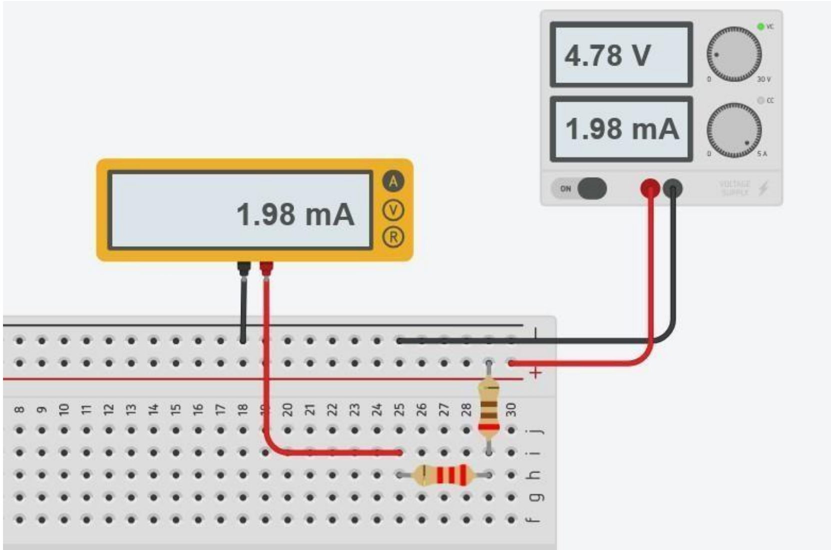
5. After the V th and R_{th} values are obtained, make a replacement circuit for Thevenin, then measure the current values on the a-b parameter resistors with different values (1kΩ, 2k2Ω, and 470Ω) using a multimeter . An example of the circuit is as follows.



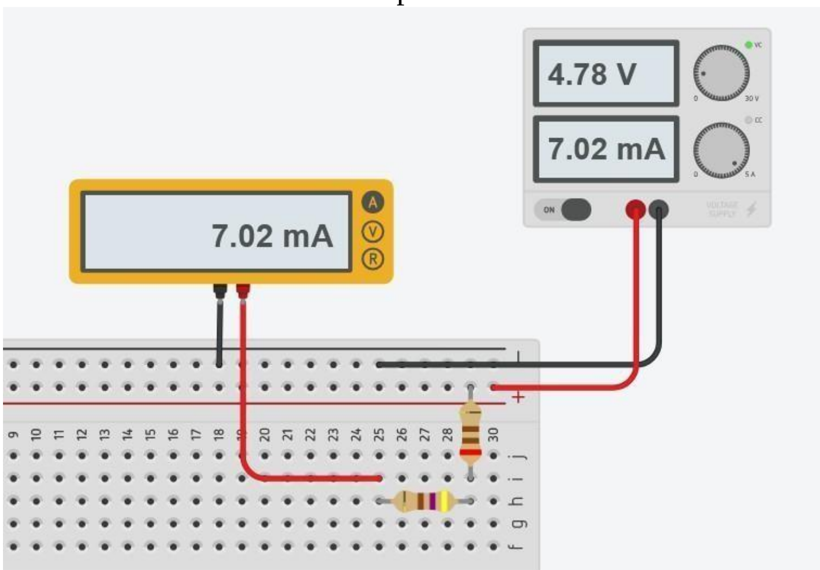
a. 1k Ω resistor as parameter a-b:



b. 2k2 Ω resistor as parameter a-b:



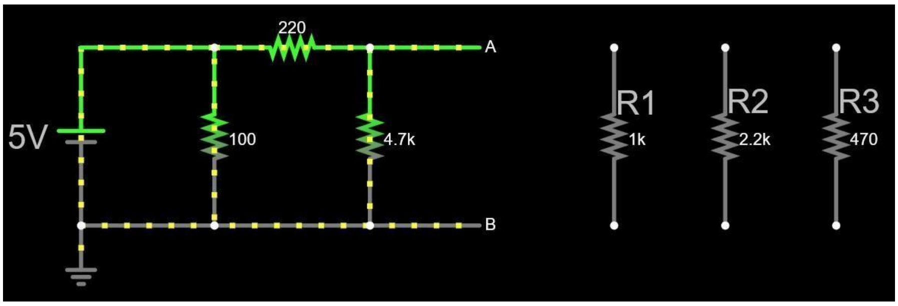
c. 470Ω resistor as parameter a-b:



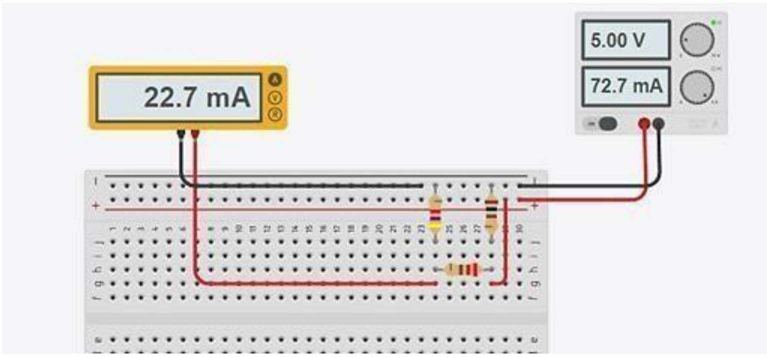
6. Record the measurement results in a journal and compare the results with measurements directly.

3.4.3 Norton's theorem

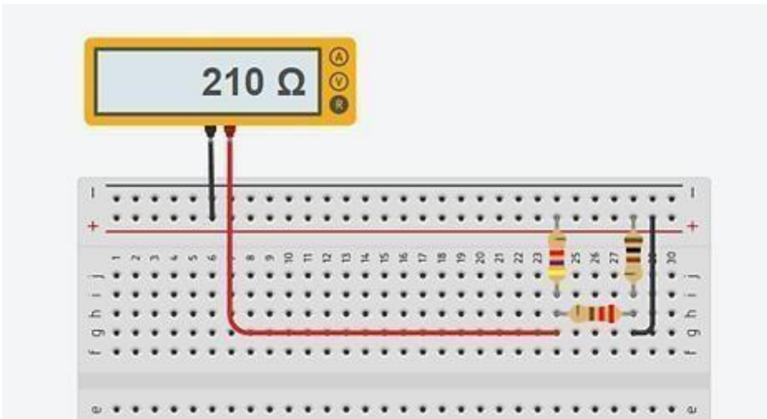
1. Make a circuit with the following scheme.



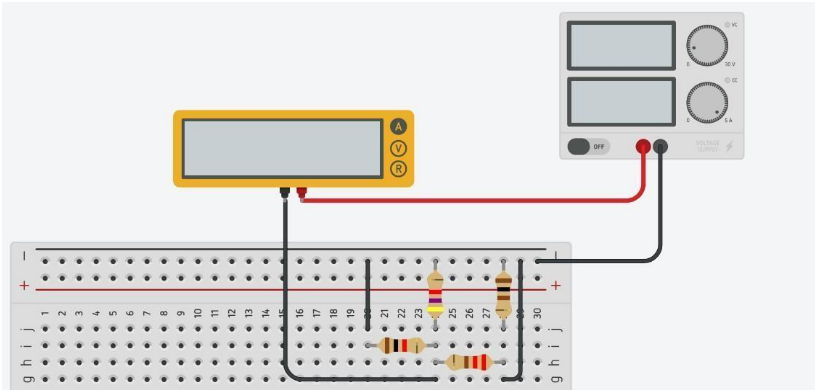
2. Remove the parameter resistors a-b, then connect point a to point b using a jumper cable (*short circuit*).
3. Calculate the value of I_N by measuring the current at point a to point b using a multimeter. An example of the circuit is as follows.



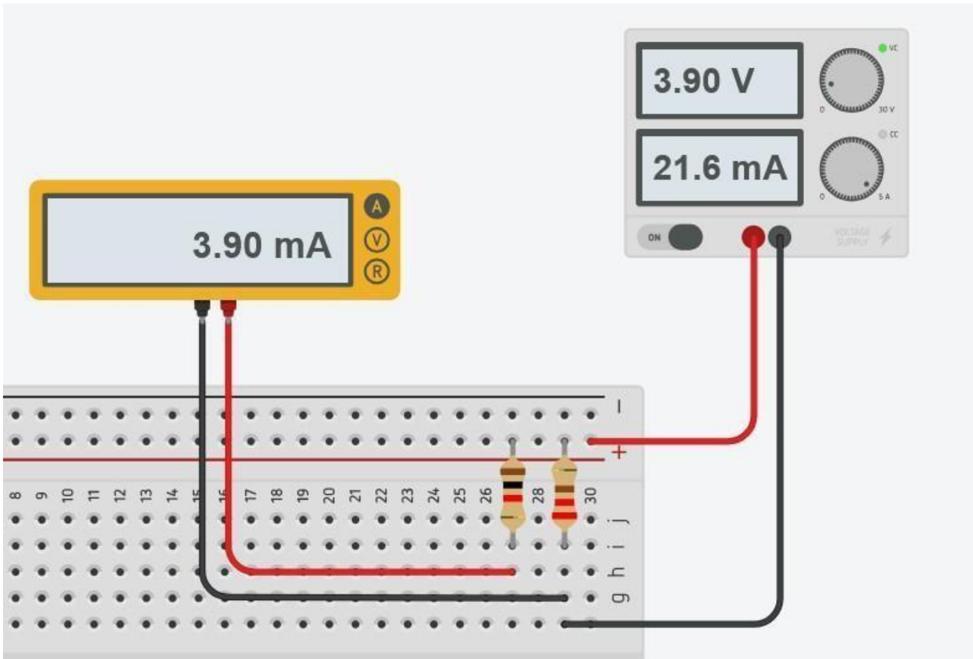
4. Disconnect the voltage source and make a short circuit using a jumper, then calculate the R_N value with a multimeter. The example circuit is as follows (R_N value is always equal to Rth).



5. Create a voltage source (V_{ds}) in place of the Norton current source by connecting an ammeter and a 1kΩ parameter a-b resistor to the circuit with the following example scheme and circuit.



6. Set the *power supply* knob until the ammeter shows the IN value previously obtained. After that, record the value of *the power supply* voltage (Vds) in the journal.
7. Change the resistor values of parameters a-b to 2k2Ω, then repeat Step number 6.
8. Change the resistor value of parameter a-b to 470Ω, then repeat Step number 6.
9. After the V ds value is obtained, create a Norton replacement circuit for the 1k parameter a-b resistor Ω with the following scheme.



10. Observe the current value on the multimeter. Record the value in your journal.
11. Repeat steps 9 and 10 for parameter resistors a-b , 2k2, Ω and 470Ω (V_{ds} values are adjusted to parameter values a-b).
12. Compare the results with the measurements directly.

Module 4
Impedance and Transfer Function

4.1 Purpose

After attending this practicum, students are expected to:

1. Able to operate oscilloscope measuring instruments and function generators.
2. Understand the definition of Impedance and Transfer Function.
3. Knowing the difference in the wave of the differentiator and integrator circuits, as well as the phase differences that occur in RC or RL series circuits.
4. Study frequency response, *cut-off* frequency and can find out the types of filters that occur in RL or RC circuits.

4.2 Tools and Materials

1. Multimeter
2. Oscilloscope
3. Function Generator
4. Project Board
5. Probe
6. Jumper
7. Resistors10 kΩ and 4,7 kΩ
8. Capacitors 22 nF
9. Inductor 10 mH

4.3 Theoretical Basis

4.3.1 Introduction to Measuring Instruments (Oscilloscopes and Function Generators)

A. Oscilloscope

An oscilloscope is a measuring instrument that serves to project frequencies and electrical signals in the form of graphs. The waveform of the measured electrical signal will be a graph of amplitude in the time domain. Oscilloscopes are of two types, namely oscilloscopes analog and oscilloscope digital. In analog oscilloscopes, the waves to be displayed are *real time* while in digital oscilloscopes the waves to be displayed are *sampled* or extracted first and then digitized. **In this practicum we will use a digital oscilloscope.**

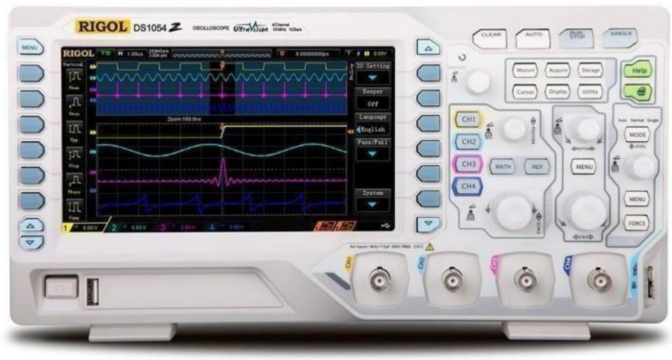


figure 4. 1: Oscilloscope

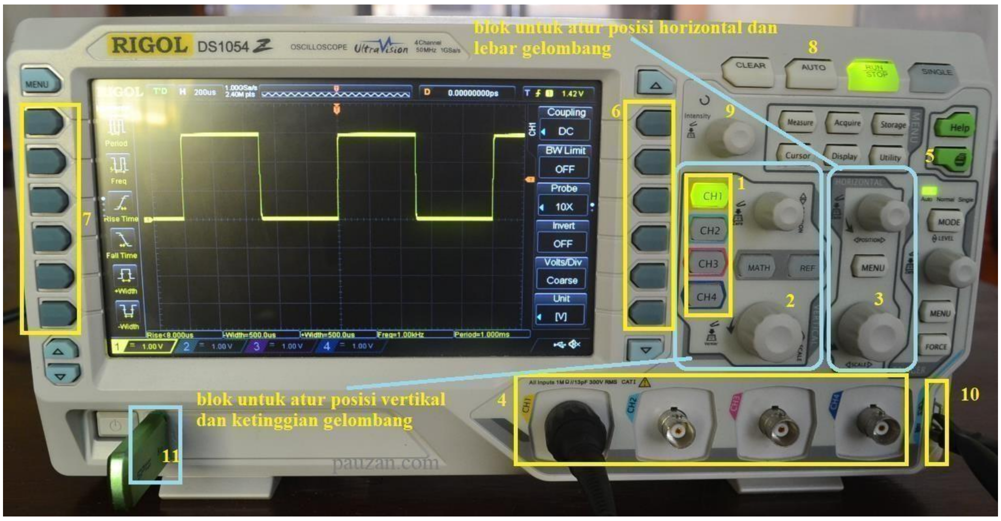


*figure 4. 2: **Oscilloscope parts.***

- **Number 1** is a collection of buttons for selecting channels. if you want to display waves on *channel* 1 then CH1 is suppressed, as well as others. According to the number of *channel buttons*, you can display two to four waves at a time
- **Number 2** is the block of oscilloscopes to arrange signals vertically. The top knob is a "*position"* knob that functions to adjust the wave height displayed on the oscilloscope screen.
- **Number 3** is the block of oscilloscopes to regulate the signal horizontally. The top knob is a "*position"* knob that functions to

adjust the horizontal position (left-right) of the waves displayed on the oscilloscope screen.

- **Number 4** is the place to connect the probe cable from channel 1 to *channel* 4 oscilloscope.
- **Number 5** is a "*print"* sign that serves as a button to save the wave image *file* of the oscilloscope display. This image *file* can later be stored in a *flash drive.*
- **Number 6** is the block of parts on the oscilloscope that is used to adjust starting from the bandwidth limit (BW limit), to display units of wave magnitude (V_{rms}, Volt / div, etc.).
- **Number 7** is the block of parts on the oscilloscope used to adjust the period, frequency, *rise* and *fall time,* to the width of the wave.
- **Number 8** is an "*auto"* button that serves to display waves in *the default* scale automatically both in terms of amplitude (Volt / div) and period (time / div).
- **Number 9** is an "*intensity"* know that functions to adjust the brightness level of the wave display on the oscilloscope screen.
- **Number 10** is a link / terminal that functions as a terminal connected to the oscilloscope *channel* during the calibration process.
- **Number 11** is the place to install a *flash drive.*

As for some measurements that can be done on an oscilloscope include:

1. Alternating Voltage Measurement

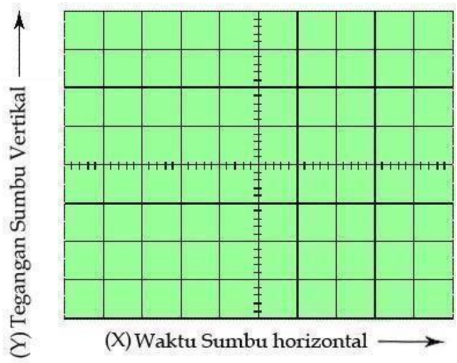


figure 4. 3: Description of the oscilloscope plot

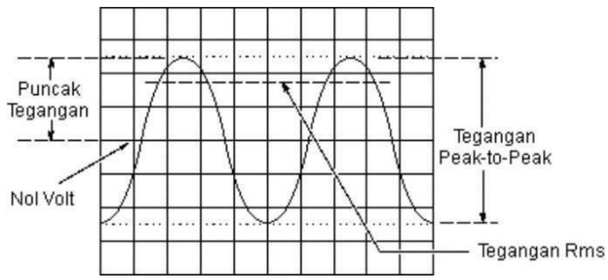


figure 4. 4: Oscilloscope wave display

Information:

- $\boldsymbol{Nilai\ Puncak} = \frac{0{,}05\ Volt}{DIV} \times 2{,}5\ DIV = 0{,}125\ VPP$
- $\boldsymbol{Nilai\ Puncak}\ (10\ probe) = \frac{0{,}05\ Volt}{DIV} \times 2{,}5\ DIV \times 10 = 125\ VPP$
- $\boldsymbol{Tegangan\ Peak\ to\ Peak}\ (VPP) = \frac{0{,}05\ Volt}{DIV} \times 5\ DIV = 0{,}25\ Volt$
- DIV *is* a unit for one square on an oscilloscope (example: 2.5 DIV at peak value means the wave height from zero to peak is 2.5 squares).
- $\boldsymbol{Harga\ Efektif} = \frac{Vpp}{2\sqrt{2}} = \frac{0{,}25}{2\sqrt{2}} = 0{,}0883\ Volt$

2. Frequency Measurement

Frequency measurement can be done directly using the frequency formula ($f = \frac{1}{T}$), where $T = \frac{TIME}{DIV} \times$ number of DIV $(horizontal)$ with T in seconds.

Frequency measurement can also be performed by connecting the component to be measured to channel 1 of the oscilloscope and connecting a function generator with a known frequency to channel 2 of the oscilloscope. Then we just set The frequency of the function generator until the wave period is equal to the wave period to be measured frequency . If it is the same, it can be ascertained that the frequency value in the function generator is the same as the frequency of the measured wave.

3. Phase difference measurement

Phase difference measurement can be done by connecting the first signal to channel 1 and the second signal to *channel 2*. After the image of the two waves is visible, the phase difference can be calculated with the following description:

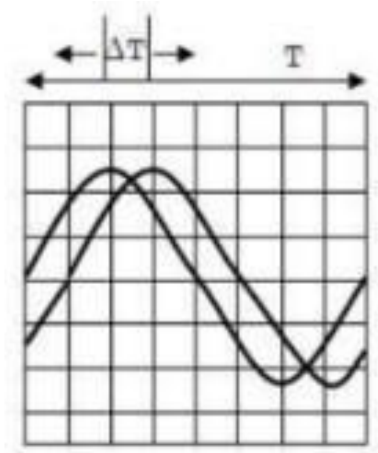


figure 4. 5: Different wave phases

- Phase Difference ($\Delta\phi$) $=\Delta T \times 360°$

Description:

- $\Delta\phi$ = phase difference
- ΔT = Time interval between waves(seconds)
- T = period (seconds)

B. Function Generator

A function generator is an electronic measuring instrument that generates or produces waveforms such as sine, triangle, ramp, square, and pulse waveforms. Typically, the waveforms generated by the function generator are displayed on the oscilloscope screen after one of the oscilloscope channels is connected to the function generator using a probe cable. Below is an example of the display of a Rigol brand function generator, model DG1022.

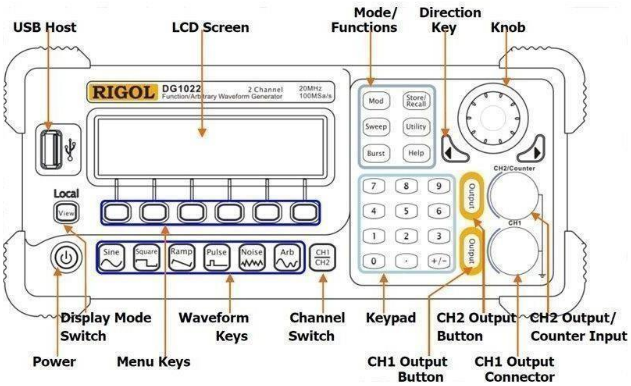


figure 4. 6: ***Display function generator.***

4.3.2 Impedance

In the analysis of an AC circuit, the term impedance means the total resistance of a circuit given an AC input signal and has a unit of Ω (Ohms). How to calculate impedance is the same as how to calculate resistance, except that the capacitor and inductor components are changed in resistance form to:

- Capacitors : $\frac{1}{j\omega C}$ (Ω)
- Inductor : $j\omega L$ (Ω)
- Resistor : R (Ω)

Description:

- j = imaginary number
- ω = angular velocity (rad/s)
- R = resistance (Ω)
- L = inductance (H)
- C = capacitance (F)

4.3.3 Transfer Function

The transfer function is a comparison between the amount of output and input, both in the form of a ratio of voltage to voltage, current to current, voltage to current and vice versa. The benefit of knowing the transfer function is to look for voltage gain, current gain, and input and output impedance. The general formula of the transfer function is as follows.

$$H(s) = \frac{besaranoutput\ (s)}{besaraninput\ (s)}$$

4.3.4 Frequency Response

Frequency response is the steady-state response of a system to the input of sinusoidal waves. This is closely related to the wave filter that will be studied in the next segment, where the wave will be passed or muted until it finally reaches a steady state *state*). Obtaining the frequency response can be done by entering the magnitude function formula as follows.

$$|H(j\omega)| = \sqrt{(Re(H(j\omega)))^2 + (Im(H(j\omega)))^2}$$

By knowing the frequency response, we can find out the type of filter of a circuit.

4.3.5 Cut-Off Frequency

The *cut-off* frequency is the frequency that causes the magnitude function $|H(j\omega)|$ value $\frac{1}{\sqrt{2}}$. The *cut-off* frequency is used as a reference point for the frequency range to be muted or passed. With this reference, we can determine or know the type of filter in a circuit.

4.3.6 Filter

A filter is a combined circuit of capacitors, inductors, and resistors that based on their cut-off frequency can miss or dampen a certain frequency region. There are four general types of filters to know, namely band *stop* filters, band pass filters, low pass filters, and *high pass* filters.

A. *Band Stop Filter* (BSF)

Band stop filter (BSF) is a filter used to suppress a certain frequency range and pass frequencies outside that *range*. Frequency bandwidth is affected by the lower cut-off frequency value and the upper *cut-off* frequency value.

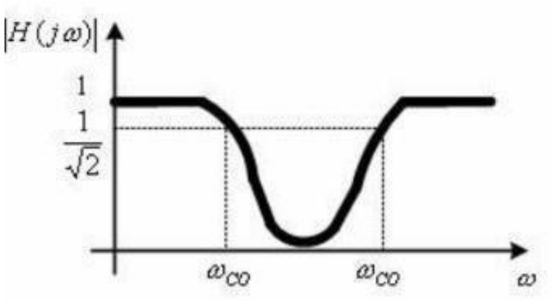


figure 4. 7: Band stop filter

B. *Band Pass Filter* (BPF)

Band pass filter (BPF) is a filter used to skip a certain frequency range and suppress frequencies outside that *range*. Frequency bandwidth is affected by the lower cut-off frequency value and the upper *cut-off* frequency value.

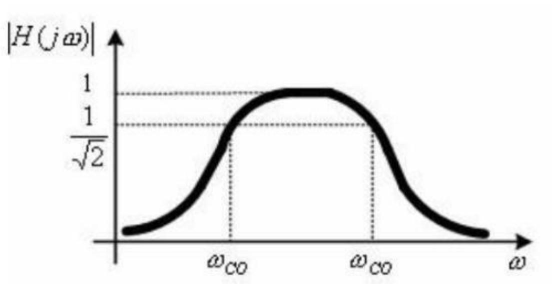


figure 4. 8: Band pass filter

C. *Low Pass Filter* (LPF)

Low pass filters (LPF) are used to skip all low frequencies up to their cut-off frequency and dampen other frequencies. Unlike BPF and BSF, this filter has only one *cut-off* frequency value. LPF can occur in two types of circuits, namely RL circuits with output in R and RC circuits with output in C.

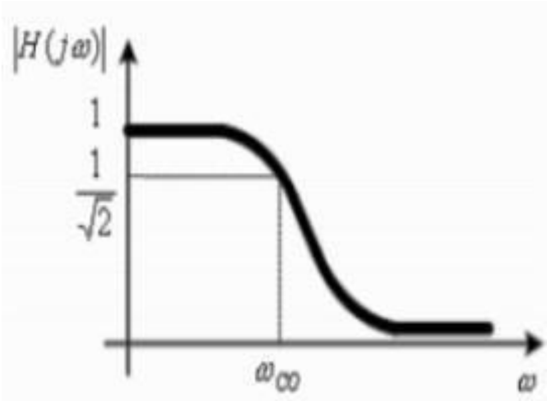


figure 4. 9: Low pass filter

D. ***High Pass Filter*** **(HPF)**

High pass filters (HPF) are used to dampen all low frequencies up to their cut-off frequency and miss other frequencies. Just like LPF, this filter has only one *cut-off* frequency value. HPF can occur in two types of circuits, namely RL circuits with output in L and RC circuits with output in R.

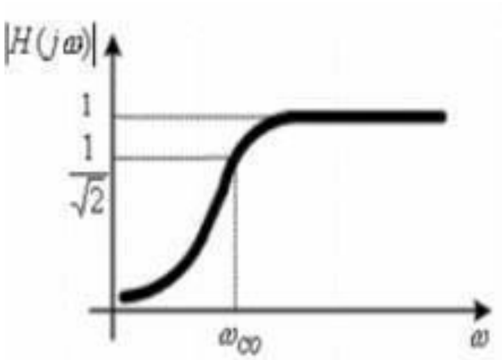


figure 4. 10: High pass filter

4.3.7 Waveforms of Circuit Differentiators and Integrators

In this module practicum, we will try to prove the change in the output wave of the integrator and differentiator circuit based on the input waveform. The circuit we use is the RC and RL circuit. Here is a table of input and output waveforms of RC and RL circuits.

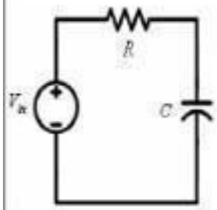
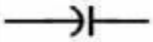
Bentuk Rangkaian	Kondisi	Komponen *Output*	Jenis sinyal	
			Input	*Output*
	*Differensiator*			
	Integrator			

Table 4. 1 RC Circuit

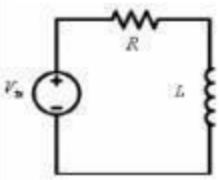
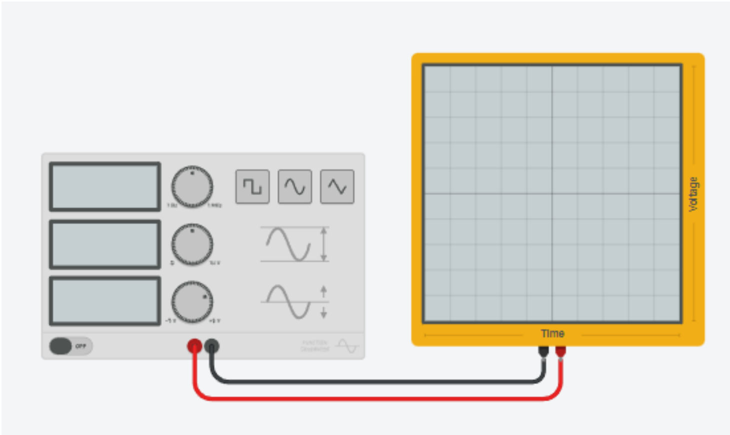
Bentuk Rangkaian	Kondisi	Komponen *Output*	Jenis sinyal	
			Input	*Output*
	*Differensiator*			
	Integrator			

Table 4. 2 RL Circuit

4.4 Practicum Procedure

4.4.1 Alternating Voltage Measurement

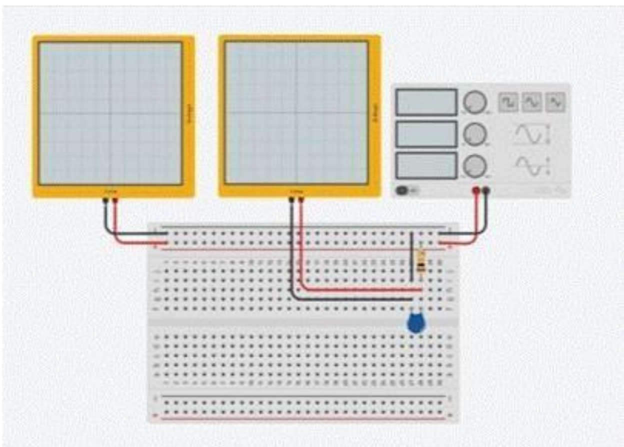
1. Create a circuit with the following schema and example .



2. Set the input voltage (V_{in}) on the function generator to 2 Vpp, then set the frequency on the function generator to 50 Hz.
3. Calculate the V_{rms} value and output frequency with the formula explained by the assistant.
4. Record the results of the calculation in a journal.
5. Change the input voltage value in the function generator to 3 Vpp with a fixed frequency at the same value (50 Hz).
6. Repeat steps number 3 and 4.

4.4.2 Measurement of Phase Difference on Component C with an Oscilloscope

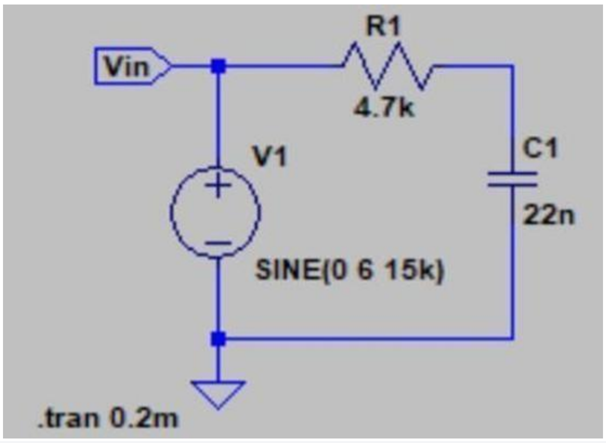
1. Create a circuit with the following scheme and example. Assume that the input wave is channel 1 of the oscilloscope and the output wave is channel 2 of the oscilloscope

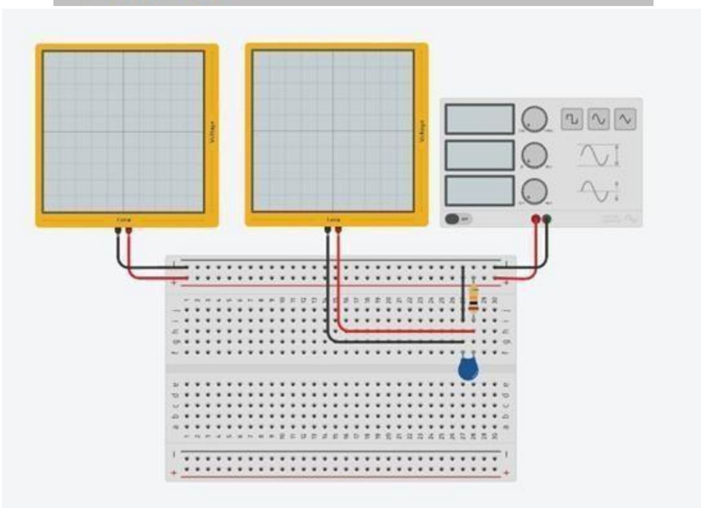


2. Set the frequency on the function generator to 15 kHz with an input amplitude (Vin) of 6 Vpp.
3. Calculate the phase difference between the input wave and the output wave using the formula or instructions provided by the assistant. Then record the calculation results in your journal.

4.4.3 Measuring Output C in RC Circuit (Integrator)

1. Build an RC circuit using a 4.7 kΩ resistor and a 22 nF capacitor, following the provided schematic. Assume the input waveform is oscilloscope channel 1 and the output waveform is oscilloscope channel 2.





2. Set the Vin amplitude on the function generator to 6 Vpp.
3. Calculate the theoretical output voltage value using the formula:

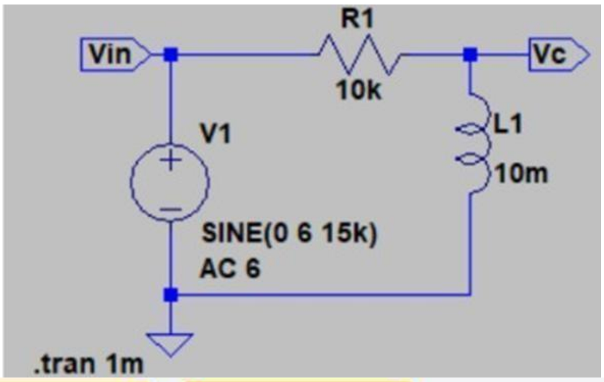
$$Vpp_{output} = \frac{Vpp_{input}}{\sqrt{2}}$$

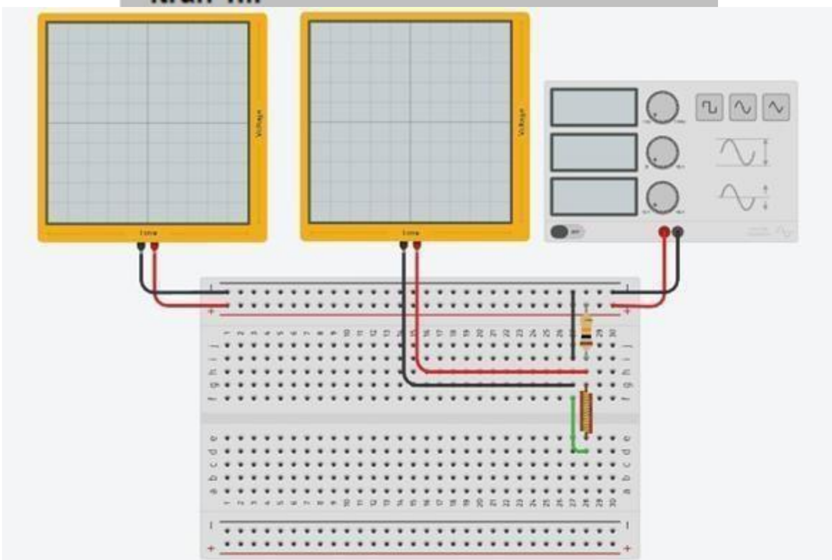
4. Press the output button on the function generator to start displaying the signal on the oscilloscope.

5. Observe the measure table display on the oscilloscope, then note the Vpp value.
6. Set the input frequency starting from 1 Hz and increase it gradually until the Vpp value read on the oscilloscope approaches the calculated value of the output Vpp.
7. Record the frequency when the output Vpp value read on the oscilloscope reaches the calculated value. The frequency value is the cut-off frequency of the circuit. Write down these observations in the journal.

4.4.4 Measuring Output L in RL Circuit (Differentiator)

1. Build an RC circuit using a 10 kΩ resistor and a 10 mH inductor, following the provided schematic. Assume the input waveform is oscilloscope channel 1 and the output waveform is oscilloscope channel 2.





2. Set the Vin amplitude on the function generator to 6 Vpp.
3. Calculate the theoretical output voltage value using the formula:

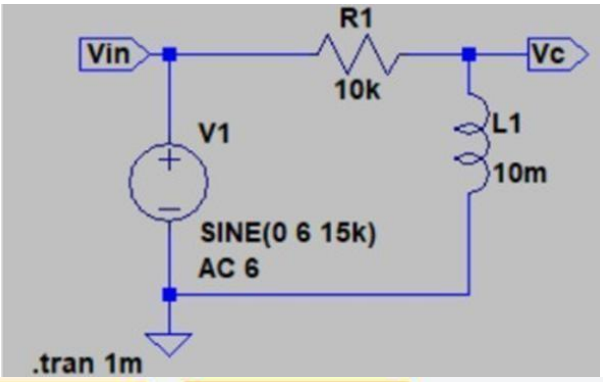
$$Vpp_{output} = \frac{Vpp_{input}}{\sqrt{2}}$$

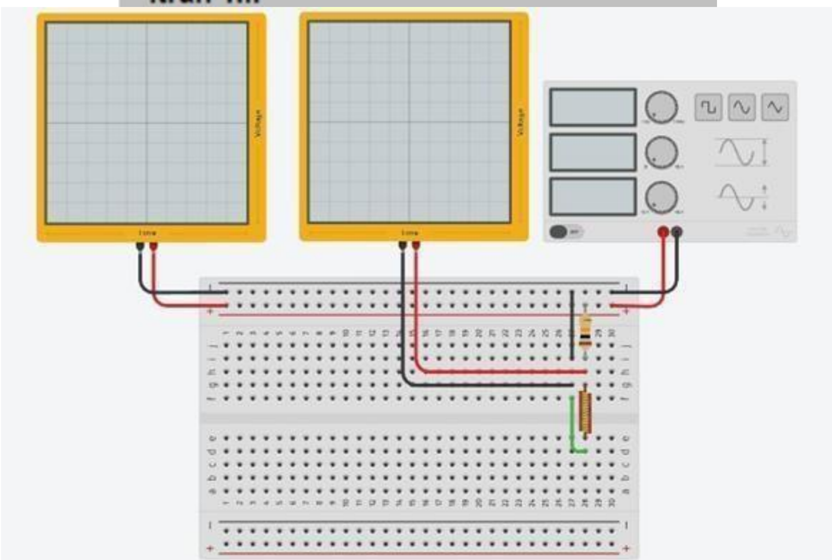
4. Press the output button on the function generator to start displaying the signal on the oscilloscope.
5. Observe the measure table display on the oscilloscope, then note the Vpp value.
6. Set the input frequency starting from 1 Hz and increase it gradually until the Vpp value read on the oscilloscope approaches the calculated value of the output Vpp.

7. Record the frequency when the output Vpp value read on the oscilloscope reaches the calculated value. The frequency value is the cut-off frequency of the circuit. Write down these observations in the journal

4.4.4 Measuring Output L in RL Circuit (Differentiator)

1. Build an RC circuit using a 10 kΩ resistor and a 10 mH inductor, following the provided schematic. Assume the input waveform is oscilloscope channel 1 and the output waveform is oscilloscope channel 2.





2. Set the Vin amplitude on the function generator to 6 Vpp.
3. Calculate the theoretical output voltage value using the formula:

$$Vpp_{output} = \frac{Vpp_{input}}{\sqrt{2}}$$

4. Press the output button on the function generator to start displaying the signal on the oscilloscope.
5. Observe the measure table display on the oscilloscope, then note the Vpp value.
6. Set the input frequency starting from 1 Hz and increase it gradually until the Vpp value read on the oscilloscope approaches the calculated value of the output Vpp.

7. Record the frequency when the output Vpp value read on the oscilloscope reaches the calculated value. The frequency value is the cut-off frequency of the circuit. Write down these observations in the journal.

Modul 5: Resonansi

5.1 Purpose

After participating in this practicum activity, students can:

1. Understand series, parallel, and parallel series resonance circuits.
2. Understanding band *stop* filters and *band pass filters*
3. Understand *bandwidth*, *cut-off* frequency, and quality factors.

5.2 Tools and Materials

1. *Function generator*
2. Oscilloscope
3. Resistor
4. Inductor
5. capacitors
6. Project board
7. Jumper

5.3 Theoretical Basis

5.3.1 Resonance

Resonance is a condition where the response value *of magnitude* $|H(j\omega)|$ has reached the maximum or minimum. A circuit is said to resonate when the voltage and current generated in the circuit are in one phase. Resonance occurs in AC current circuits that have passive components (resistors, inductors, and capacitors) caused by the resonant frequency (f_r). The characteristics of resonance are as follows.

1. The circuit is resistive.
2. The current phase is the same as the voltage phase.
3. There is an imaginary element.

In addition, there are several variables that are generally asked in a resonance circuit, namely wave quality factors (Q), *bandwidth*, and resonance frequency. The details are as follows.

1. *Quality Factor (Q)*

   The Quality Factor (Q) is a measure of the selectivity of the resonator circuit.The resonantor circuit is a series of *Band Pass Filter* (BPF) with a narrow bandwidth. The wider the Q value, the narrower the bandwidth. The formula for calculating the quality factor is as follows.

$$\mathbf{Q} = \frac{fr}{BW} = \frac{fr}{f2-f1} = \frac{\sqrt{f1.f2}}{f2-f1}$$

2. *Bandwidth*

   Bandwidth is the bandwidth of a wave. *Bandwidth* can also be said to be the difference between the upper cut-off frequency and the lower cut-off frequency.The formula is as follows.

$$\mathbf{BW} = \frac{fr}{Q} \quad \text{or } \mathbf{BW} = f2 - f1$$

3. Resonant Frequency

   Is the frequency that becomes the reference value when the circuit is resonant. The formula is as follows.

$$f_r = \sqrt{f1.f2}$$

- Description:

  Q = quality factor

  F1 = Bottom *cut-off* frequency

  F2 = Upper *cut-off* frequency

  f_r = resonant frequency

  BW = *bandwidth*

After knowing some variables in resonance circuits, we will study the types of resonance circuits, namely series, parallel, and parallel series resonance circuits.

5.3.1.1 Series Resonance Circuit

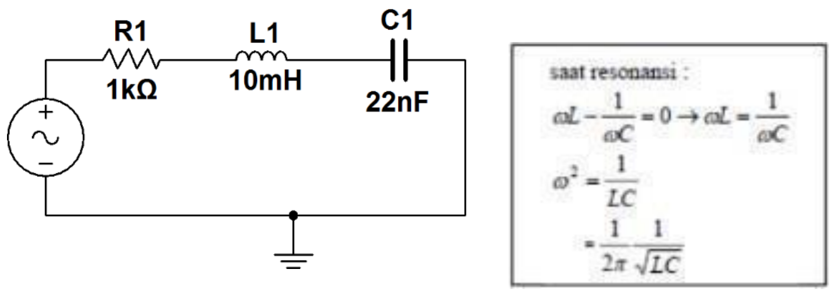


figure 5. 1: Series resonance circuit

The quality factor in the series resonance circuit can be written as follows.

$$Q = \frac{X_L}{R} = \frac{X_C}{R}$$

Where:

- X L (inductive reactance) = ω.L
- Xc (capacitive reactance) = $\frac{1}{\omega C}$
- ω = 2f
- R = resistance

As for calculating the total impedance of the series, the formula is as follows.

$$Z = \sqrt{R^2 + X_r^2} = \sqrt{R^2 + (X_L - X_C)^2}$$

If there is an imaginary number (j) then:

- X L (inductive reactance) = $j\omega L$
- Xc (inductive reactance) = $\frac{1}{j\omega C}$
- ω = 2f

The formula for total impedance when there are imaginary numbers is as follows

$$Z_{tot} = R + \left(j\omega L - \frac{1}{\omega C}\right)$$

In addition, there are also several things that must be considered in the series resonance circuit.

1. Impedance (Z) is a minimum value so that the current (I) is maximum.
2. If the output is in R, then the circuit will function as a *band pass filter* (BPF), where a certain frequency region will be passed and frequencies outside that region will be muted.

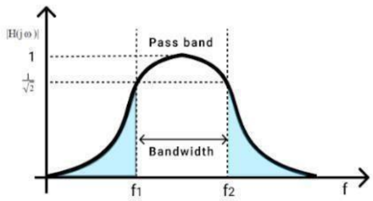


figure 5. 2: BPF

3. If the output is at points L and C (points B-C), then the circuit will function as a *band stop filter* (BSF), where certain frequency regions will be muted frequencies outside that region will be passed.

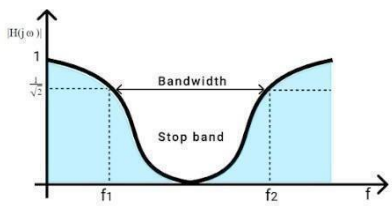


*figure 5. 3**: BSF***

B. Parallel Resonance Circuit

Parallel resonance occurs when inductive susceptance equals capacitive susceptance. Suspension is a function of frequency that measures the level of vulnerability of the circuit in conducting varying currents.

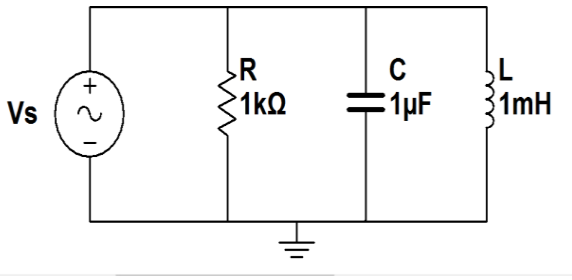


figure 5. 4: Parallel resonance circuit

$$\underbrace{Y_{tot}}_{\text{Admitansi}} = \overbrace{G}^{\text{Konduktansi}} + j\underbrace{(BC - BL)}_{\text{Suseptansi}} = Y = G + jB_C - jB_L$$

$$Y = G + j(\omega C - \frac{1}{\omega L})$$

saat resonansi :

$$\omega C - \frac{1}{\omega L} = 0 \rightarrow \omega C = \frac{1}{\omega L}$$

$$\omega^2 = \frac{1}{LC}$$

$$= \frac{1}{2\pi} \frac{1}{\sqrt{LC}}$$

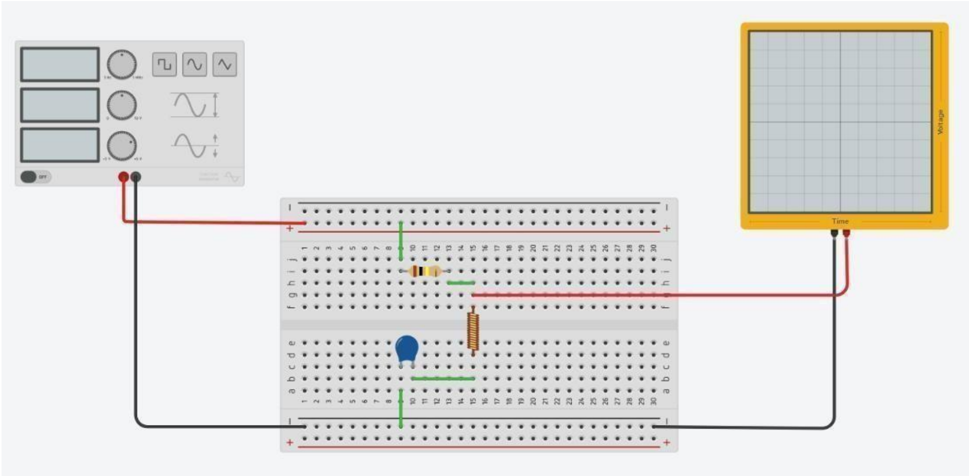
C. Series-Parallel Resonance Circuit

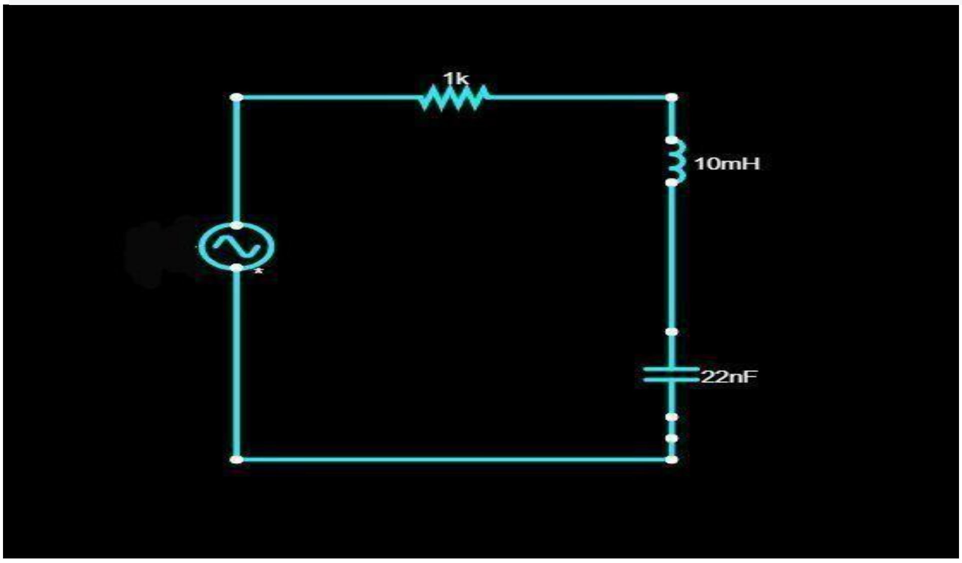
Series-parallel resonance is a resonance that occurs in RLC circuits which is also a combination of series and parallel relationships.

5.4 Practicum Procedure

5.4.1 RLC Series Family

1. Create a circuit with the following schema and example .





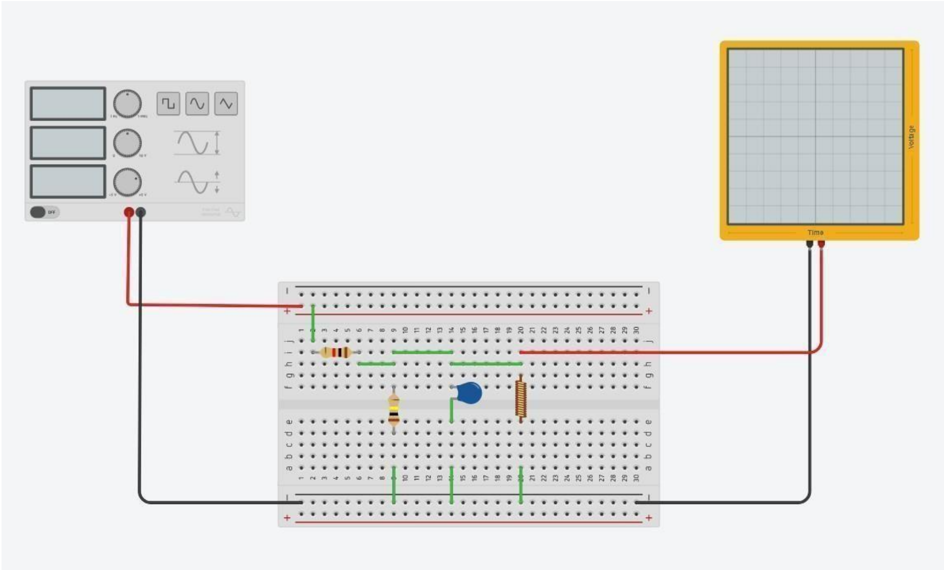
2. Connect the oscilloscope probe on the 10mH inductor leg and *ground* to measure the signal.
3. Connect the *output of the function generator* to the horizontal path as shown in the *breadboard* image above.
4. Set the function generator to output a sinusoidal signal with an amplitude of 1V.
5. Set *the function generator* frequency from 100Hz to 100kHz . Observe the magnitude of the signal amplitude measured by the oscilloscope and plot the magnitude of its voltage on a graph in the journal.
6. Find the top cut-off frequency and the bottom *cut-off* frequency by

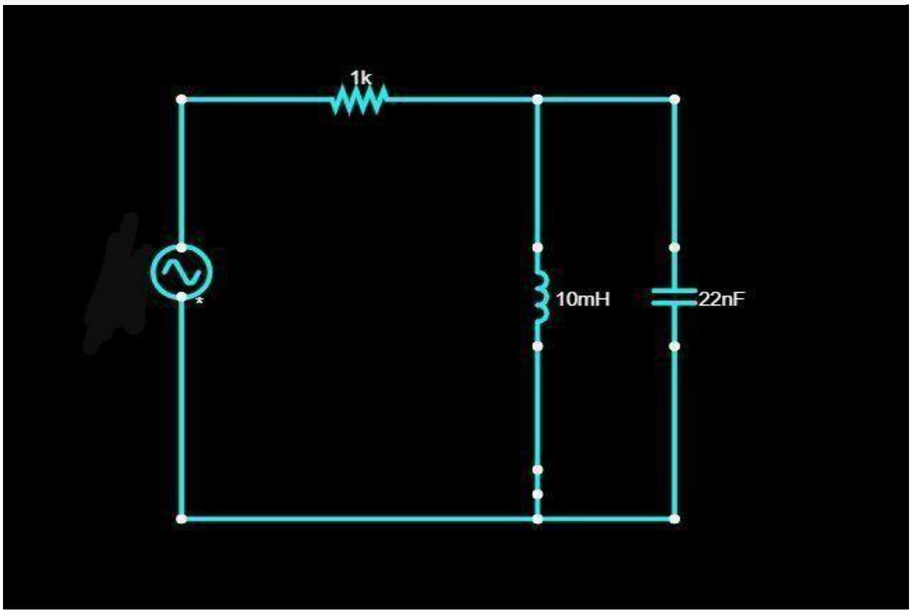
observing at what frequency between 100Hz to 100kHz has an amplitude of 0.7V.

7. Find the resonant frequency by observing the results of the graph plot and the measured amplitude magnitude change when changing the frequency from 100Hz to 100kHz or can use a formula that has been learned.
8. From the graph plot, determine the filter type, and then set the frequency of the *function generator* according to the resonant frequency.
9. Record the measured amplitude in the journal.
10. Calculate *bandwidth* as well as quality factor

5.4.2 RLC Parallel Circuit

1. Create a circuit with the following schema and example .





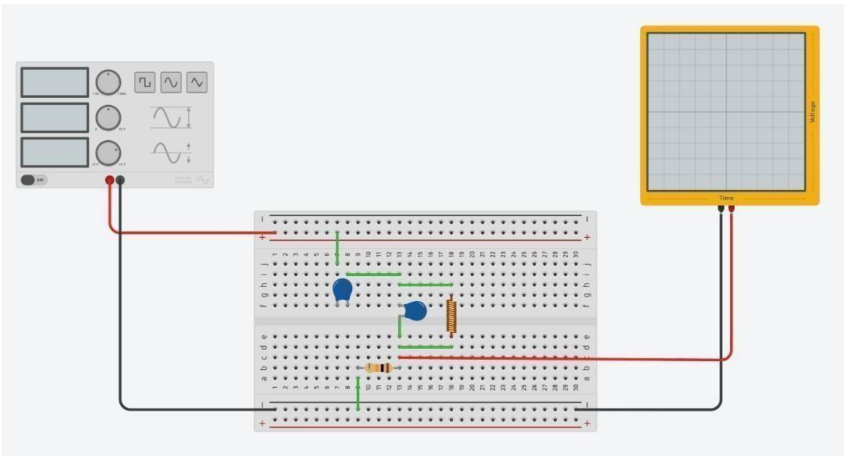
2. Connect the oscilloscope probe on the 10mH inductor leg and *ground* to measure the signal.
3. Connect the *output of the function generator* to the horizontal path as

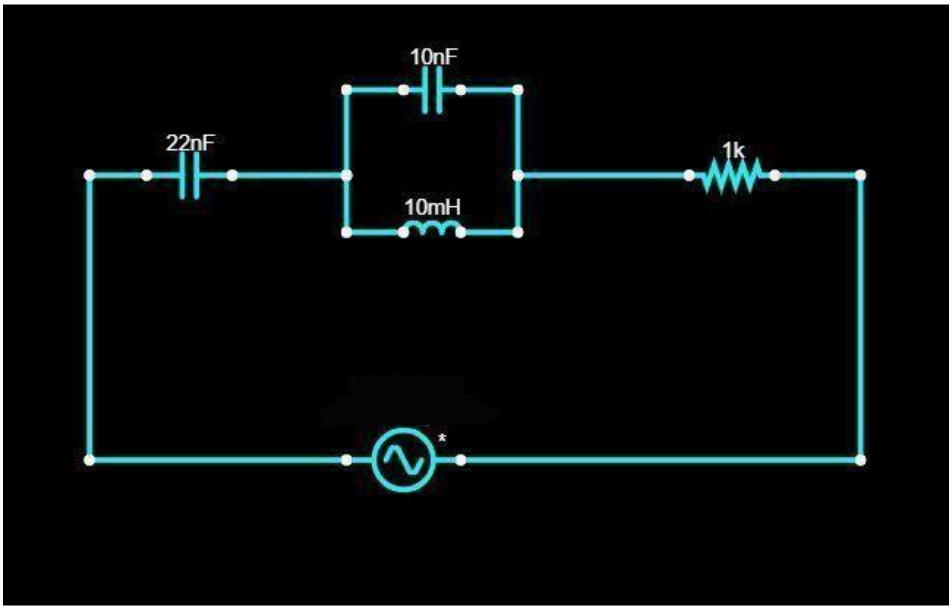
shown in the breadboard image above.

4. Set the function generator to output a sinusoidal signal with an amplitude of 1V.
5. Set *the function generator* frequency from 100Hz to 50kHz. Observe the magnitude of the signal amplitude measured by the oscilloscope and plot the magnitude of its voltage on a graph in the journal.
6. Find the top cut-off frequency and the bottom *cut-off* frequency by observing at what frequency between 100Hz to 50kHz has an amplitude of 0.7V.
7. Find the resonant frequency by observing the results of the graph plot and the measured amplitude magnitude change when changing the frequency from 100Hz to 50kHz or can use a formula that has been learned.
8. From the graph plot, determine the filter type, and then set the frequency of the *function generator* according to the resonant frequency.
9. Record the measured amplitude in the journal.
10. Calculate *bandwidth* as well as quality factors.

5.4.3 Parallel Series Circuit

1. Create a circuit with the following schema and example .





2. Connect the iloscope os probe to the 10K resistor leg and *ground* to measure the signal.
3. Connect the output *of the function generator* to the horizontal path as shown in the *breadboard* image above.
4. Set *the function generator* to output a sinusoidal signal with an amplitude of 1V.
5. Set *the function generator* frequency from 100Hz to 50kHz. Observe the magnitude of the signal amplitude measured by the oscilloscope and plot the magnitude of its voltage on a graph in the journal.

Modul 6: Two-port Theorem

6.1 Purpose

Through this module practicum, students are expected to:

1. Understand the definition of the four-pole theorem .
2. Calculates the parameters of two port networks.
3. Measure the parameters of a two port networks whose contents are unknown.
4. Determine the properties of the four poles based on their parameters.

6.2 Tools and Materials

1. Power Supply
2. Multimeter
3. Breadboard
4. Resistor 100Ω, 220Ω, 4k7Ω
5. Jumper Cable
6. Probe Cable

6.3 Theoretical Basis

A four-pole circuit (K-4) is a circuit that has one pair of terminals on the input side and another pair on the output side . The four-pole circuit theory is widely used in communication systems, control systems, power systems and electronic circuits.

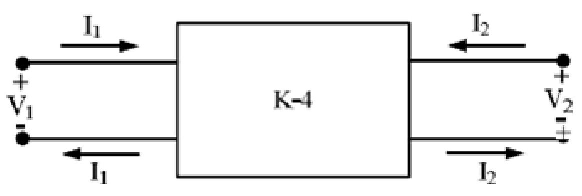


figure 6. 1: Four-pole circuit scheme

In the Two-port circuit, various kinds of relationships between voltage and current are needed called parameters. There are several kinds of parameter analysis that can be used, namely:

1. Parameter Z (Impedance)
2. Parameter Y (Admittance)
3. h&g (Hybrid) parameters
4. ABCD (Transmission) Parameters

6.3.1 Parameter Z (Impedance)

The parameter Z is also known as impedance or resistance. The Z parameter is commonly used in filter synthesis, *impedance matching* network analyzers and power system distribution. A four-pole circuit exists with voltage sources and current sources. Here are pictures of simple blocks of a four-pole circuit with parameter Z.

Two-port circuit with current source.

figure 6. 2: The poles of the four current sources.

Two-port circuit with a voltage source.

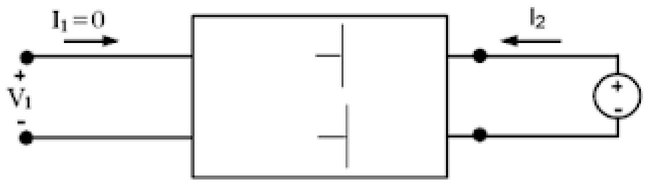


figure 6. 3: Two port networks voltage source

An example of a Two-port theorem in Z parameter.

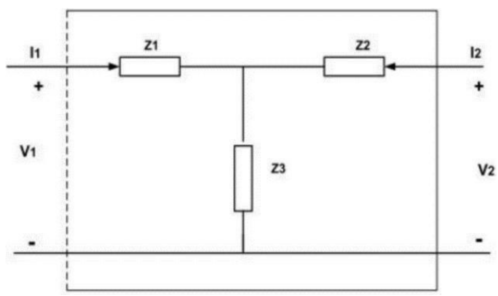


figure 6. 4: Z parameter set

The form of the voltage relationship in this Z parameter is as follows.

$$V_1 = Z_{11}I_1 + Z_{12}I_2$$
$$V_2 = Z_{21}I_1 + Z_{22}I_2$$

To determine the price of the Z parameter this can be done by conditioning port 1 open circuit and *port 2 open circuit* alternately. Two-port analysis with parameter Z can be done by looking at two sides, namely the primary side (V_1) and the secondary side (V_2). The analysis is as follows:

If port 2 is open circuit ($I_2=0$)

$$Z_{11} = \frac{V_1}{I_1} \Big| I_{2=0} \qquad Z_{21} = \frac{V_2}{I_1} \Big| I_{2=0}$$

If port 1 is open circuit ($I_1=0$)

$$Z_{12} = \frac{V_1}{I_2} \Big| I_{1=0} \qquad Z_{22} = \frac{V_2}{I_2} \Big| I_{1=0}$$

Information:

Z_{11} = primary port impedance when secondary port is open circuit

Z_{22} = primary port impedance when primary port is open circuit

Z_{12} = Z_{21} = transfer impedance where the ratio of voltage in one port versus current in another.

6.3.2 Parameter Y (Admittance)

Parameter Y is also known as the admittance or the level of ease of a circuit to be given an electric current. This makes the parameter Y the inverse of the parameter Z where . $Y = \frac{1}{Z}$ The Y parameter is widely used in filter synthesis, network *matching* analysis planning and power system distribution. An example of a simple block of a two port theorem with Y parameter is as follows.

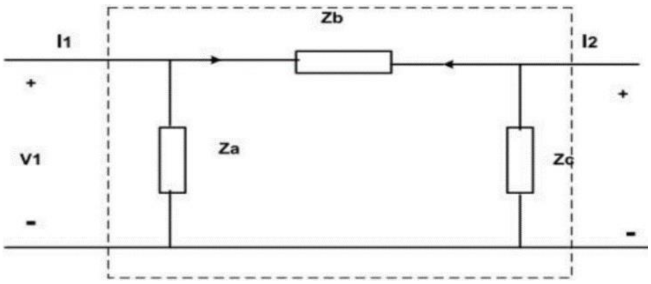


figure 6. 5: Y parameter set

The parameter Y shows the currents expressed in terminal voltage n with the following equation:

$$I_1 = Y_{11}V_1 + Y_{12}V_2$$

$$I_2 = Y_{21}V_1 + Y_{22}V_2$$

Similar to parameter Z, circuit analysis with parameter Y also looks at both sides, namely the primary side and the secondary side. The analysis is as follows;

- If port 2 *short circuit* (V_2=0)

$$Y_{11}= \frac{I_1}{V_1} \Big| V_{2=0} \qquad\qquad Y_{21}= \frac{I_2}{V_1} \Big| V_{2=0}$$

- If port 1 *is short circuit* (V_1=0)

$$Y_{12}= \frac{I_1}{V_2} \Big| V_{1=0} \qquad\qquad Y_{22}= \frac{I_2}{V_2} \Big| V_{1=0}$$

Description:

- Y_{11} = primary port admittance when secondary *port is short circuit.*
- Y_{22} = primary port admittance when primary *port is short circuit.*
- Y_{12} = Y_{21} = transfer admittance where the ratio of current in one *port* versus voltage in another.

The resulting admittance is a short circuit (SC) admittance or *a short circuit* parameter or Y parameter.

6.3.3 h&g (Hybrid) parameters

The *Hybrid* parameter is the Z parameter and the Y parameter combined. This parameter is commonly used in complex electrical modeling and design.

Parameter formula h

$$V_1 = h_{11}I_1 + h_{12}V_2$$
$$I_2 = h_{21}I_1 + h_{22}V_2$$

Where:

- $h_{11} = \frac{V_1}{I_1}|V_2 = 0$
- $h_{21} = \frac{I_2}{I_1}|V_2 = 0$
- $h_{12} = \frac{V_1}{V_2}|I_1 = 0$
- $h_{22} = \frac{I_2}{V_2}|I_1 = 0$

Parameter g Formula

$$I_1 = g_{11}V_1 + g_{12}I_2$$
$$V_2 = g_{21}V_1 + g_{22}I_2$$

Where:

$$g_{11} = \frac{I_1}{V_1}|I_2 = 0$$

$$g_{21} = \frac{V_2}{V_1}|I_2 = 0$$

$$g_{12} = \frac{I_1}{I_2}|V_1 = 0$$

$$g_{22} = \frac{V_2}{I_2}|V_1 = 0$$

6.3.4 Transmission Parameters (ABCD)

ABCD parameters use only variable voltages and currents without impedance or admittance. This parameter is commonly used in the analysis of transmission networks and modeling of various types of electrical equipment (transformers, *amplifiers* and so on). Each of these parameters has the following roles.

1. Parameter A connects the input and output voltages.
2. Parameter B measures the relationship between the input current and the output voltage.
3. Parameter C measures the extent to which wave energy comes from behind the device and creates voltage at the input poles.
4. Parameter D measures the extent to which wave energy coming from behind the device creates current at the input pole.

The formula for the ABCD parameter is as follows.

$$\boldsymbol{V_1 = AV_2 . BV_2}$$

$$\boldsymbol{I_1 = CV_2 . DV_2}$$

Where:

$$A = \frac{V_1}{V_2} \Big| I_{2=0} \qquad B = \frac{V_1}{-I_2} \Big| V_{2=0}$$

$$C = \frac{I_1}{V_2} \Big| I_{2=0} \qquad D = \frac{I_1}{-V_2} \Big| V_{2=0}$$

Description:

A: Voltage ratio when the secondary side *is open circuit.*

B: Transfer impedance when the secondary side *is short circuit.*

C: Transfer administration when the secondary side *is open circuit..*

D: current ratio when the secondary side *is short circuit.*

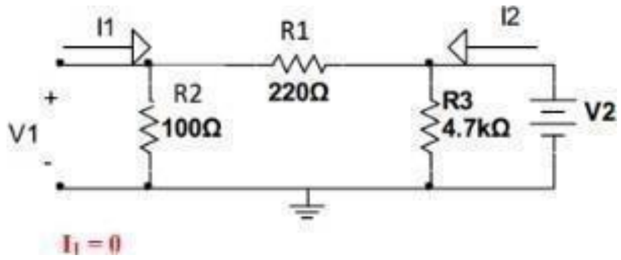
This parameter is important for transmission techniques because the primary side (sender) consists of variables V 1 and I_1. While the receiver consists of variables V 2 and I_2.

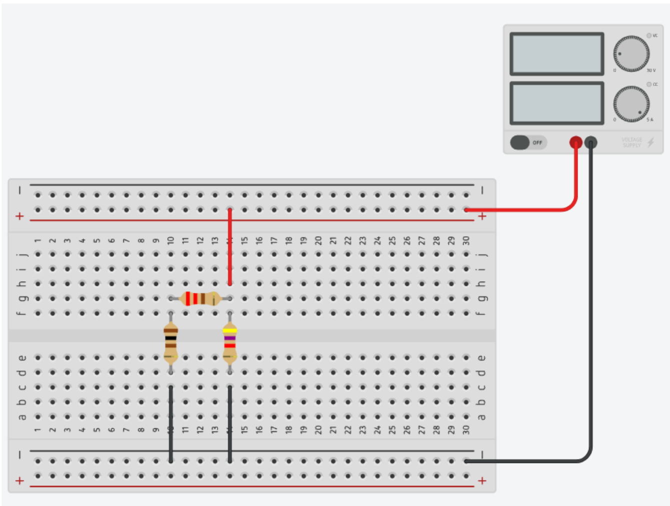
6.4 Practicum Procedure

6.4.1 Z Parameter Measurement

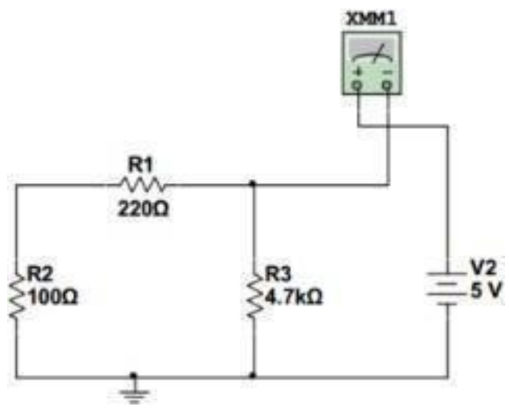
A. Condition 1 (V1 Open Circuit)

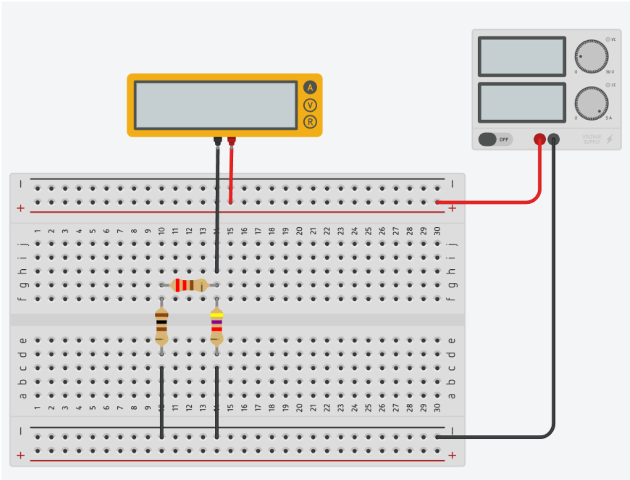
1. Create a circuit with V1 *open circuit* on breadboard with the following example and scheme.



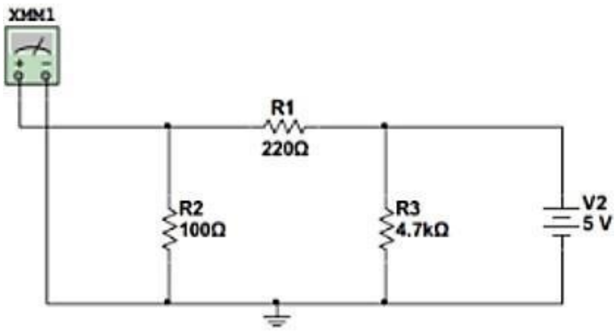


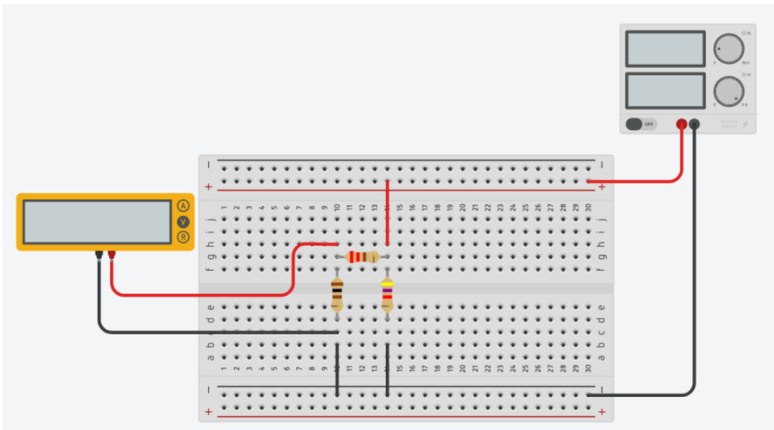
2. Set the *V2 power supply* voltage to 5V.
3. Measure the size of I2 by connecting the ammeter with a 5V breadboard line and *nodes* 4k7Ω and 220Ω. Examples of schemes and circuits are as follows.





4. Measure the size of V1 by measuring the voltage on a 100 resistor then connect the 5V breadboard line with nodes 4k7Ω and 220 Ω using a jumper. Examples of circuits and schemes are as follows.

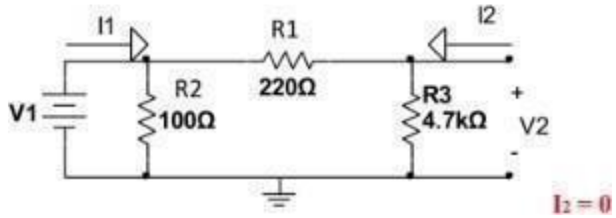




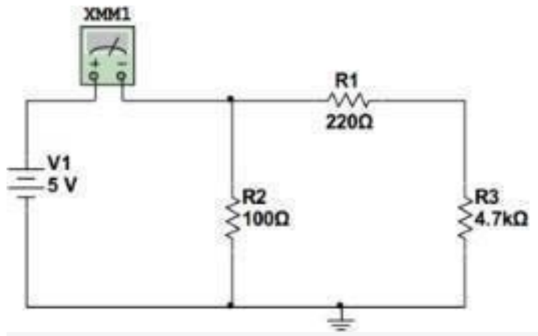
5. Record the measurement results I_2 and V1 in the journal and enter the values V2 = 5V and $I_1 = 0$.

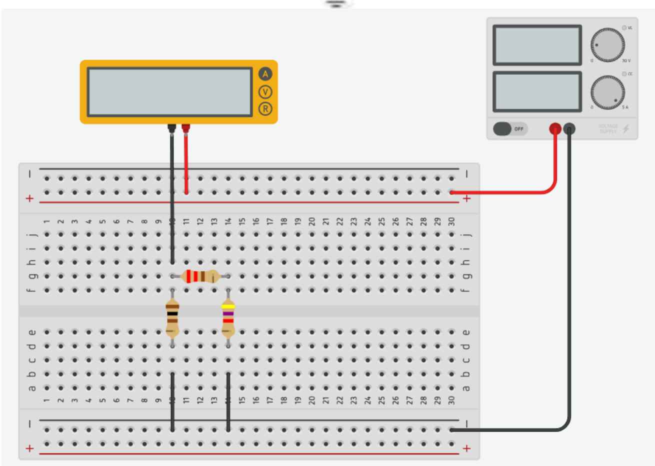
B. Condition 2 (V2 Open Circuit)

1. With the same circuit scheme with condition 1, transfer the 5V voltage to port V1, let V2 be the open circuit.

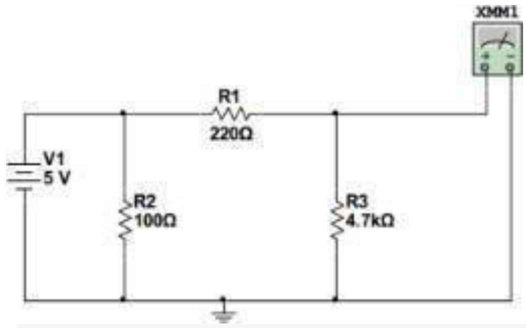


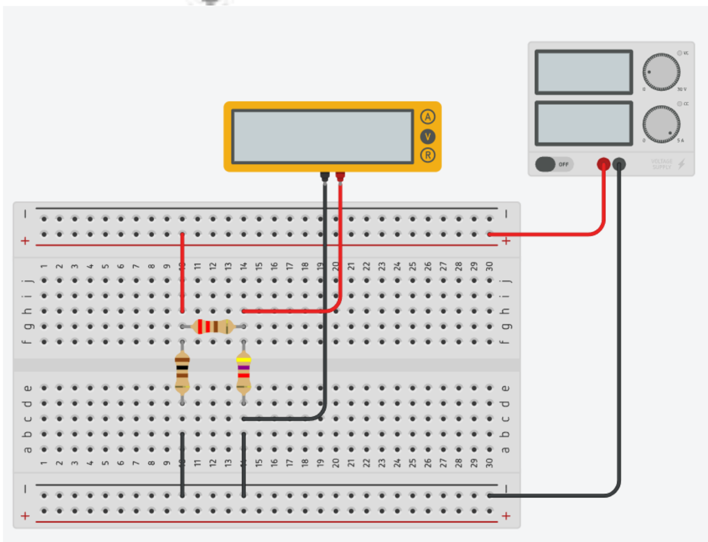
2. Measure the size of I 1 by connecting the ammeter with a 5V breadboard line and nodes 100Ω and 220Ω. Examples of schemes and circuits are as follows.





3. Measure the magnitude of V2 by measuring the voltage on a 4k7Ω resistor and connect the 5V breadboard line with nodes 100Ω and 220Ω with jumpers.



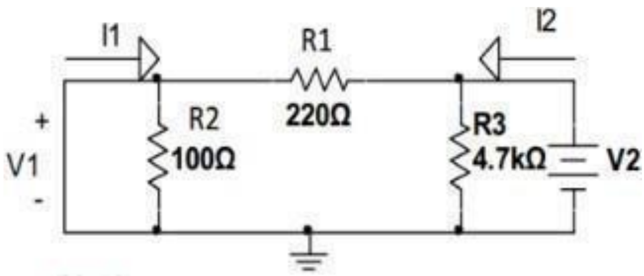


4. Record the measurement results I 1 and V 2 in the journal and enter the values V_1 = 5V and I_2 = 0.

6.4.2 Measurement of Parameter Y

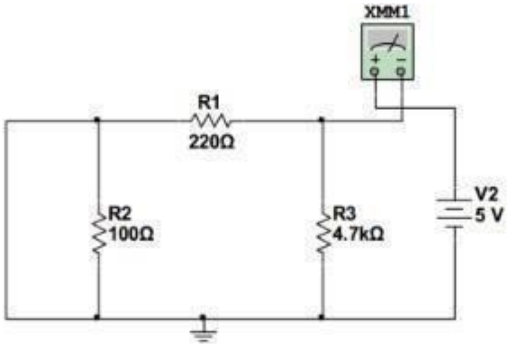
A. Condition 1 (V1 Short Circuit)

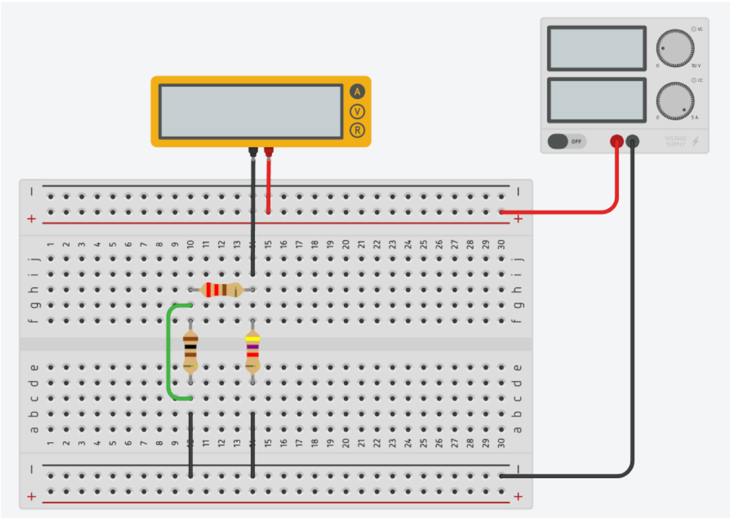
1. Make a circuit on the breadboard with the following scheme and circuit example.



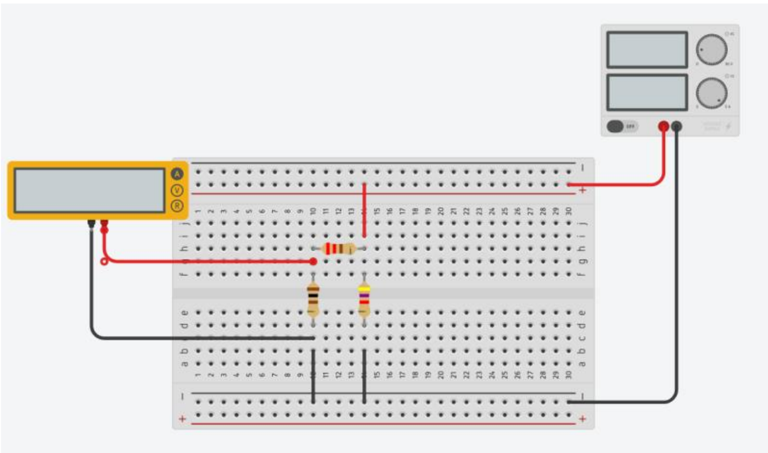
$V_1 = 0$

2. Measure the size of I2 by connecting the ammeter with a 5V breadboard line with nodes 4k7Ω and 220Ω. Short circuit the resistor 100Ω with the jumper.





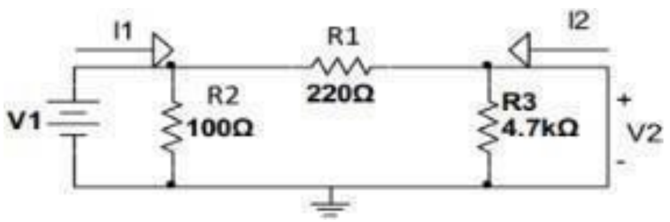
3. Measure the size of I1 by connecting the ammeter with nodes 220 Ω and 100Ω and ground. Connect nodes 4k7Ω and 220Ω with 5V breadboard lines with jumpers.



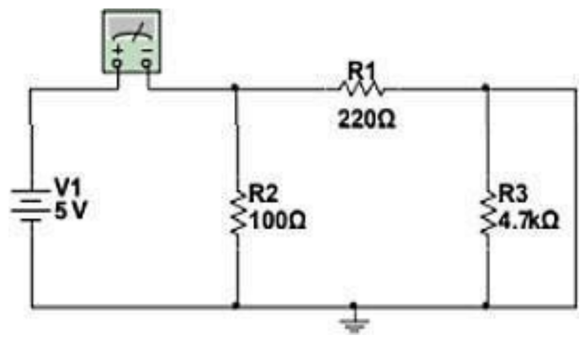
4. Record the measurement results I 1 and I 2 in the journal and enter the values $V_1 = 0V$ and $V_2 = 5V$

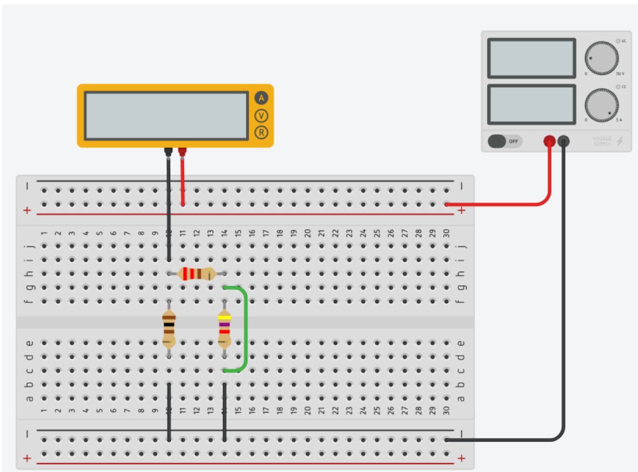
B. Condition 2 (V2 Short Circuit)

1. With the same circuit, move the power supply to port V2, then short-circuit port V1.



2. Measure the size of I1 by connecting the ammeter with a 5V breadboard line with 100 and 220 nodes. Short circuit the 4K7 resistor with a jumper.





3. Measure the size of I2 by connecting the ammeter with nodes 220 and 4K7 and ground. connect 100 and 220 nodes with 5V breadboard lines with jumpers.