

Symbulator

A symbolic simulator of linear electric circuits

Symbulator 9

Python / SymPy

Roberto Perez-Franco

Contents

Introduction	5
What is Symbulator	5
Why use Symbulator?	5
Get Symbulator	6
Input files and entries	7
 Direct current analysis	 9
1.1 Run a direct current analysis	9
1.2 How to describe a resistor	10
1.3 What answers do you get	11
1.4 How to describe a voltage source	12
1.5 A numerical DC simulation, step by step	13
1.6 Instructive numerical examples, solved	17
 Symbolic circuits and expert mode	 29
2.1 About symbolic problems	29
2.2 Numerical from symbolic, with solve	31
2.3 Numerical from symbolic, using ex	33
2.4 Instructive symbolic examples, solved	34
 Current sources, conductances and dependent sources	 39
3.1 How to describe a current source	39
3.2 What about conductances?	40
3.3 How to reduce parallel resistors	41
3.4 How to describe dependent sources	43
3.5 Instructive solved examples	44
 Shorts, equivalent resistance and Thévenin/Norton	 71
4.1 How to describe a short circuit	71

4.2	The equivalent resistance script: er	72
4.3	The Thévenin / Norton script: th	74
4.4	Instructive solved examples	78
	Operational amplifiers	107
5.1	How to describe an ideal operational amplifier	107
5.2	Instructive Op Amp examples	109
	Transient analysis	135
6.1	How to describe a capacitor	135
6.2	How to describe an inductor	136
6.3	Running a transient analysis	136
6.4	Two useful tools	138
6.5	Instructive solved examples	139
	Alternating current analysis	178
7.1	AC analysis in Symbulator	178
7.2	Solved numerical examples	180
7.3	Solved numerical-from-symbolic examples	189
7.4	Solved symbolic examples	190
	Power in AC circuits	193
8.1	Average power	193
8.2	RMS and PF	194
8.3	Complex power	195
	Three-phase circuits	200
9.1	About solving three-phase circuits	200
9.2	Wye-Wye	200
9.3	Wye-Delta	206
9.4	Delta-Delta	208
9.5	Delta-Wye	211
	Mutual inductance and ideal transformers	213
10.1	How to describe mutual inductance	213

10.2 Instructive m problems	214
10.3 How to describe ideal transformers	216
10.4 Instructive t problems	216
Bode plots and resonance	218
11.1 Do Bode plots	218
11.2 Resonance	220
The complex frequency domain	223
12.1 s-domain analysis	223
12.2 The t2s and s2t shortcuts	225
12.3 Instructive FD problems	225
12.4 Transfer function problems	226
Two-ports	227
13.1 Find two-port equivalents	227
13.2 Use two-ports as elements	228
13.3 Instructive two-port problems	229
13.4 The gain tool	233
Roll the credits	236
About the author	236
Acknowledgements	237
Problem credits	238
Praise received	239
Licence	240
Index	241

Introduction

Learn about Symbulator and how it can help you focus on learning how circuits work, instead of struggling with math. Get the latest version of Symbulator and get ready to roll.

What is Symbulator

Symbulator is a program to solve linear electric circuits symbolically and numerically on your device. It takes numerical and symbolic values, and returns numerical and symbolic results. The odd-sounding name is a portmanteau of “*symbolic simulator*”. Symbulator is widely regarded as the best symbolic simulator of linear electric circuits ever made for a handheld device.

What analyses can it do?

Symbulator can solve circuits using four types of analysis: direct current (DC), alternating current (AC), transient time-domain (TR), and complex frequency-domain (FD). It analyses numerical and, more importantly, symbolic linear circuits, returning the voltages in all nodes and the currents, voltage drops and power in all elements. It also finds the passive, Thévenin / Norton and two-port equivalent of a circuit, and draws time domain and Bode plots.

What elements does it accept?

Symbulator accepts elements such as resistors, inductors, mutual inductance, capacitors, independent and dependent current and voltage sources, ideal operational amplifiers, ideal transformers and six types of two-ports.

Where does it run?

Symbulator 9, released in 2026, is a port of the classic code to Python, powered by the SymPy computer algebra library. It runs anywhere Python runs: your mobile phone, tablet, laptop, a Jupyter notebook, or a browser on any computer. Since it is not bound by the memory of a calculator, version 9 solves circuits faster than versions 7 and 8.

Why use Symbulator?

Symbulator is extremely useful on the circuit theory problems engineering students face in Circuits I and II. It lets the student focus on understanding circuit analysis rather than on the mathematics of solving it.

Strengths

- A fantastic tool to solve circuits composed of ideal, linear elements.
- Shines in solving circuits with symbolic values, and symbolic results.
- Offers the absolute simplest way to define dependent sources.
- Includes ideal transformers and six different two-ports.

Limitations

- Does not simulate non-linear elements, such as diodes and transistors.
- Will not answer under-defined problems that rely on applying formulas selectively rather than solving a circuit completely.
- Is not a replacement for your brain or an excuse to not study. You must understand circuit theory to use it.

Symbulator is free!

Symbulator has always been free of cost, and always will be. *(You are welcome!)*

Since 2026, Symbulator is also open source under the MIT licence. You are free to use it, change it and port it to other platforms; all the licence asks is that the copyright notice and the attribution to the original author travel with it.

Get Symbulator

You can use Symbulator 9 in two ways:

- as an online app running on a remote server, or
- as an offline app running locally on your device.

The easiest way to start is the online app, currently hosted at PythonAnywhere. Click **here** to go to it.

To install Symbulator 9 as a local app that runs on your device even offline, follow the instructions at the bottom of the online app, under “**Run Symbulator 9 locally**”.

CAUTION

Use at your own risk!

Every effort has been made in the development of this software, and there are no known bugs in it. However, Symbulator is provided “as is”, without warranty of any kind, express or implied, including but not limited to the warranty of fitness for a particular purpose. Every time you use Symbulator, you do so at your own risk. Please report any problems to help@symbulator.com

Input files and entries

Everything you put into Symulator — the circuit, the analysis, the settings, and whatever is in the **Evaluate** and **Solve** cards — can be kept and used again. There are two words for it.

- An **entry** is one set of inputs meant to be solved together: a circuit, the analysis to run on it, and the settings to run it with, along with a name.
- An **input file** holds as many entries as you like, along with a name, such as *Circuits for Lesson 1 of the Tutorial*, and that is what you will see it called.

Input files are plain text and require the extension **.cir**. Using them is optional; the buttons to load, download and create them are in the **Input File** card at the top of the app.

Loading input files

Symulator arrives with input files of its own: one for each lesson in this tutorial, holding an entry for every simulation that lesson runs. It is not always one entry per problem — a few problems take more than one simulation, and each gets its own entry — so a lesson file may hold more entries than the lesson has problems. Between them, they let you follow the whole tutorial without typing a circuit at all.

Built-in examples lists the input files available for the lessons; click a title to load it. That link is always there. These files are **read-only**: load one and change it as much as you like, but you cannot write back to it.

To open an input file of your own, click **Upload** and choose the **.cir** file. The dropdown beneath then reads *Entries in* followed by the file's name. Choosing an entry fills in every box on the page at once — circuit, analysis, settings and all. Then solve it exactly as you would something you had just typed.

Trying to save an entry while a read-only file is loaded, or clicking **Create new**, starts a new input file of your own, blank by default.

Saving your work

Two links sit under the circuit box, and each appears only when it applies:

- **Save inputs to new entry** adds what is on screen as a new entry and asks you to name it. This is the one you want the first time.
- **Update inputs in this entry** writes your changes back into the entry you loaded. It appears once you have actually changed something.

Beside the dropdown, **Rename** and **Delete** act on the entry itself. Neither is offered for the built-in examples, since they are read-only.

 WARNING

Nothing reaches your disk until you press Download

Saving an entry writes it into the file open **in your browser**. The only thing that produces a `.cir` on your computer is **Download**.

Symbulator does what it can in the meantime: your work is mirrored into the browser's own storage, so closing the tab does not lose it, and you are warned before anything replaces a file holding entries you have not downloaded. None of that is a file, though. Download when you are done.

What an entry remembers

An entry keeps everything you typed, not just the circuit description: the analysis and whatever it needs — the frequency, the two node names for an equivalent, which kind of two-port — the Expert Mode equations, unknowns and conditions, everything in **Settings**, whatever is in **Evaluate** and its **Conditions**, the **Solve** card, the plot, and a note of your own shown when the entry is loaded.

An entry is not just a circuit: it is everything you give Symbulator to run the analysis. Load one a year later and you are back where you left off, with the same rounding, asking for the same analysis.

The **About input file (.cir) format** section inside the **Input File** card explains what input files carry, in detail.

LESSON 1

Direct current analysis

Learn to describe a circuit in Symbulator, and how to run a *direct current* analysis using **dc**. Learn how to describe a *voltage source* using **e**, and a *resistor* (or a conductance) using **r**.

I firmly believe one learns best by *doing*, so I have written this documentation as a tutorial, where you learn Symbulator by working through a series of progressively more diverse and complex examples. In a moment, I'll teach you how to swim. But for now, let me throw you in the deep end!

1.1 Run a direct current analysis

Open Symbulator 9. If you already have it installed as a local app, then just open that app. If you do *not* have it locally, open the **online app** in a browser connected to the Internet. Let's click on **Clear all inputs**, found at the top right of the app page, to make sure there is nothing anywhere in the input fields. Then, in the box labelled **Circuit Description**, type the following circuit description, one element per line:

```
e,1,0,36
r1,1,2,1'k
r2,2,3,3'k
r3,3,0,2'k
```

Scroll down to the section called **Analysis & Settings**. If it is collapsed, open it with the little blue triangle to its left; under it are two drop-down menus. Set **Type of analysis** to *Solve circuit* and **Analysis** to *DC — direct current*. Then click **Run Symbulator**.

If you typed everything correctly, the word *Solving...* appears on the button for a moment. When it changes to *Solved!*, the results are ready. Scroll down to the Outputs part of the page, where the **Results** section is now active, holding the results for every node and every element of the circuit you just solved.

Congratulations! You have just run your first simulation in Symbulator 9.

How does it work

Let's now go over what we just did, one piece at a time.

The selections you made in **Analysis & Settings** tell Symbulator *what kind of question* you are asking (solve a circuit) and *which analysis* to perform (direct current). Everything else there is optional: you will meet the other menus and cards in later lessons, and until then they can be left exactly as they are.

The most important input is the description of the circuit you want to analyse. It goes in the **Circuit Description** area, as a series of lines of text, each describing one element.

```
e,1,0,36  
r1,1,2,1'k  
r2,2,3,3'k  
r3,3,0,2'k
```

Using one line per element keeps a long circuit readable, and it makes mistakes easier to find. Symbolator also accepts a colon as a separator. This is the same circuit in one line:

```
e,1,0,36:r1,1,2,1'k:r2,2,3,3'k:r3,3,0,2'k
```

To reuse this circuit later, save it as an **entry** in an **input file**. You will be asked to name both. Input files must have the `.cir` extension.

Look at the elements

Now let's examine the contents of the circuit description. Study it, and you will see it has four lines. Each line describes one element.

At the moment they may look cryptic. But by the end of this lesson you will read them with ease.

1.2 How to describe a resistor

Notice that the second, third and fourth elements in the list all start with the letter **r**. These are resistors.

NOTE

Describing resistors

An ideal resistor is described with four pieces of information, separated by commas: a unique name (which must start with the letter **r**), the names of its first and second node, and its value in ohms (Ω).

For example, an ideal resistor called **r1**, between nodes **a** and **b**, of **300** Ω , is described `r1,a,b,300`

How to name resistors

Every resistor must have a name. As long as it starts with the letter **r** and is unique, it can be whatever you want. *Unique* means no other element or node has the same name.

How to name nodes

Naming the nodes is the first thing we do to solve a circuit. Every node must have a name, and node names must be unique: no other node or element may have the same one. Otherwise they can be whatever you want, with one constraint: at least one node

should be called **0** (zero). This will be considered the *ground node* of your circuit and will have, by definition, a voltage of 0 volts.

 WARNING

You need a ground node

Every circuit must have a node called 0 to serve as reference or ground.

Values can use SI prefixes

The values of circuit elements are often given with prefixes of the International System (SI) — kilo, milli, micro. So Symbulator has a shorthand for them: an SI prefix in an element's value, preceded by an apostrophe, multiplies it by the corresponding factor, exactly.

 TIP

Prefixes welcome!

You can use SI prefixes, preceded by an apostrophe, in the values of elements.

For example, an 8 k Ω resistor can be entered in many ways: 8000, 8'k, 8000. and 8E3 are all equivalent, except that the first two are treated as exact and the other two as approximate.

1.3 What answers do you get

After the simulation in DC is complete, Symbulator fills in the **Results** section for you, under **OUTPUTS**. Nothing is hidden away: every answer it worked out is on the page, and reading them is a matter of scrolling and looking.

The **Results** are arranged in groups. **Node voltages** first, one line per node, and then **Results by element**, which gives each element a small block of its own headed by its name and its kind — r1 resistor, e voltage source.

Every answer is written the way you would write it by hand, as a named quantity and its value: $v_1 = 36 \text{ V}$, $i_{r1} = 6 \text{ mA}$.

How answers are shown

Out of the box, Symbulator answers *exactly*. It works symbolically, so it would rather tell you a current is $3/500 \text{ A}$ than round it off. That is the right default for a machine doing algebra, and exactly what you want for symbolic results: an answer of $v_{in} \cdot r_2 / (r_1 + r_2)$ cannot and should not be rounded.

For numerical work like the example in this lesson, decimals read better. Open the **Settings** card and set **Rounding** to *approx to n digits* with **n** = 3, then tick **Use SI prefixes in answers**. The current above then reads 6 mA instead of $3/500 \text{ A}$.

One thing to expect: ticking the prefix box moves **Rounding** off *exact* by itself, because a prefixed value is a decimal. Symbulator says so on screen rather than doing it silently,

and the reverse happens too — choose *exact* again and the prefixes switch off.

Throughout this tutorial the numerical examples assume approximate answers to three or four figures with prefixes on, and the symbolic ones assume *exact*. Where a change is needed, the text says so.

Answer for each node

For each node, its voltage with reference to ground is shown under **Node voltages**, labelled v with the node's name below it. The voltage in node 1 appears as $v1$.

Answers for each resistor

For each resistor, the following answers are calculated:

- The voltage drop in the resistor, defined as the voltage in the first node minus the voltage in the second node, in volts. For a resistor called $r5$, this is the line marked **voltage drop**, labelled v_{r5} .
- The current through the resistor, flowing from the first node towards the second, in amperes. For a resistor called rx , this is the line marked **current through**, labelled i_{rx} .
- The power consumed by the resistor, in watts. For a resistor called $r12$, this is the line marked **power consumed**, labelled p_{r12} .

By now you should understand the description of the second, third and fourth elements in our example.

1.4 How to describe a voltage source

The first element in our example, on the other hand, started with the letter **e**. This element is a voltage source.

NOTE

Describing voltage sources

In Symbulator, a voltage source is described with four pieces of information, separated by commas: a unique name (which must start with the letter **e**), the names of its positive and negative node, and its value in volts (V).

For example, an ideal voltage source called **e1**, with its positive terminal in node **3**, its negative terminal in the ground node **0**, and a value of 12 volts, is described `e1,3,0,12`

How to name voltage sources

Every voltage source in your circuit must have a unique name that starts with the letter **e**, and it can be whatever you want.

Values can use SI prefixes

A voltage source's value can use SI prefixes too — they work in the values of every element.

Answers for a voltage source

For each voltage source, the following answers are calculated:

- The voltage drop in the source, defined as the voltage in the first node minus the voltage in the second, in volts. For a source called e5, the **voltage drop** line, v_{e5} .
- The current through the source, flowing from the first node towards the second, in amperes. For a source called ex, the **current through** line, i_{ex} .
- The power consumed — attention: not delivered, but consumed — by the source, in watts. For a source called e12, the **power consumed** line, p_{e12} . If we want the power delivered, we ask for the negative of this value.
- The equivalent resistance of the rest of the circuit, as seen by the source. For a source called e2, the **resistance seen** line, r_{e2} .

NOTE

This last one belongs to sources only

That equivalent resistance is a property of the *view from a source*, not a per-element quantity, so it exists only for sources. There is no r_{r1} for a resistor — a resistor's resistance is the value you gave it. Look at any resistor's block in **Results by element** and you will see three lines, not four: current, voltage and power, and no resistance seen. The same applies to the impedance answers in AC analysis, which you will meet in Lesson 7 (page 178).

1.5 A numerical DC simulation, step by step

Let's go back now to the simulation we ran earlier. That simulation corresponds to the circuit given in Example 5.7 in Boylestad's *Introductory Circuit Analysis* (11ed). Moving forward, I will refer to that textbook as **B11**.

The problem statement and the circuit schematic are reproduced exactly as they appear in the textbook, as they will be for every other problem in this tutorial. Since this is for educational purposes, it is my understanding that it falls squarely within the “fair use” doctrine of copyright law. In any event, no copyright infringement is meant.

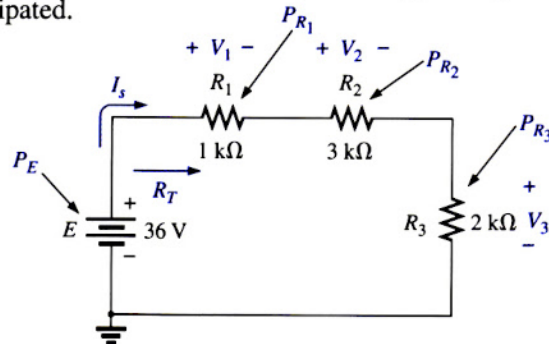
B11's Example 5.7

SOLUTION

All the values in this circuit are numbers: no element value is unknown. This is a **numerical circuit** — one where we know the numerical value of every element in it.

I will now walk you step by step through the solution. The process applies to most numerical simulations in Symbulator: first you describe the circuit, then you run the sim-

- Determine the total resistance R_T .
- Calculate the current I_s .
- Determine the voltage across each resistor.
- Find the power supplied by the battery.
- Determine the power dissipated by each resistor.
- Comment on whether the total power supplied equals the total power dissipated.



B11's Example 5.7

ulation, then you get the answers.

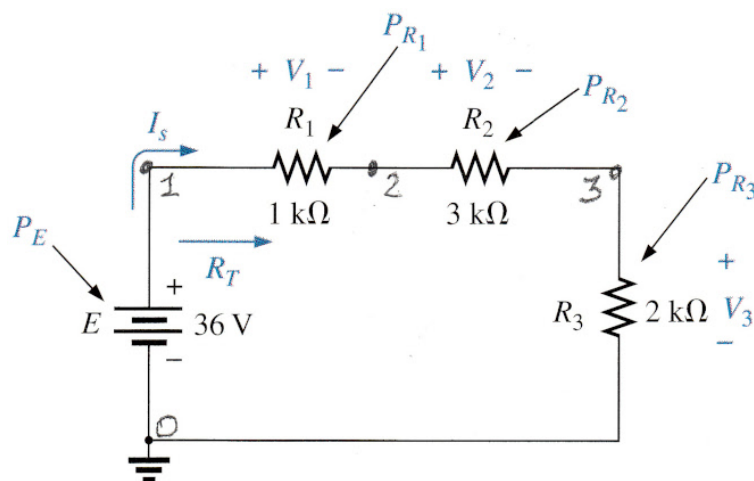
Step 1: describe the circuit. Description starts with **naming the nodes**. As we said, you can call them anything you want, number or letter, as long as the name is unique — but one node must always be called 0 (zero), the ground node, with a voltage of 0 V. In this circuit the ground node is marked with the ground symbol. That is not always so; when it is not marked, you pick a node to serve as zero.

I labelled the nodes in this circuit, starting in the ground and moving clockwise, as 0, 1, 2 and 3. It helps me to pencil the names in the schematic itself.

TIP

Write 'em down!

Write the names of your nodes and elements onto the circuit schematic itself, so you don't forget who's who.



B11's Example 5.7, with the node names pencilled in

After naming the nodes, I am ready to describe the elements of the circuit in Symbolator

notation.

Let's start with the source: when I only have one voltage source, like here, I enjoy giving it a short name: `e`. So the voltage source is `e,1,0,36`: its name is `e`, its positive node is 1, its negative node is 0, and its value is 36 volts between them in that order.

Now the resistors. I named the first `r1` and described it `r1,1,2,1'k`: its name is `r1`, its first node 1, its second node 2, and its value 1 k Ω . The second is `r2,2,3,3'k`, and the third `r3,3,0,2'k`.

 NOTE

An important point about SI prefixes

In the case of kilo, we can write it as either `'k` or `'κ`. But that's not the case for every other SI prefix: their case matters. For example, `'m` means milli, while `'M` means mega.

Those four descriptions, gathered together, are the whole circuit. Put them in the **Circuit description** box, one to a line:

```
e,1,0,36
r1,1,2,1'k
r2,2,3,3'k
r3,3,0,2'k
```

Step 2: run the simulation. The two menus under the box say what kind of simulation to run. We want a plain solve of the circuit as described, in direct current, so:

- **Type of analysis** → *Solve circuit*
- **Analysis** → *DC – direct current*

Then press **Run Symulator**.

The page fills in below almost immediately — this circuit takes well under a second — and a line at the foot of the **Results** section tells you what just happened: *DC analysis · 16 result(s)*, and how long it took. Sixteen answers, found and displayed in one go.

For a DC analysis, these answers are as follows:

- The **voltage of each node**, so the voltages of nodes 1, 2 and 3 are in the three lines under **Node voltages**: `v1`, `v2` and `v3`.
- The **current through each element**. The direction of the current is defined as going from the first node in the element's description to the second.
- The **voltage drop in each element**, defined as the difference in voltage between the first node in the element's description and the second, in that order.
- The **power consumed in each element**. An important point is that this is the power consumed, not the power delivered.
- Finally, **for each source, the equivalent resistance** of the rest of the circuit **as seen by that source**.

NOTE

The sign is in the details

To read the signs of your answers correctly, mind the details:

- The current in the elements is defined as going through the element from the first node to the second node.
- The voltage drop is defined as the voltage of the first node minus the voltage of the second node.
- The power given is the power consumed. So, if you want power delivered, evaluate the negative of that power.

Step 3: get the answers. We can now answer the six questions in the problem.

Answer to question (a). The equivalent resistance as seen by the source e :

Nothing to compute. Find the block headed e *voltage source* in **Results by element**, and read the line marked **resistance seen**:

$$r_e = 6 \text{ k}\Omega$$

That is 6 k Ω . Correct.

Answer to question (b). Current I_s is defined in the schematic as the current flowing through the source, in the direction that goes from node 0 to node 1. One way to find this value is evaluating the negative of the current through the source, which as you know flows in the opposite direction:

The **current through** line in the same block reads $i_e = -6 \text{ mA}$. The answer we want is its opposite, 6 mA.

That is 6 mA. Another way, since this is a series circuit where every element carries the same current, is to read the **current through** line of any resistor — all three say 6 mA.

Answer to question (c). The voltage drop in resistor R_1 — since its polarity is defined in the schematic in the same way it is defined in our circuit description — is as follows.

It is already on screen. In the block for r_1 , the **voltage drop** line reads $v_{r1} = 6 \text{ V}$. Look at the blocks for r_2 and r_3 and you will find $v_{r2} = 18 \text{ V}$ and $v_{r3} = 12 \text{ V}$. These are all the right answers.

Answer to question (d). Since the problem asks for the power supplied by the source, and we know that the **power consumed** line of the source's block, p_e , holds the power consumed by it, we need its opposite, which is 216 mW, that is 216 mW delivered.

Answer to question (e). The power consumed by the resistors is on the **power consumed** line of each resistor's block: $p_{r1} = 36 \text{ mW}$, $p_{r2} = 108 \text{ mW}$ and $p_{r3} = 72 \text{ mW}$, that is 36 mW, 108 mW and 72 mW consumed, respectively.

Answer to question (f). Let's ask the computer whether the sum of the consumed power in the resistors equals the power supplied by the source:

The tidiest way to check a balance like this is to add everything up and expect nothing. If the resistors consume exactly what the source delivers, then all four powers together must come to zero. In **Evaluate**:

```
pr1 + pr2 + pr3 + pe
```

The answer is **0** — not a small number, but zero. If you want to be sure it is genuinely zero and not three digits' worth of rounding, set **Rounding** back to *exact* for a moment: Symulator returns to fractions and still says 0.

This is the right answer, and concludes the solution to this, your first ever problem in Symulator.

1.6 Instructive numerical examples, solved

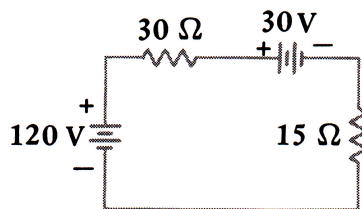
Numerical problems in DC using e and r

These practice problems are taken from several textbooks, chosen because they apply only the concepts you have learned so far. They let you practise and reinforce them before Lesson 2.

The problem below comes from Figure 1-26 (a) in Hyatt and Kemmerly's *Engineering Circuit Analysis* (5ed). Moving forward, we will refer to that textbook as **HK5**.

HK5's Figure 1-26

We are asked for the current, voltage drop and power consumed in each resistor, the power *delivered* by each voltage source, and a check that the powers in the circuit add up to zero.



Here is my solution.

I named the nodes thus: the bottom node **0**, and the top nodes, from left to right, **1**, **2** and **3**. My description of the circuit:

```
e1,1,0,120
r1,1,2,30
e2,2,3,30
r2,3,0,15
```

Run it, and the answers are all on the page already:

- The **current through** line of either resistor gives the current in the resistors: **2 A**
- The **voltage drop** line of r1 gives the voltage drop in the 30Ω resistor: **60 V**
- vr2 is the voltage drop in the 15Ω resistor: **30 V**
- pr1 is the power consumed in the 30Ω resistor: **120 W**

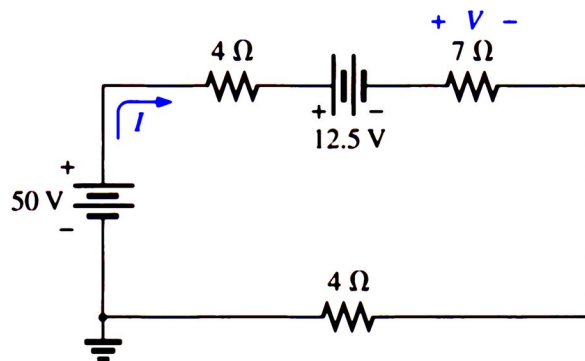
- p_{r2} is the power consumed in the 15Ω resistor: **60 W**
- Flipping the sign of p_{e1} gets the power delivered by the 120V source: **240 W**
- Flipping the sign of p_{e2} gets the power delivered by the 30V source: **-60 W**. This means this source is actually consuming 60W.
- $p_{r1}+p_{r2}+p_{e1}+p_{e2}$ is the sum of powers: **0 W**. As expected.

Wasn't that easy?

B11's Example 5.20

The practice problems get progressively more complicated, so you build up your 'symbolizing' skills with confidence.

Determine I and the voltage across the 7Ω resistor



I named the nodes clockwise from the ground: **0**, **1**, **2**, **3** and **4**. My circuit description, as an argument of the DC simulation command:

```
e1,1,0,50
r1,1,2,4
e2,2,3,12.5
r2,3,4,7
r3,4,0,4
```

When it's done, ask for the answers we need. i_{r1} is the current I : **2.5 A**. v_{r2} is the voltage drop in the 7Ω resistor: **17.5 V**

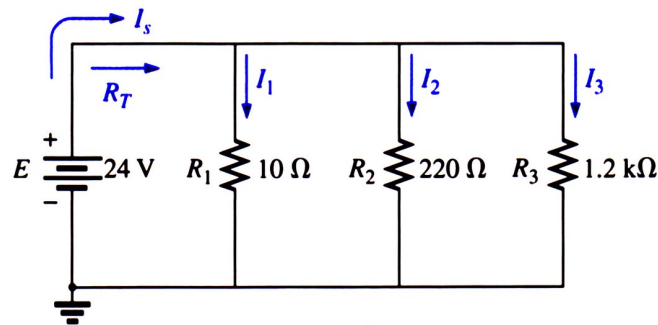
B11's Example 6.13

Since the ground is the bottom node, I named it **0**. I named the top node **1**.

```
e,1,0,24
r1,1,0,10
r2,1,0,220
r3,1,0,1.2'k
```

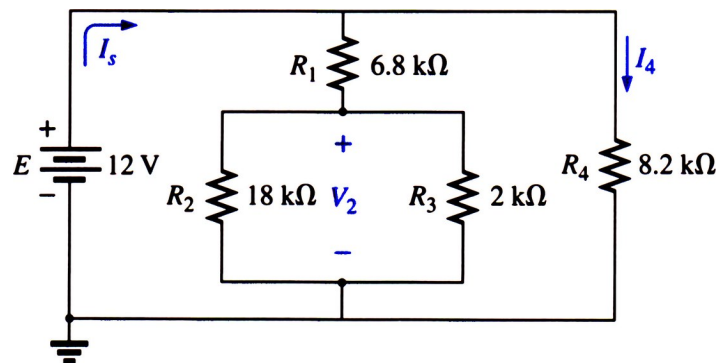
- r_e is the total resistance: **9.49 Ω**
- Flipping the sign of i_e gets us the source current: **2.53 A**
- i_{r1} is I_1 : **2.4 A**, i_{r2} is I_2 : **109 mA**, and i_{r3} is I_3 : **20 mA**.

- Find the total resistance.
- Calculate the source current.
- Determine the current through each branch.



B11's Example 7.2

Determine I_4 , I_s and V_2 . My solution: I named the top node **1**, and the other **2**:



```
e,1,0,12
r1,1,2,6.8'k
r2,2,0,18'k
r3,2,0,2'k
r4,1,0,8.2'k
```

Answers: v_2 is **2.51 V**, the opposite of i_e (i.e. I_s) is **2.86 mA** and i_{r4} is **1.46 mA**.

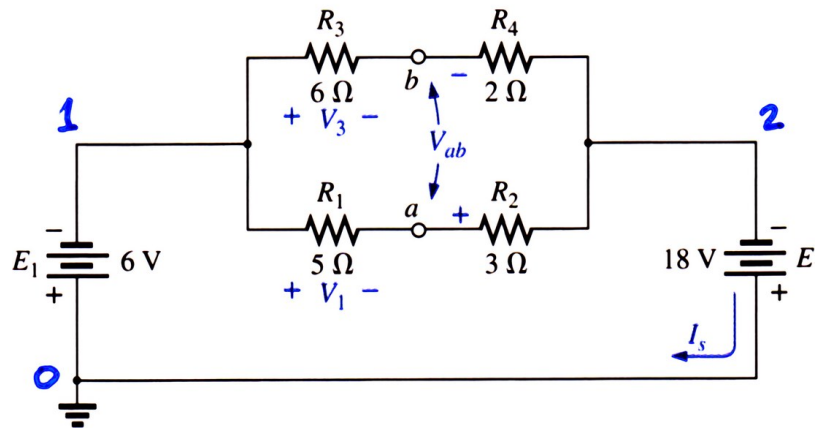
B11's Example 7.7

I have labelled the node names I used. My solution:

```
e1,0,1,6
e2,0,2,18
r1,1,a,5
r2,a,2,3
r3,1,b,6
r4,b,2,2
```

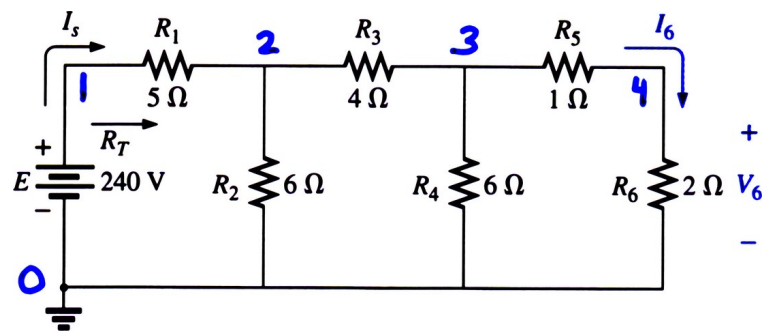
Answers: v_{r1} is **7.5 V**, v_{r3} is **9 V**. For V_{ba} , $v_b - v_a$ is **-1.5 V**. For I_s , the opposite of i_{e2} is **3 A**.

- Find the voltages V_1 , V_3 , and V_{ab} for the network
- Calculate the source current I_s .



B11's Figure 7.32

Determine I_6 and V_6 .



My solution:

```
e,1,0,240
r1,1,2,5
r2,2,0,6
r3,2,3,4
r4,3,0,6
r5,3,4,1
r6,4,0,2
```

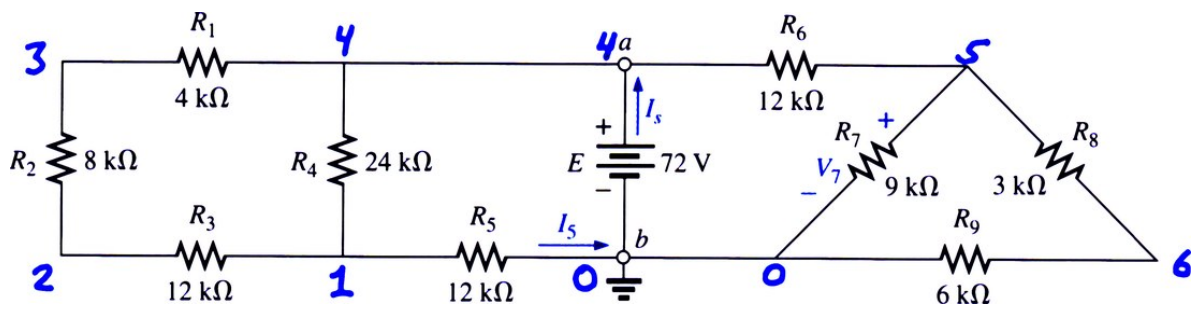
Answers: i_{r6} is **10** A, and v_{r6} is **20** V.

B11's Example 7.10

Calculate the indicated currents and voltages.

My solution:

```
r2,2,3,8'k
r1,3,4,4'k
r3,1,2,12'k
```

```

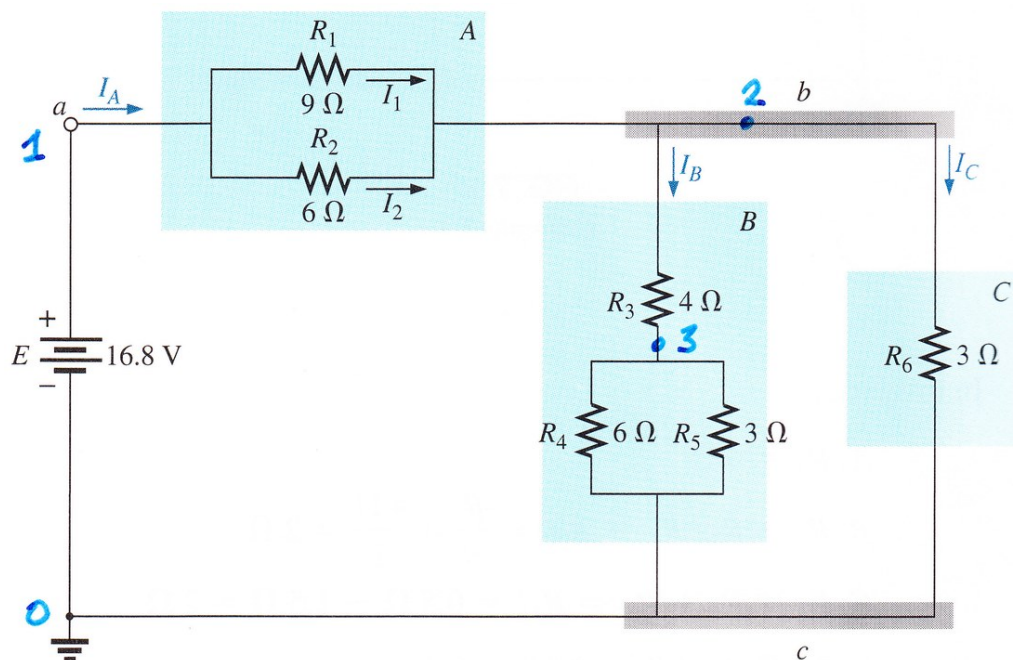
r4,1,4,24'k
r5,1,0,12'k
e,4,0,72
r6,4,5,12'k
r7,5,0,9'k
r8,5,6,3'k
r9,0,6,6'k

```

Answers: i_{r5} is **3** mA, the opposite of i_e (i.e. I_s) is **7.36** mA, and v_{r7} is **19.6** V.

B11's Example 7.4

Determine the currents I_1 , I_2 , I_A , I_B and I_C , and the voltage drops A, B and C.



My solution:

```

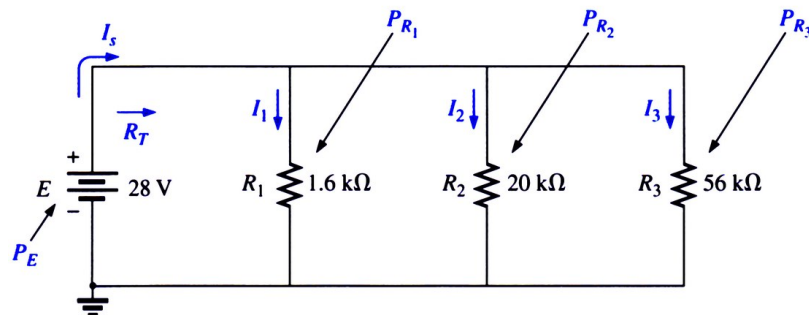
e,1,0,16.8
r1,1,2,9
r2,1,2,6
r3,2,3,4
r4,3,0,6
r5,3,0,3
r6,2,0,3

```

Current I_1 is found via $i_{r1} = 1.2$ A, I_2 via $i_{r2} = 1.8$ A, I_A , via the opposite of $i_e = 3$ A, I_B via $i_{r3} = 1$ A and I_C via $i_{r6} = 2$ A. The voltage drop in area A is $v_{r1} = 10.8$ V; in both B and C it is $v_2 = 6$ V.

B11's Example 6.15

- Determine the total resistance R_T .
- Find the source current and the current through each resistor.
- Calculate the power delivered by the source.



My solution:

```
e,1,0,28
r1,1,0,1.6'k
r2,1,0,20'k
r3,1,0,56'k
```

- r_e is the total resistance: **1.44** kΩ
- i_{r1} is **17.5** mA, i_{r2} is **1.4** mA, and i_{r3} is **0.5** mA
- Flipping the sign of p_e gets the power: **543** mW

These are the correct answers.

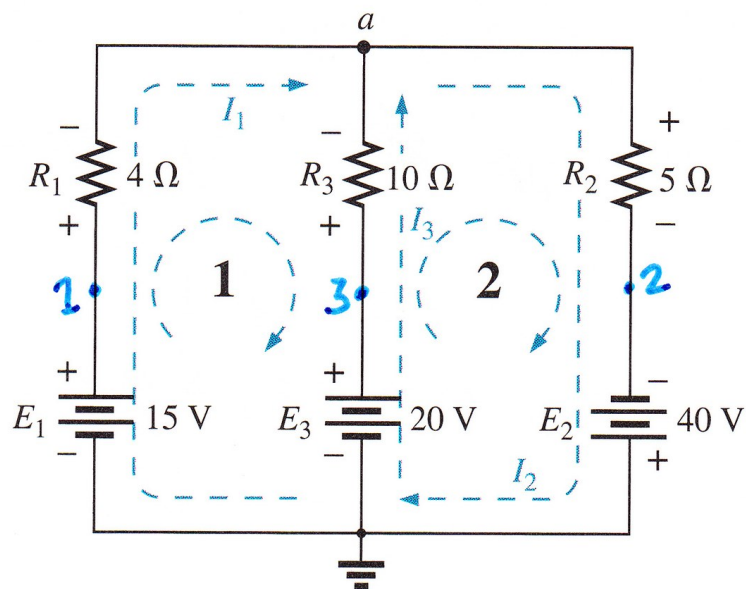
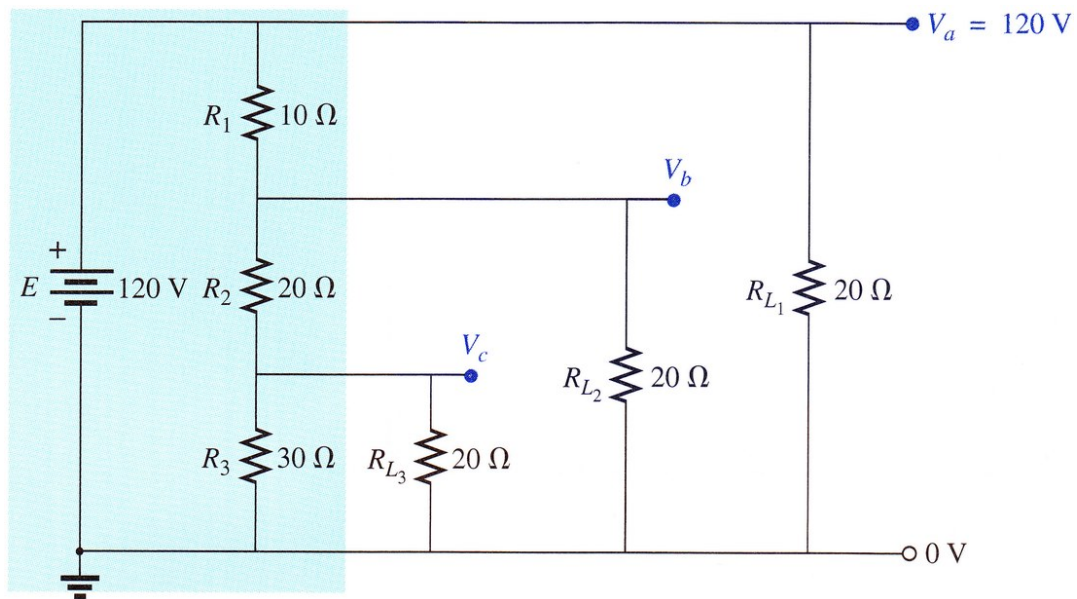
B11's Figure 7.40

Determine V_b and V_c .

My solution:

```
e,a,0,120
r1,a,b,10
r2,b,c,20
r3,c,0,30
r11,a,0,20
r12,b,0,20
r13,c,0,20
```

Answers: v_b is **66.21** V, and v_c is **24.83** V.



B11's Example 8.10

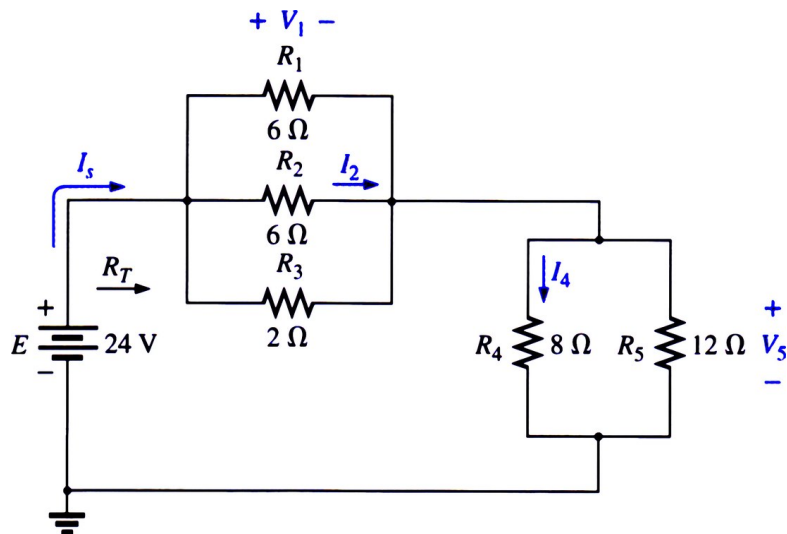
Determine the current through each resistor.

My solution:

```
e1,1,0,15
r1,1,a,4
e3,3,0,20
r3,3,a,10
e2,0,2,40
r2,a,2,5
```

Reading the **current through** line of each of the three resistors: $I_{R1} = 4.77\text{A}$, $I_{R2} = 7.18\text{A}$ and $I_{R3} = 2.41\text{A}$. These are the correct answers.

B11's Example 7.6



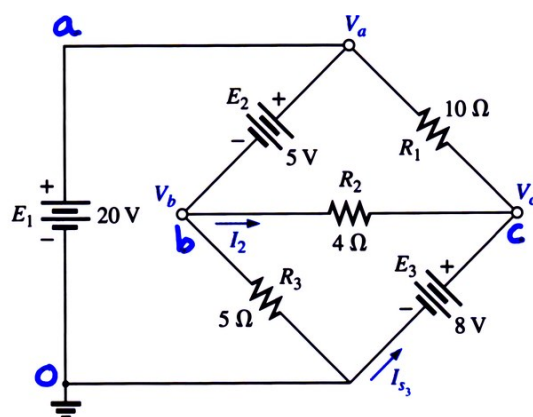
My solution:

```
e,1,0,24
r1,1,2,6
r2,1,2,6
r3,1,2,2
r4,2,0,8
r5,2,0,12
```

Answer: The opposite of i_e is $I_S=4$ A, i_{r2} is $I_2=.8$ A, i_{r4} is $I_4=2.4$ A, v_{r1} is $V_1=4.8$ V, v_{r5} is $V_5=19.2$ V.

B11's Example 7.11

- Determine voltages V_a , V_b , and V_c .
- Find voltages V_{ac} and V_{bc} .
- Find current I_2 .
- Find the source current I_{s3} .



My solution:

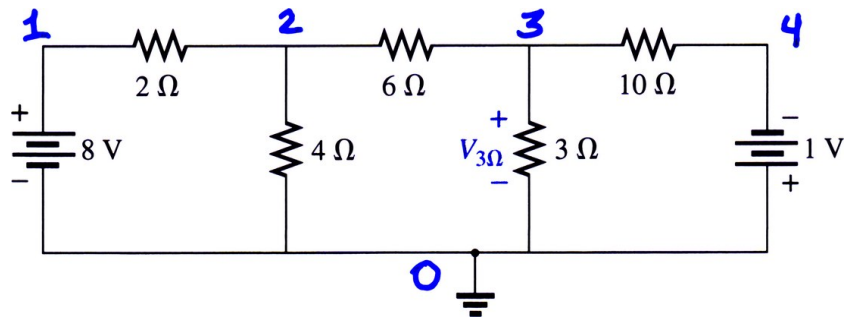
```
e1,a,0,20
e2,a,b,5
e3,c,0,8
r1,a,c,10
```

r2,b,c,4
r3,b,0,5

Answers: $v_a=20$ V, $v_b=15$ V, $v_c=8$ V, $v_a-v_c=12$ V, $v_b-v_c=7$ V, $i_{r2}=1.75$ A, I_S via the opposite of $i_{e3}=-2.95$ A

B11's Example 8.24

Find the voltage drop in the 3Ω resistor.



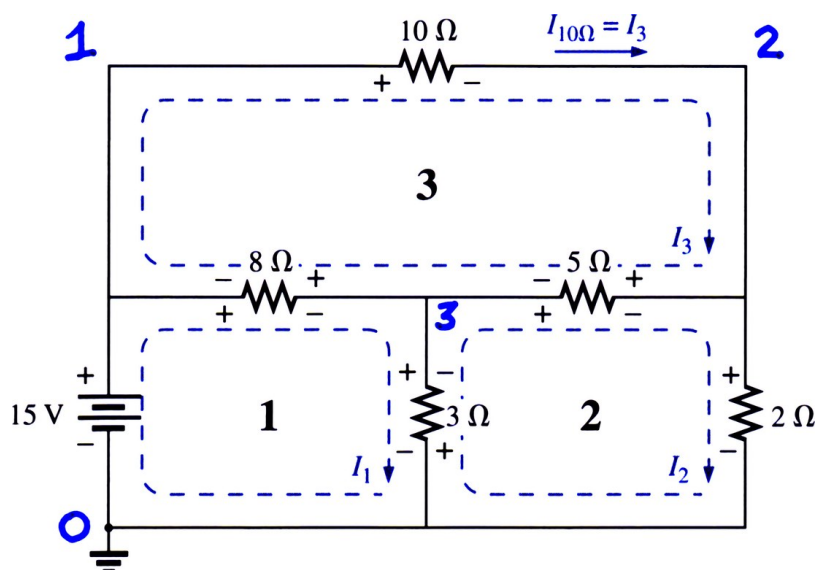
My solution:

e8,1,0,8
r2,1,2,2
r4,2,0,4
r6,2,3,6
r3,3,0,3
r10,3,4,10
e1,0,4,1

The **voltage drop** line of the r3 block shows that $V_{3\Omega}$ is **1.1** V. This is correct.

B11's Example 8.18

Find the current through the 10Ω resistor in the network shown.



My solution:

```
e15,1,0,15
r10,1,2,10
r8,1,3,8
r5,3,2,5
r3,3,0,3
r2,2,0,2
```

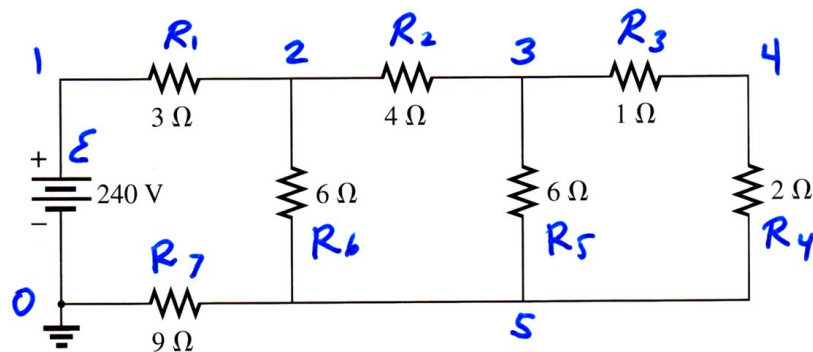
The answer you want is `ir10`, in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

We find that `ir10` = **1.22** A.

B11's Example 8.26

Find the voltage drop in the 2Ω resistor.



My solution is below:

```
e,1,0,240
r1,1,2,3
r2,2,3,4
r3,3,4,1
r4,4,5,2
r5,3,5,6
r6,2,5,6
r7,5,0,9
```

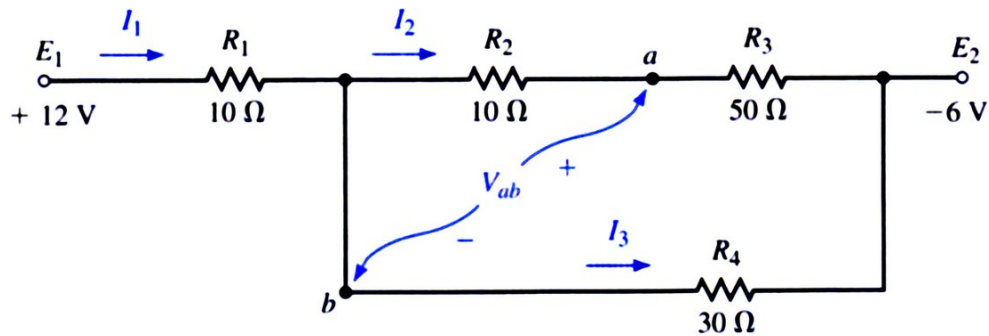
The `r4` block gives the voltage drop in R_4 (the 2Ω resistor): it is **10.67** V

Circuits with 'hidden source'

Sometimes the schematics of circuits are presented in such a way that sources of voltage are not shown explicitly, yet their voltage is provided. These are what I call 'hidden source' problems. Below I offer two examples of these types of problems. Both are taken from the textbook *Circuit Analysis: Theory and Practice* (3ed) by Allan H. Robbins and Wilhelm C. Miller, to which from this point on we will refer as **RM3**.

RM3's Example 7-5 (Hidden source)

Find the indicated currents and voltages.



In my solution below, notice I introduced two sources, one for 12V and one for -6V:

```
e1,1,0,12.  
r1,1,b,10  
r2,b,a,10  
r3,a,2,50  
r4,b,2,30  
e2,2,0,-6
```

The answers you want are `ir1`, `ir2` and `ir4`, in **Results**.

Ask **Evaluate** for:

```
va-vb
```

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

The answer, **{.6,.2,.4,-2.}**, indicates $I_1 = .6$ A, $I_2 = .2$ A, $I_3 = .4$ A and $V_{ab} = -2$ V. This is correct.

RM3's Figure 7-16 (Hidden source)

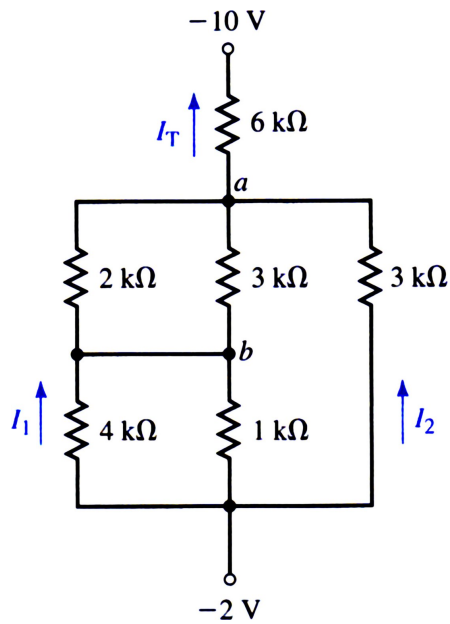
Find the total circuit resistance, and the indicated currents and voltages.

My solution:

This is how I named the nodes: the top, **1**; the bottom, **2**, and **a** and **b** as in the figure. In my solution below, notice I introduced two sources, one for -10V and one for -2V:

```
r1,1,b,4'k  
r2,1,a,3'k  
r3,b,a,2'k  
r4,b,a,3'k  
r5,1,b,1'k  
rt,a,2,6'k  
e1,1,0,-2  
e2,2,0,-10
```

The answers you want are `irt`, `ir1` and `ir2`, in **Results**.



Ask **Evaluate** for:

```
(v1-v2)/irt
va-vb
```

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

The answer we get indicates that the equivalent resistance, given by $(v_1 - v_2)/I_T$, is **7.2 kΩ**, and that $I_T = \mathbf{1.11 \text{ mA}}$, $I_1 = \mathbf{.133 \text{ mA}}$, $I_2 = \mathbf{.444 \text{ mA}}$ and $V_{ab} = \mathbf{-0.8 \text{ V}}$. This is correct.

Symbolic circuits and expert mode

Learn how to solve different types of *symbolic* circuits in Symbulator. Use the solve command on Symbulator answers. And explore **ex**, the powerful *expert mode*, for the ambitious user.

Here you will learn about the types of symbolic circuits and how to solve them in Symbulator, using the solve command when needed, and — for the more adventurous — the powerful *expert mode*.

2.1 About symbolic problems

The problem we saw in Lesson 1 (page 9) was a numerical problem because it involved a numerical circuit: from the start, we knew the values of all its elements. None were unknown.

A circuit with one or more elements whose value we do not know is a **symbolic circuit**. A symbolic problem is one that involves a symbolic circuit. It gives me great joy when I encounter a symbolic problem, because the ability to simulate symbolic circuits is what sets Symbulator apart from other programs.

NOTE

Variables are case-sensitive; names are not

Element and node names ignore case — $R1$ and $r1$ are the same resistor, and so is any reference built from a name: $2*VR1$, $2*v_{r1}$ and $2*v_{r1}$ all mean $r1$'s voltage drop. A variable that names nothing in the circuit is different: e, a, b, c and e, a, b, c are two *different* symbolic sources, and telling Symbulator that $c = 5$ says nothing about c . Pick one spelling for each unknown and keep it.

I find it useful to distinguish between two types of symbolic problems:

- **Purely symbolic problems** use symbolic circuits whose answers are also symbolic, given as expressions of unknown variables. Solving them in Symbulator is as straightforward as solving numerical problems.
- **Numerical-from-symbolic problems** start with a symbolic circuit but ask for numerical answers. That is possible when the problem gives us more information about the circuit.

Numerical-from-symbolic problems can be solved in two ways. If they are simple — one or two unknown values, one or two numerical answers wanted — it is quicker to simulate the circuit symbolically and then solve for the numerical answers with the **Solve** card, which sits under the results and does exactly that job. If they are not simple, it is easier

to use **expert mode**, which version 9 offers in its **Expert Mode** box.

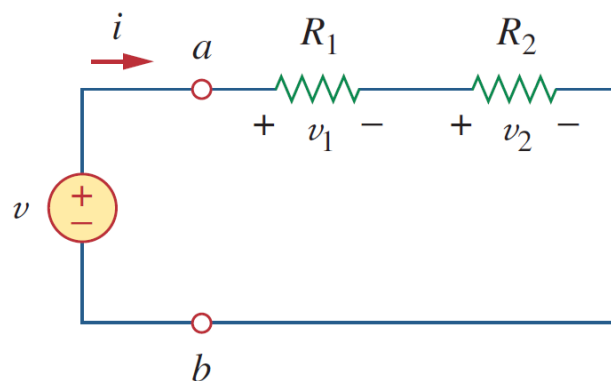
Let's see an example of each.

Solving a purely symbolic problem

Since Symbolator is natively a symbolic simulator, there is nothing special about solving a purely symbolic problem. The only limitation comes from the computing capacity of the computer: while small circuits are solved almost instantly, large symbolic circuits can produce expressions too large to be useful.

Let's see a simple purely symbolic problem, from *Fundamentals of Electric Circuits* by Alexander and Sadiku (5th edition), from now on referred to as **AS5**.

AS5's Figure 2.29



AS5's Figure 2.29

SOLUTION

All the values in this circuit are variables: no element value is known. This is an example of a symbolic circuit. Imagine you are asked to find symbolic expressions for the current i , the voltage drop in R_2 , and the power consumed by R_1 and R_2 . Since we are being asked to find symbolic answers from a symbolic circuit, this is a purely symbolic problem.

This is how I would describe this circuit. First, the nodes. The node called a in the schematic stays node a . The node called b is my ground, so it becomes node 0. The node between the two resistors I call c .

Then the elements. I name the source ev and the resistors $r1$ and $r2$, and give them variables for values: v for the source, $r1$ and $r2$ for the resistors. It is no problem that a symbolic value carries the same name as its element.

TIP

Turn rounding back to exact for this lesson

Lesson 1 (page 9) had you set **Rounding** to *approximate to n significant digits*, which is right for numbers and pointless here: there is nothing to round. Open **Settings** and put **Rounding** back to *exact*. The **Use SI prefixes** tick will clear itself when you do, which is what we want — a prefix has nothing to attach itself to in an answer

like $v/(r1 + r2)$.

Type the three elements into the box, one to a line:

```
ev,a,0,v  
r1,a,c,r1  
r2,c,0,r2
```

Leave **Type of analysis** on *Solve circuit*, set **Analysis** to *DC – direct current*, and press **Run Symulator**.

The answer comes back almost at once – symbolic circuits this small cost nothing. To find the current i , we look at the block for $r1$ or for $r2$ and read the **current through** line. To find the voltage drop on $r2$, we read the **voltage drop** line in the $r2$ block.

Both are expressions rather than numbers:

$$i_{r1} = \frac{v}{r1 + r2} \quad \text{and} \quad v_{r2} = \frac{r2 v}{r1 + r2}$$

which is the voltage divider you would have written by hand.

To find the power consumed in $r2$, we read the **power consumed** line of the $r2$ block. To find the power delivered by the source, we take the opposite of the source's **power consumed** line – type $-pev$ into **Evaluate**.

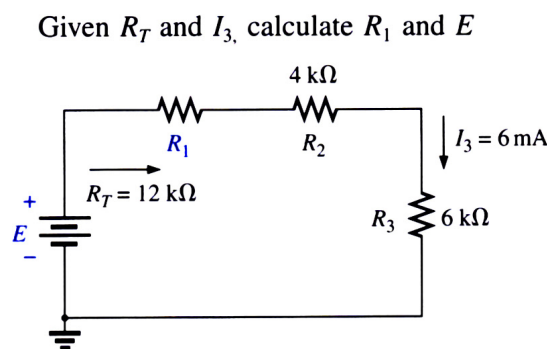
$$p_{r2} = \frac{r2 v^2}{(r1 + r2)^2} \quad \text{and} \quad -pev = \frac{v^2}{r1 + r2}$$

It is this ability to simulate symbolically, as if it were nothing, that puts Symulator in a league of its own. Symbolic answers from a symbolic circuit take no extra effort.

2.2 Numerical from symbolic, with solve

Getting numerical answers from a symbolic circuit, on the other hand, takes an extra step: the solve command, as in this next example, or expert mode, as in the one after.

B11's Example 5.6, with solve



B11's Example 5.6

SOLUTION

This is a very nice numerical-from-symbolic problem. We can solve it into numbers because, although it hides two values from us (the source E and the resistor $R1$), it gives us in exchange two answers (the equivalent resistance R_T and the current I_3) to solve for them with.

Since this circuit is structurally identical to B11's Example 5.7 solved in Lesson 1 (page 9), we will use the same names for the nodes. The description is identical except for the values. Symbulator accepts numbers, variables or even algebraic expressions as values. Here I use e for source e , and $r1$ for resistor $r1$.

```
e,1,0,e  
r1,1,2,r1  
r2,2,3,4'k  
r3,3,0,6'k
```

Run it in DC, as before.

A moment later Symbulator is done. From its symbolic answers and the answers the problem gives us, we write two equations and solve them for the two unknowns.

The problem says that I_3 is 6 mA. In Symbulator, I_3 is the **current through** line of the $r3$ block, i_{r3} , the current through resistor $r3$. Look at it and you will see an expression rather than a number — $e/(r1 + 10000)$ — in terms of the two unknowns, e and $r1$. This is what we call a symbolic answer. The problem also says that R_T is 12 k Ω ; the equivalent resistance as seen by the source e is the **resistance seen** line, r_e , which is $r1 + 10000$. We can then write two new equations and solve them for e and $r1$.

That is what the **Solve** card is for. You will find it under the results, beside **Evaluate**. Open it and fill in two boxes.

In the first, headed **Equation(s) to solve in terms of the results**, write one equation per line, using the same names the results use:

```
r_e = 12'k  
i_r3 = 6'm
```

In the second, headed **Unknown(s) to solve for**, name what you want found, separated by commas:

```
e, r1
```

Then press **Solve equations**.

The calculator's restriction does not carry over: the shorthand works in these boxes too, so $r_e = 12'k$ reads as 12000.

An instant later we get the answers: the card lists $e = 72.0$ and $r1 = 2000.0$, that is a source of 72 V and a resistor of 2000 Ω . These are the right answers. Not many other circuit simulators allow this flexibility.

2.3 Numerical from symbolic, using ex

Symbulator's true strength is seen in numerical-from-symbolic problems like the one we solved above, when we use its expert mode of simulation. Expert mode cracks these problems open in a single call, and gives fully numerical answers whenever the problem provides as many answers as it hides values.

The three things the calculator stopped to ask you for — extra equations, extra unknowns and extra conditions — are three boxes in Symbulator 9, and they are hidden until you ask for them. Open the **Expert Mode** box and tick **Enable Expert Mode**; three new fields appear:

- **Add equations** — one per line, written in the names the results use.
- **Add unknowns** — comma-separated.
- **Add conditions** — for narrowing a solution down, which we will use later.

They apply to whatever analysis you run, so expert mode works the same way in DC, AC, FD and TR.

B11's Example 5.6, using ex

SOLUTION

We use the very same circuit description as before, `r1` as the symbolic value included — version 9 has no objection to it; the calculator needed a different name here, and the note below says why.

The statement of the problem gives us the information we need to write the two additional equations: the source sees $12\text{ k}\Omega$, and the current through R_3 is 6 mA . Both are written in the same names the answers come back under, and the calculator's SI shorthand works here too:

```
e,1,0,e
r1,1,2,r1
r2,2,3,4'k
r3,3,0,6'k
```

```
re = 12'k
ir3 = 6'm
```

```
e, r1
```

Run it in DC as usual. This time nothing comes back symbolic: Symbulator has enough to pin every value down, so the whole circuit arrives in numbers. The node voltages read $v_1 = 72\text{ V}$, $v_2 = 60\text{ V}$ and $v_3 = 36\text{ V}$, and at the foot of **Results by element** you will find the two unknowns you asked for, listed like any other answer: $e = 72$ and $r1 = 2000$.

Both are right, and they arrived alongside everything else rather than in a separate step — the whole circuit is solved, not just the two unknowns.

The speed advantage of the expert mode is not necessarily evident in this simple prob-

lem. It does give you an idea of what the expert mode is all about: you get to hand Symulator extra information before it solves, rather than after. Had this circuit been larger, the benefit in computation time would be clear.

NOTE

What you can write an equation about

Anything the simulation reports: node voltages $v_1, v_2, \dots$, element currents i_{r3} , branch voltages v_{r1} , powers p_{r2} , and the resistance or impedance a source sees, r_e and z_e . The one exception is the AC power family — s_-, p_- and ap_- in an AC analysis — which is defined through complex conjugation and cannot be solved as part of the system; Symulator will tell you so rather than quietly ignoring the equation, and you can restate the constraint in voltages and currents instead.

One thing to watch: a quantity that depends on an unknown quadratically, such as a power, can be satisfied by two different component values. Symulator returns one of them. If the answer surprises you, check whether the other root is the one the problem meant.

NOTE

Coming from a calculator version?

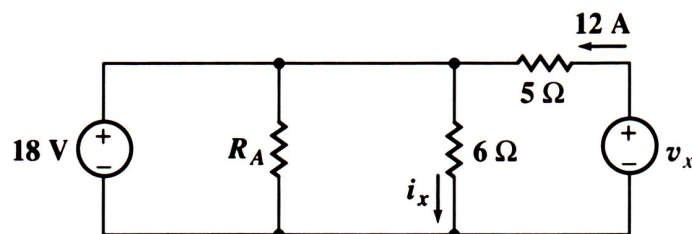
Two of the calculator's precautions are gone. There is no need to avoid r_1 as a symbolic value — Symulator 9 keeps its answers to itself, so there is nothing to collide with — and there are no follow-up dialogs to click through. The rest maps across directly: the three boxes carry the calculator's own prompts as their names — **Add equations**, **Add unknowns** and **Add conditions**.

2.4 Instructive symbolic examples, solved

Numeric-from-symbolic, with solve

HK5's Figure 1-24a (solve)

Determine i_x and v_x in the following circuit:



I named the nodes thus: bottom is **0**, top left is **1**, top right is **2**. I named the elements according to their value: this facilitates remembering who's who in the circuit. I also defined the resistors' nodes in the direction of the current indicated in the diagram.

```
e18,1,0,18
ra,1,0,ra
r6,1,0,6
r5,2,1,5
evx,2,0,vx
```

Explore the answers. Since **ir5** (which we know is 12 A) is in terms of v_x , we can find v_x :
Solve it in DC, then use the **Solve equations** card:

```
ir5 = 12
```

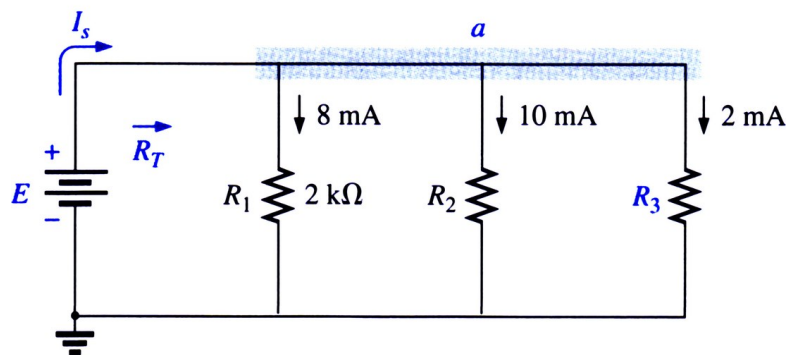
```
vx
```

We get that v_x is **78 V**, which is correct. The **r6** block already shows that i_x is **3 A**. The fact that we can find numerical answers in this problem can be quite puzzling until one realizes that ignoring the value of R_A doesn't matter: due to the circuit's structure, it is not needed to answer the two questions we have been asked.

Numeric-from-symbolic, with Expert

B11's Example 6.19 (Expert)

- Determine the source current I_s .
- Find the source voltage E .



This problem, having three unknown element values and three known answers, is a perfect candidate for the expert mode. Below is my circuit description.

```
e,1,0,e
r1,1,0,2'k
r2,1,0,r2
r3,1,0,r3
```

Select DC. Add these three equations:

```
ir1 = 8'm
ir2 = 10'm
ir3 = 2'm
```

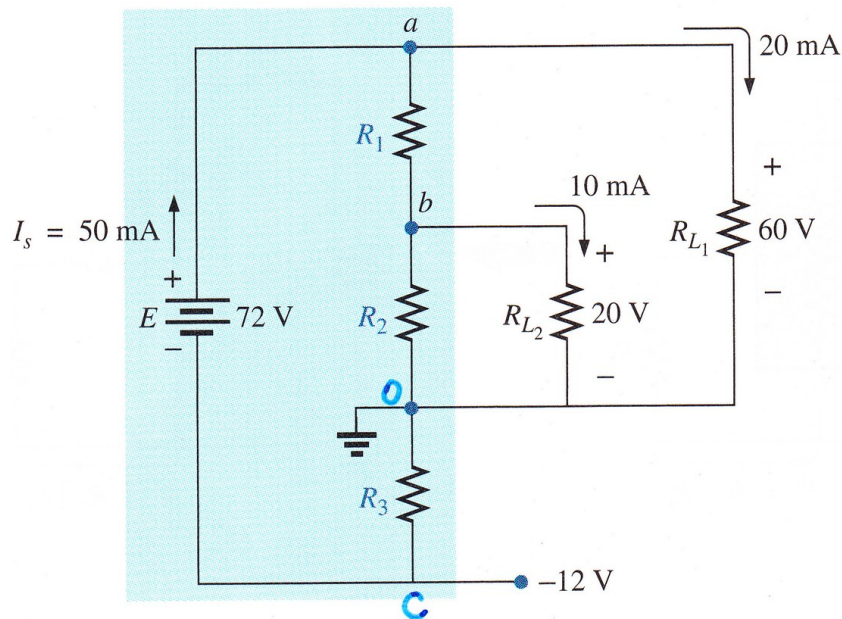
Add these unknowns:

e, r_2, r_3

Run it. Everything comes back numerical: $e = 16$ appears among the results, so E is **16** V, and the source's **current through** line gives $i_e = -0.02$ A, so I_s is **0.02** A. These are correct. The solver also fills in the two resistors it had to find on the way, $r_2 = 1600 \Omega$ and $r_3 = 8000 \Omega$.

B11's Example 7.12 (Expert)

Determine R_1 , R_2 and R_3 for the voltage divider supply. Can 2W resistors be used?



This problem is also perfect for the expert mode, because: (a) the target is to obtain numerical values, and (b) we have N unknown element values, and in turn we are given N numerical answers. Here is how I solved it:

$e, a, c, 72$
 r_1, a, b, r_1
 $r_2, b, 0, r_2$
 $r_3, 0, c, r_3$
 $r_{l1}, a, 0, r_{l1}$
 $r_{l2}, b, 0, r_{l2}$

Choose DC. Add these five equations:

$i_{r1} = 20 \text{ 'm}$
 $v_{r1} = 60$
 $i_{r2} = 10 \text{ 'm}$
 $v_{r2} = 20$
 $-i_e = 50 \text{ 'm}$

List these five as the unknowns to solve for:

$r_1, r_2, r_3, r_{l1}, r_{l2}$

Run it, and read the answers straight off the results: $r_1 = 1333.3 \Omega$, $r_2 = 1000 \Omega$, $r_3 = 240 \Omega$. These are correct.

The **power consumed** line of each resistor's block gives:

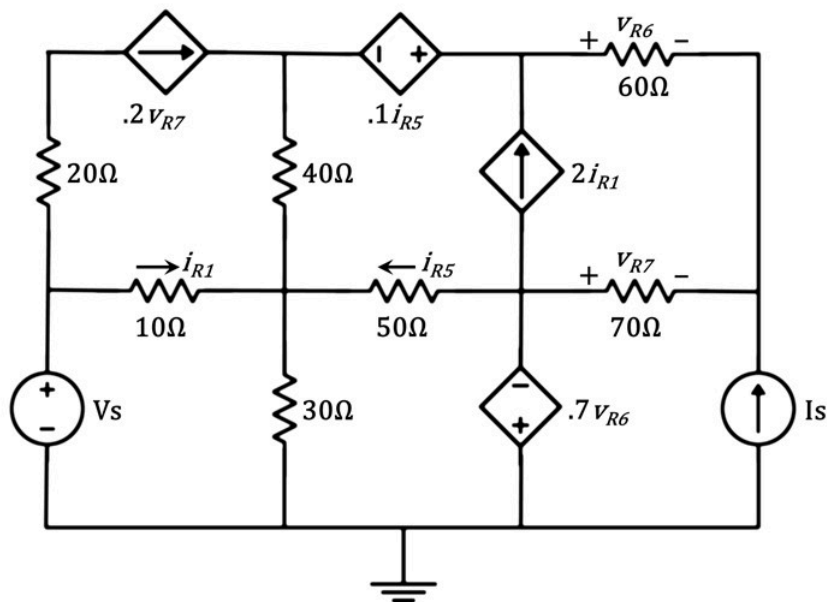
p_{r1} is **1.2 W**, p_{r2} is **0.4 W**, and p_{r3} is **0.6 W**

Since all are smaller than 2W, it is possible to use 2W resistors in the design.

The Showing-off Problem (Expert)

In 2014, I invented a problem to showcase how easy it is to describe dependent sources in Symulator.

Question: Find positive values for V_s and I_s that will result in 80W delivered by the VCCS and 0W dissipated in the CCVS.



Use this circuit description and equations:

```
es,e,0,vs
js,0,d,is
r1,e,m,10
r2,a,e,20
r3,m,0,30
r4,b,m,40
r5,n,m,50
r6,c,d,60
r7,n,d,70
jd1,a,b,0.2*vr7
ed2,c,b,0.1*ir5
jd3,n,c,2*ir1
ed4,0,n,0.7*vr6
```

Symbulator reports the power *consumed* by each element, so delivering 80W is consuming -80W. In **Expert Mode**:

```
pjd1 = -80 and ped2 = 0
```

```
vs, is
```

The problem asks for *positive* values, and that is what the conditions are for — the two power equations are quadratic, so four solutions fit the circuit, and these pick the one that was meant:

```
is > 0 and vs > 0
```

Run it in DC, with **Rounding** at 4 significant digits. Everything comes back numerical: $v_s = 17.61$ V and $i_s = 0.3973$ A.

And the answers show *how* the zero-dissipation constraint is met: i_{R5} comes back exactly 0 A. The CCVS's voltage is $0.1 i_{R5}$, so the constraint drove its own controlling current to zero — the source dissipates nothing because the circuit arranged for it to *be* nothing.

LESSON 3

Current sources, conductances and dependent sources

Learn how to describe a *current source* using the **j** element. Simplify *parallel resistors* using the **pr** tool and its shorthand. Learn to describe *conductances* and *dependent sources*.

In this lesson you will learn how to describe a *current source* with the **j** element, and a trick to simplify *parallel resistors* with the **pr** tool or its shorthand. You will also learn to describe *conductance* and *dependent sources* using elements you already know.

3.1 How to describe a current source

NOTE

Describing current sources

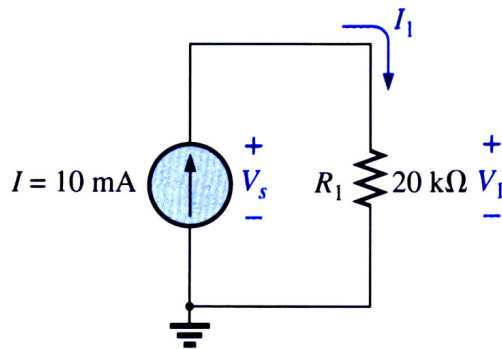
In Symbulator, an ideal current source is described with four pieces of information, separated by commas: a unique name (which must start with the letter **j**), the names of its first and second node, and its value in amperes (A). The value is the current flowing through the source from the first node towards the second: how much current leaves by the second node, and equally how much enters by the first. For example, an ideal current source called **j1**, between nodes **0** and **3**, carrying **5** A from the first node to the second, is described: `j1,0,3,5`

You can use SI prefixes here as well. The value of a current source is often given in milliamps; Symbulator will interpret any 'm' given in the value as a division by a thousand.

Answers for a current source

For each current source you get the same answers as for a voltage source, with the same polarity conventions: the voltage drop in it, the current through it, the power consumed by it (for the delivered power, ask for the negative), and the equivalent resistance of the rest of the circuit as seen by that source.

Let's see an example.



B11's Example 8.1

B11's Example 8.1

Given the circuit, determine the current and voltage drop in R_1 .

SOLUTION

We ask Symulator to run a DC simulation of the circuit described between quotations:

```
j,0,1,10'm
r1,1,0,20'k
```

Two lines, two elements. Under **Type of analysis** leave *Solve circuit*, set **Analysis** to *DC – direct current*, and click **Run Symulator**. This is a numerical circuit, so it is worth putting **Rounding** back to *approximate to n significant digits* with $n = 3$ and ticking **Use SI prefixes**, as in Lesson 1 (page 9).

The $r1$ block gives both answers at once: $i_{r1} = 10$ mA on the **current through** line, and $v_{r1} = 200$ V on the **voltage drop** line – that is a 10 mA current and a 200 V voltage drop.

3.2 What about conductances?

Conductances are really resistors by another name. So we describe them as resistors, using the element **r**, and give as the value the inverse of the conductance: one divided by the conductance is the resistor in Ω .

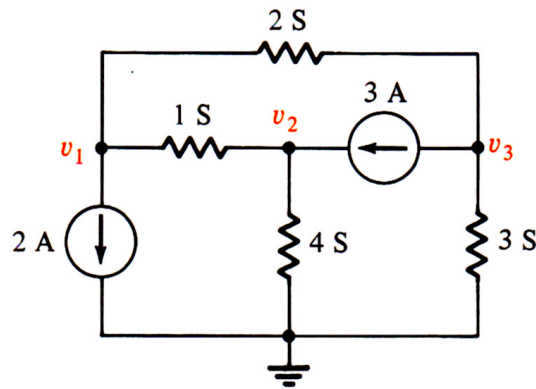
The next example is from *Elementary Linear Circuit Analysis* (2ed) by Leonard S. Bobrow, which from here on I call **Bo2**.

Bo2's Example 2.2

Given the circuit, determine the voltages in the nodes.

SOLUTION

As I explained, all conductances are simulated as resistors: the 4 siemens conductance becomes a $1/4$ resistor, and so on. My description:



Bo2's Example 2.2

```
j10,1,0,2
r12,1,2,1
r20,2,0,1/4
r30,3,0,1/3
r13,1,3,1/2
j32,3,2,3
```

Run it in DC. The answers are the first thing on the page, under **Node voltages**:

That indicates $v_1 = -1.3$ V, $v_2 = .34$ V, $v_3 = -1.12$ V. This is correct.

3.3 How to reduce parallel resistors

Symbulator 9 has the same tool, spelled `pr`, and you can use it in two places.

Inside a circuit description, write it where a value goes — that is what the `re,3,0,[6,3]` shorthand below does, and `pr(6,3)` means the same thing.

On its own, type it into the **Evaluate** card. To reduce three resistors in parallel with values of 10 Ω , 20 Ω and 30 Ω :

```
pr(10, 20, 30)
```

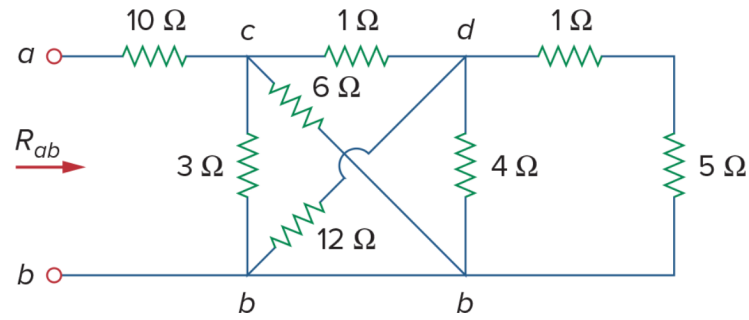
You get **60/11** Ω with **Rounding** on *exact*, or **5.45** Ω approximately. It nests as deeply as you like — the two problems below lean on that — and it works on symbols as readily as on numbers: `pr(r1, r2)` evaluates to $r_1 r_2 / (r_1 + r_2)$, useful when the resistors in your circuit are still unknowns.

AS7's Example 2.10

Find the equivalent resistance.

SOLUTION

No simulation is needed — the whole reduction is one nested expression in the **Evaluate** card, working from the far end of the ladder outward:



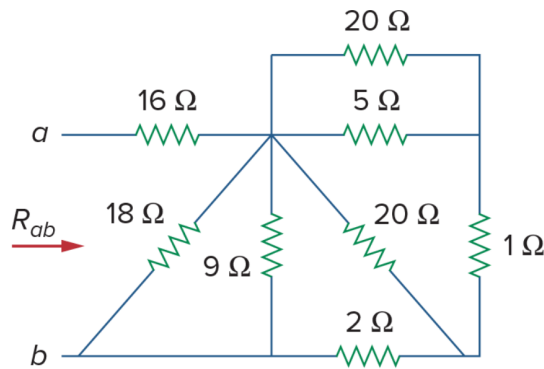
AS7's Example 2.10

```
10 + pr(3, 6, 1 + pr(12, 4, 1 + 5))
```

The answer is **11.2** Ω. This is correct.

AS7's Practice Problem 2.10

Find the equivalent resistance.



AS7's Practice Problem 2.10

SOLUTION

Again, no simulation:

```
16 + pr(18, 9, 2 + pr(20, 1 + pr(5, 20)))
```

The answer is **19** Ω. This is correct.

The [r,r,r...] shorthand for descriptions

Calling `pr()` separately while describing a circuit is not very practical, so there is a shorthand: Symulator reads any values inside square brackets, such as `[10, 20, 30]` or `[r1, r2, r3, r4]`, as input to `pr`. The **Evaluate** card runs what you type through the same shorthand, so the brackets work there too; this book writes `pr(...)` outside descriptions anyway, where square brackets are easy to misread as a list.

When to reduce resistors

A simulation where it makes sense to use `pr` is B11's Example 7.4, which you saw in the practice problems of Lesson 1 (page 9). It makes sense to reduce R4 and R5 to an equivalent resistor, since we do not need to know their individual currents or power use:

```
e,1,0,16.8
r1,1,2,9
r2,1,2,6
r3,2,3,4
re1,3,0,[6,3]
r6,2,0,3
```

Run it in DC, then ask **Evaluate** for `-ie`, the current the source delivers. The answer is **3 A**.

An example where using `pr` makes no sense is B11's Example 8.3, because you need to know the value of the current through R1.

Finally, a simulation where you can reduce part of the resistors is B11's Example 6.22. We must leave R1 alone, because we need the current through it, but we can reduce R2 and R3:

```
jt,0,1,12'm
r1,1,0,1'k
re,1,0,[10'k,22'k]
```

Moving forward, we will use the `pr( , , )` tool, or its shorthand `[ , , ]`, in circuit descriptions whenever we feel it is appropriate.

NOTE

Reduce only what you don't need to see

The reason to reduce resistors at all is speed, and the cost is information: once two resistors become one, you can no longer ask for the current through, or the power consumed by, either of them individually. Reduce only the ones the problem does not ask about.

NOTE

One difference worth knowing

Where doing the reciprocal arithmetic by hand would divide by zero, Symbolator's `pr` checks for it: if any of the values you give it is exactly zero, the combination is zero, because a short circuit across a parallel network wins. The calculator tool did the same.

3.4 How to describe dependent sources

One of my favourite scenes in cinema comes from *The Dark Knight*: the Joker, played masterfully by Heath Ledger, is rolling on the floor of a Gotham City prison, taking a bare-knuckle beating from an ever-more-frustrated Batman. Master of the situation

and laughing hysterically, the Joker says: *“You have nothing! Nothing to threaten me with!”*

Even though the movie had not been made yet, I remember feeling something similar — if less hysterical — back in 1999, when I realised that a 100% symbolic implementation let me make any element’s value depend on any answer of the circuit. I could simulate voltage or current sources dependent on any voltage, current or combination of them as easily as a 12 V source.

Here is what you need to know for simulating dependent sources in Symbulator: nothing. There is nothing special to it at all. Just write the value as a function of the circuit’s answers, using the variables you know by now, and run the simulation like it’s nobody’s business. For example:

- if the source depends on the current through a resistor called `r1`, you define its value as `ir1`
- if the source depends on the voltage drop in a resistor called `r2`, you define its value as `vr2`
- if the source depends on the current through a short called `s3`, you define its value as `is3`
- if the source depends on the difference between the voltage of two nodes `a` and `b`, you define its value as `va-vb`

With Symbulator, instead of fearing them, you will laugh in the face of dependent sources, thinking: *“You have nothing!”* Booyah!



TIP

Dependent sources are just sources

For Symbulator, dependent sources are just sources and need no special notation. State the expression that describes the value, using variables Symbulator knows and minding their polarity, and you are off to the races.

3.5 Instructive solved examples



NOTE

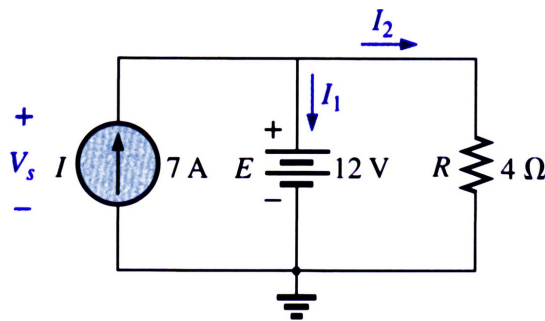
Some of this narration still describes the calculator

Every problem below carries a **Circuit Description** panel you can type straight into Symbulator 9, and the circuits and answers are identical across the versions. What has not all been rewritten is the narration between them: where it says to evaluate a name or press a key, do the version 9 equivalent — the results are already on screen.

Circuits with E, J and R

B11's Example 8.2

Determine the values of V_S , I_1 and I_2 .



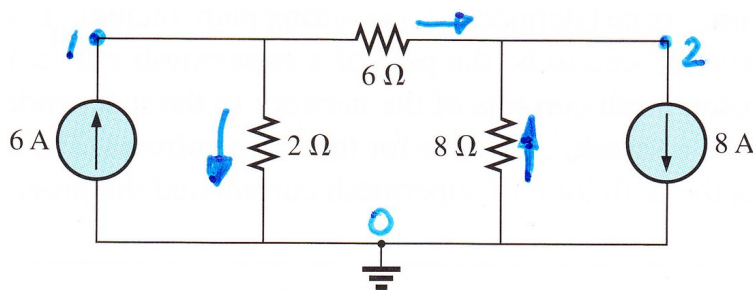
```
j,0,1,7  
e,1,0,12  
r1,1,0,4
```

The answers you want are v_1 , i_e and i_{r1} , in **Results**.

The answers show V_S is 12V, I_1 is 4A and I_2 is 3A. These are the correct answers.

B11's Example 8.15

Determine the current through each resistor. My solution, with the direction marked in blue:

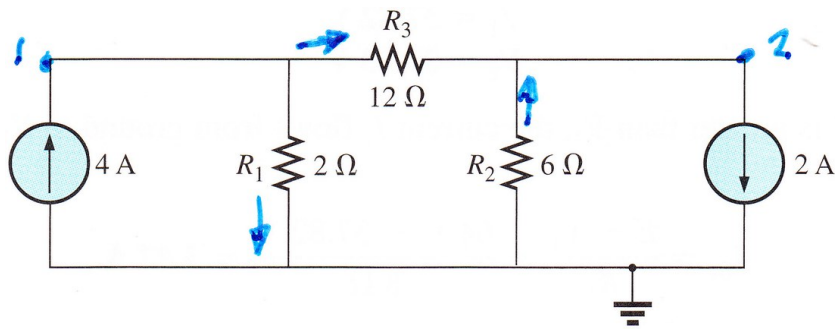


```
j6,0,1,6  
r2,1,0,2  
r6,1,2,6  
r8,0,2,8  
j8,2,0,8
```

The answers you want are i_{r2} , i_{r6} and i_{r8} , in **Results**.

The calculator versions wrap this in approx to get a decimal; version 9 does it through **Rounding** — approx to n digits, with $n = 3$ here.

We get these answers: **{1.25,4.75,3.25}**. So I_{R2} is 1.25A, I_{R6} is 4.75A, and I_{R8} is 3.25A.



B11's Example 8.21

Determine the voltage in each node and the current through each resistor.

My solution:

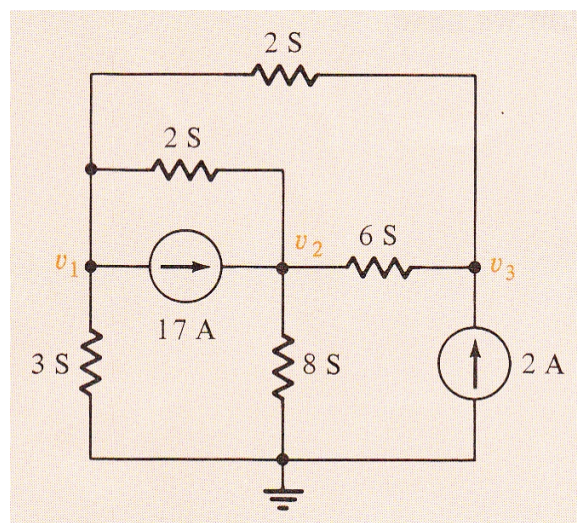
```
j1,0,1,4
r1,1,0,2
r3,1,2,12
r2,0,2,6
j2,2,0,2
```

The answers you want are v_1 , v_2 , i_{R1} , i_{R2} and i_{R3} , in **Results**.

We get the following answers: **{6,-6,3,1,1}**. So $V_1=6V$, $V_2=-6V$, $I_{R1}=3A$, and $I_{R2}=I_{R3}=1A$.

Bo2's Drill Exercise 2.2 (Conductances)

Given the circuit, determine the voltages in the nodes.



Below my solution:

```
r10,1,0,1/3
r12,1,2,1/2
r13,1,3,1/2
```

```
r23,2,3,1/6
r20,2,0,1/8
j12,1,2,17
j03,0,3,2
```

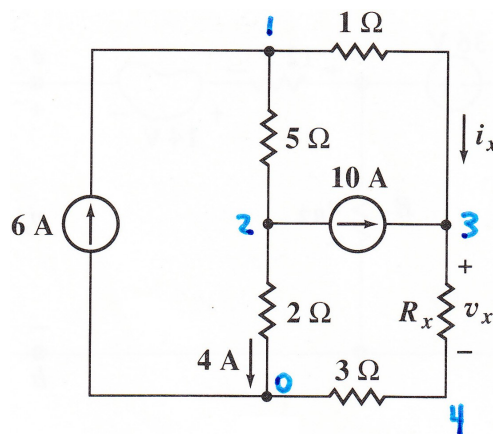
The answers you want are v_1 , v_2 and v_3 , in **Results**.

The calculator versions wrap this in approx to get a decimal; version 9 does it through **Rounding** – *approx to n digits*, with $n = 3$ here.

The answer, **{-2.,1.,.5}**, indicates $v_1 = -2\text{V}$, $v_2 = 1\text{V}$, $v_3 = 0.5\text{V}$. This is correct.

HK5's Figure 1-24b (Expert)

Determine i_x and v_x in the following circuit.



My solution:

```
j6,0,1,6
r5,1,2,5
r2,2,0,2
r1,1,3,1
r3,0,4,3
j10,2,3,10
rx,3,4,rx
```

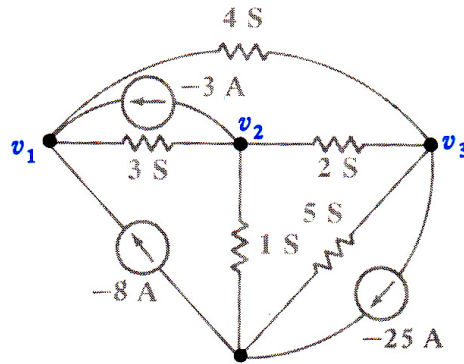
Set **Analysis** to *DC – direct current*. This one needs **Enable Expert Mode** ticked in the **Expert Mode** box; the equations and unknowns go in the fields it reveals.

The answers you want are i_{r1} and v_{rx} , in **Results**.

Select DC. Add equation $i_{r2}=4$. Add unknown r_x . Run the simulation. The answer, **{-8,80}**, means that I_x is -8A and that V_x is 80V .

HK5's Example 2.2 (Conductances)

Determine the voltages in the nodes. My solution:



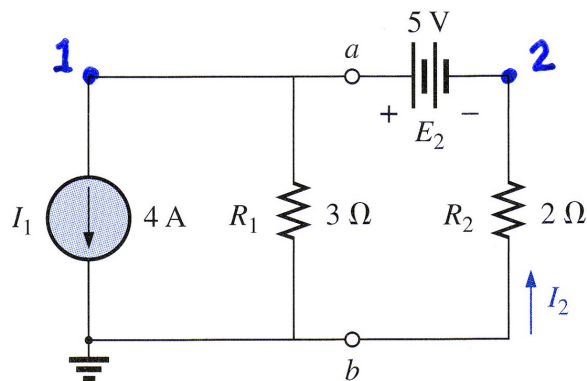
```
j01,0,1,-8
j30,3,0,-25
j21,2,1,-3
r12,1,2,1/3
r23,2,3,1/2
r13,1,3,1/4
r20,2,0,1
r30,3,0,1/5
```

The answers you want are v_1 , v_2 and v_3 , in **Results**.

The answer, **{1,2,3}**, is correct: $v_1=1\text{V}$, $v_2=2\text{V}$, $v_3=3\text{V}$.

B11's Example 8.5

Determine the current I_2 in the circuit shown. My solution:

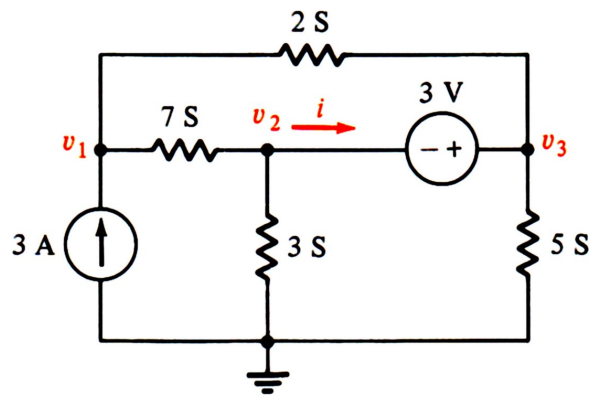


```
j1,1,0,4
r1,1,0,3
e2,1,2,5
r2,0,2,2
```

This gives us a value for I_2 of **3.4 A**. This is correct.

Bo2's Example 2.5 (Conductances)

Determine the voltages in the nodes, and the current through the voltage source.



```
j,0,1,3
e,3,2,3
r12,1,2,1/7
r20,2,0,1/3
r30,3,0,1/5
r13,1,3,1/2
```

The answers you want are v_1 , v_2 and v_3 , in **Results**.

Ask **Evaluate** for:

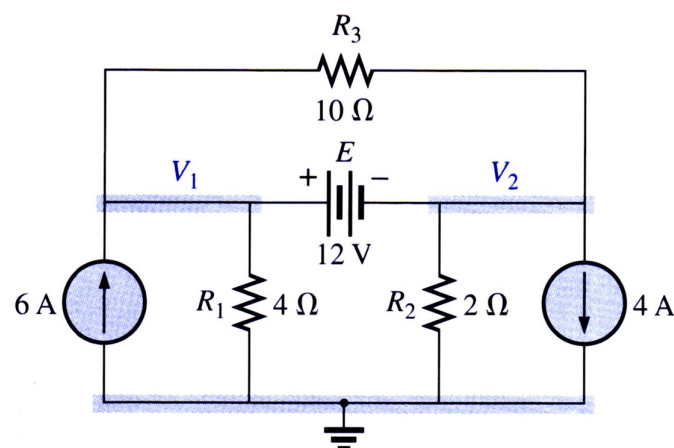
```
-ie
```

The calculator versions wrap this in approx to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with $n = 3$ here.

The answer, **{-.5,-1.5,1.5,11.5}**, is correct: $v_1 = -.5$, $v_2 = -1.5$, $v_3 = 1.5$, $i = 11.5$

B11's Example 8.22

Determine V_1 and V_2 . My solution:



```
j1,0,1,6
r1,1,0,4
e,1,2,12
r3,1,2,10
```

r2,2,0,2
j2,2,0,4

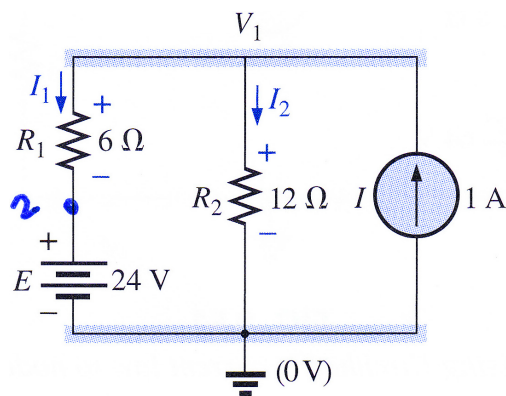
The answers you want are v_1 and v_2 , in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** – *approx to n digits*, with **n** = 3 here.

The answer, **{10.67,-1.33}**, tells us that V_1 is 10.67V and V_2 is -1.33V.

B11's Example 8.19

Determine V_1 , I_1 and I_2 , in the circuit:

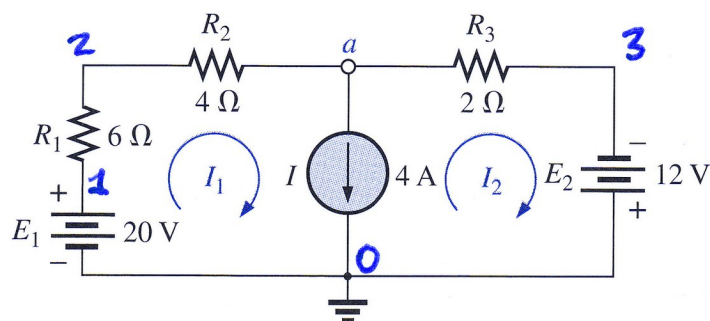


My solution:

e,2,0,24
r1,1,2,6
r2,1,0,12
j,0,1,1

The answer, **{20.,-.667,1.67}**, tells us that V_1 is 20V, I_1 is -.667A and I_2 is 1.67 A.

B11's Example 8.14



Determine I_2 and I_3 .

e1,1,0,20
r1,1,2,6
r2,2,a,4

```
j,a,0,4
r3,a,3,2
e2,0,3,12
```

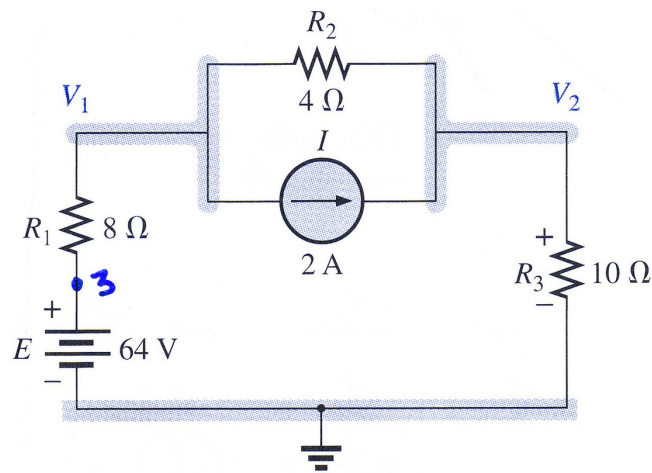
The answers you want are i_{r2} and i_{r3} , in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with $n = 3$ here.

Answer: **{3.33,-.667}**. This is correct.

B11's Example 8.20

Determine V_1 , V_2 , I_1 , I_2 and I_3 . My solution:



```
e,3,0,64
r1,3,1,8
r2,1,2,4
j,1,2,2
r3,2,0,10
```

The answers you want are v_1 , v_2 , i_{r1} , i_{r2} and i_{r3} , in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with $n = 3$ here.

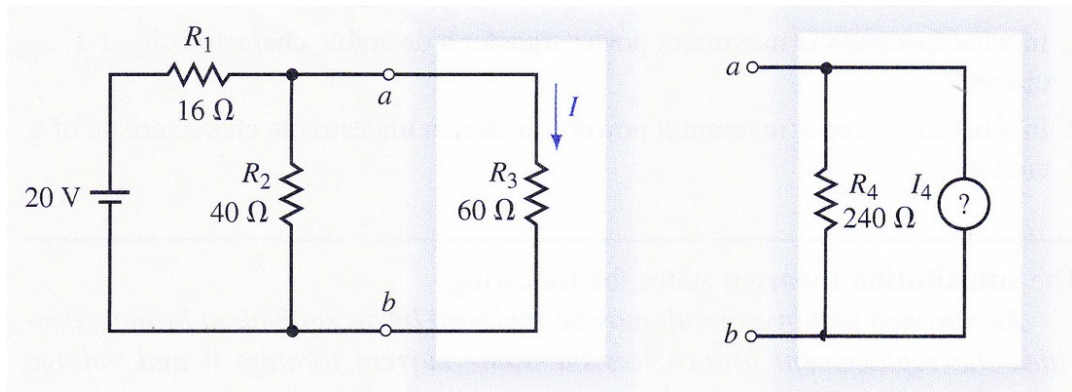
Answer: **{37.82,32.73,3.27,1.27,3.27}**. You should know how to read these by now, but here it is just in case: $V_1 = 37.82\text{V}$, $V_2 = 32.73\text{V}$, $I_1 = 3.27\text{A}$, $I_2 = 1.27\text{A}$, $I_3 = 3.27\text{A}$.

RM3's Example 9-12 (solve)

If R_3 is to be replaced with R_4 and I_4 , determine the value and direction of the source.

So that we can keep the name of node b, we will use the top left node as reference.

First, make sure you understand what this problem is asking you to do. The idea is that, despite the change, we keep the same voltage drop and current flow between nodes **a** and **b**. We must first know what they are. So we simulate the original circuit:



```
e,0,b,20
r1,0,a,16
r2,a,b,40
r3,a,b,60
```

The answer you want is `ir3`, in **Results**.

Ask **Evaluate** for:

```
va-vb
```

We find that the voltage drop is 12V and the current is 0.2A. These are the currents and voltages that we have to keep once we do the replacement. Simulate the circuit now replacing R_3 with a resistor R_4 of 240Ω and a source j with value I_4 . Run this:

```
e,0,b,20
r1,0,a,16
r2,a,b,40
r4,a,b,240
j,a,b,i4
```

Notice that both the voltage drop (given by `va-vb`) and the current (given by `ir4+ij`) are algebraic functions in terms of `i4`. Now you can find `i4` solving by voltage drop:

Both routes go in the **Solve** card, which solves against the answers the circuit just produced:

```
va-vb = 12
```

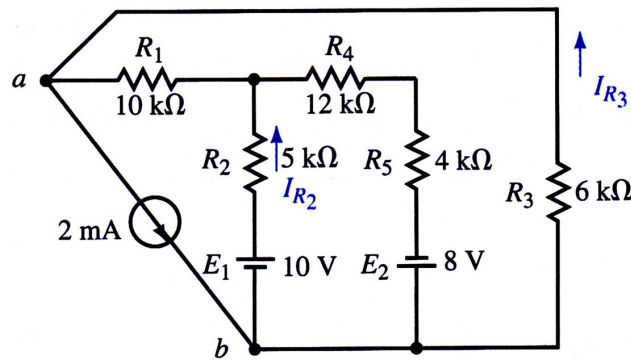
with `i4` as the unknown — or `ir4+ij = 0.2` for the same answer by current instead.

The result is the same: `i4 = .15 A`. The required current source is .15A from **a** to **b**.

RM3's Example 8-13

Solve for the currents through R_2 and R_3 in the circuit shown. My solution:

```
r1,a,0,10'k
r2,1,0,5'k
r3,b,a,6'k
```

```
r4,0,2,16'k
j,a,b,2'm
e1,1,b,10
e2,b,2,8
```

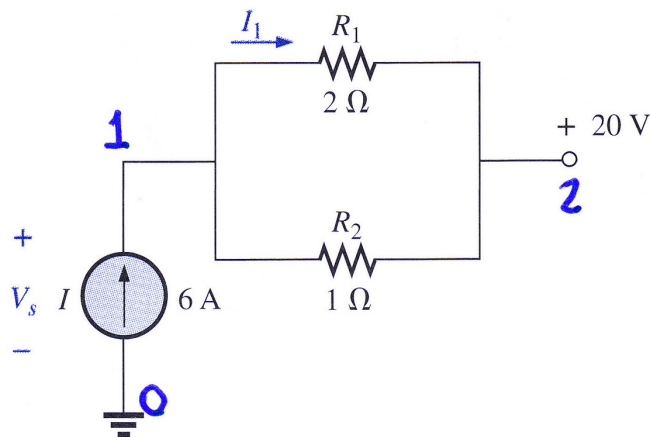
The answers you want are `ir2` and `ir3`, in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

The answer, **{.00154,.00111}** is correct: $I_{R2} = 1.54 \text{ mA}$ and $I_{R3} = 1.11 \text{ mA}$.

B11's Example 6.3 (Hidden source)

Determine V_S and I_1 .



The problem presents a seemingly 'hanging' node with 20V. To simulate this, imagine a 'hidden' 20V voltage source connected to the node.

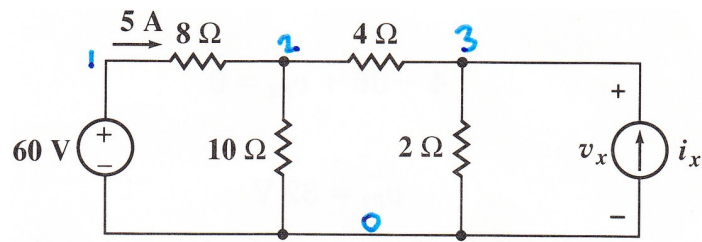
```
j,0,1,6
r1,1,2,2
r2,1,2,1
e,2,0,20
```

The answers you want are `v1` and `ir1`, in **Results**.

We get the answer: **{24,2}**, which is correct. V_S is **24V** and I_1 is **2A**.

HK5's Figure 1-24c (Expert)

Determine i_x and v_x in the following circuit.



```
e,1,0,60
r8,1,2,8
r10,2,0,10
r4,2,3,4
r2,3,0,2
j,0,3,ix
```

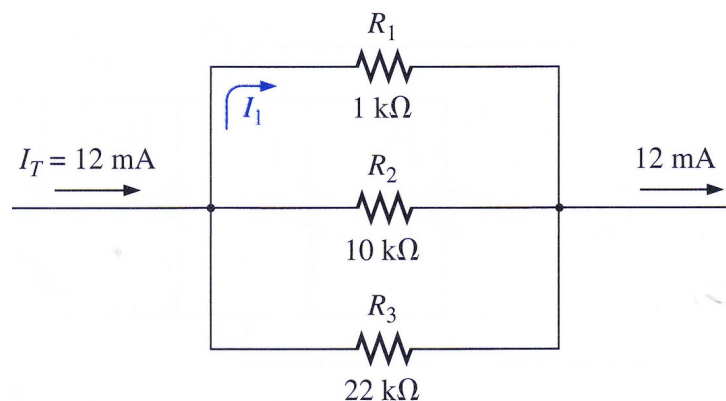
Set **Analysis** to *DC – direct current*. This one needs **Enable Expert Mode** ticked in the **Expert Mode** box; the equations and unknowns go in the fields it reveals.

The answers you want are i_x and v_x , in **Results**.

Select DC. Add equation $i_{r8}=5$. Add unknown i_x . Run the simulation, and you will get: **{1,8}**. This is correct: I_x is 1A and that V_x is 8V.

B11's Example 6.22 (Hidden source)

Determine I_1 .



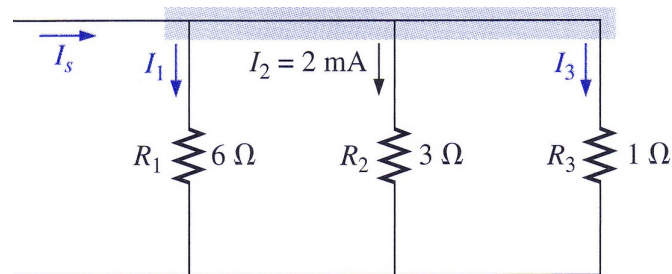
Although no source is shown, the circuit has a current. I use a 'hidden' current source.

```
jt,0,1,12'm
r1,1,0,1'k
r2,1,0,10'k
r3,1,0,22'k
```

We get $I_1 = 10.48$ mA, which is correct.

B11's Example 6.21 (Hidden source, Expert)

Determine I_S , I_1 and I_3 .



This 'hidden source' problem is perfect for Expert. My solution:

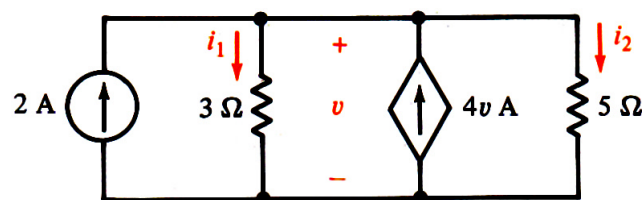
```
js,0,1,is1  
r1,1,0,6  
r2,1,0,3  
r3,1,0,1
```

Choose DC. Open **Expert Mode**, put $ir2 = 2'm$ in **Add equations** and $is1$ in **Add unknowns**, then **Run Simulator**.

The answer, **{.009,.001,.006}**, is correct, since the currents are as follows: I_S is 9mA, I_1 is 1mA and I_3 is 6mA.

Bo2's Example 1.9

Determine I_1 , I_2 and v .



This is my solution.

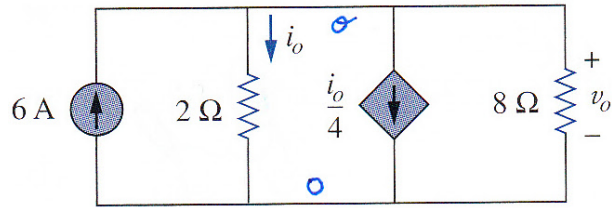
```
ji,0,1,2  
r1,1,0,3  
jd,0,1,4*v1  
r2,1,0,5
```

The answers you want are $ir1$, $v1$ and $ir2$, in **Results**.

The answer, **{-5/26,-15/26,-3/26}**, is correct: $I_1 = -5/26$ A, $I_2 = -3/26$ A and $v = -15/26$ V.

AS2's Practice Problem 2.7

Find v_o and i_o in the circuit. My solution:



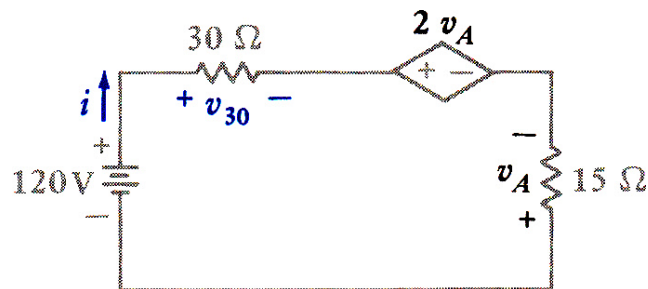
```
ji,0,0,6
ro,0,0,2
jd,0,0,iro/4
r8,0,0,8
```

The answers you want are v_o and i_o , in **Results**.

The answer, **{8,4}**, is correct: $v_o = 8$ V and $i_o = 4$ A.

HK5's Example 1-3

Determine the power delivered by each source and consumed by both resistors.



```
ei,1,0,120
r1,1,2,30
ed,2,3,2*vra
ra,0,3,15
```

Ask **Evaluate** for:

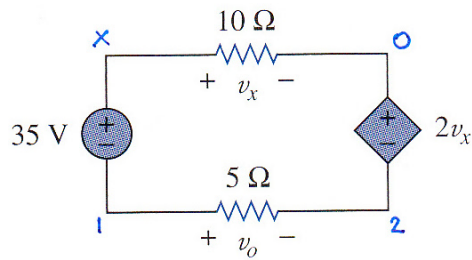
```
-pei
-ped
pr1+pra
```

The answer, **{960,1920,2880}**, is right: the independent source delivers 960W, the dependent source delivers 1920W, and the resistors consume 2880W together.

AS2's Practice Problem 2.6

Find v_x and v_o in the circuit. My solution:

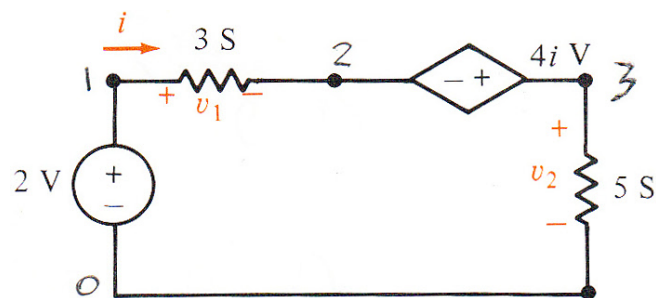
```
ei,x,1,35
rx,x,0,10
ed,0,2,2*v_x
ro,1,2,5
```



The answers you want are v_x and v_o , in **Results**.

The answer, **{10,-5}**, is correct: $v_x = 10$ and $v_o = -5$.

Bo2's Example 1.10



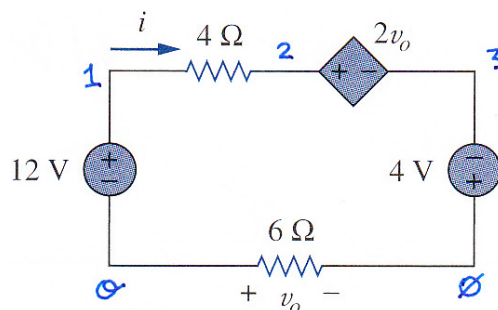
Determine v_1 , v_2 and i . My solution:

```
ei,1,0,2
r1,1,2,1/3
ed,3,2,4*ir1
r2,3,0,1/5
```

The answers you want are i_{r1} , v_{r1} and v_{r2} , in **Results**.

The answers, **{-15/26,-5/26,-3/26}**, is correct: $v_1 = -5/26$ V, $v_2 = -3/26$ V and $i = -15/26$ A.

AS2's Example 2.6



Determine v_o and i in the circuit. My solution is shown below:

```
e12,1,0,12
ri,1,2,4
ed,2,3,2*v_o
```

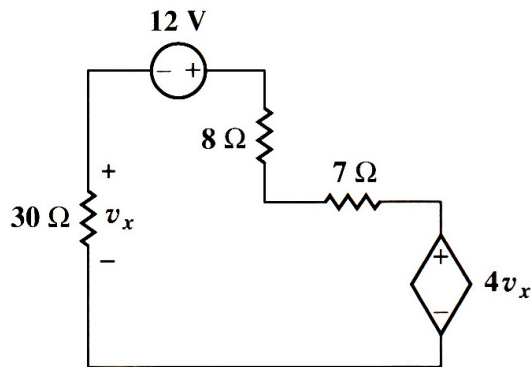
e4,0,3,4
r0,0,0,6

The answers you want are v_o and i_{ri} , in **Results**.

The answer, **{48,-8}**, is correct: $v_o = 48$ and $i = -8$.

HK5's Drill Problem 1.11

Find the power absorbed by each element in the circuit. My solution:



r1,x,0,30
ei,1,x,12
r2,1,2,8
r3,2,3,7
ed,3,0,4*vx

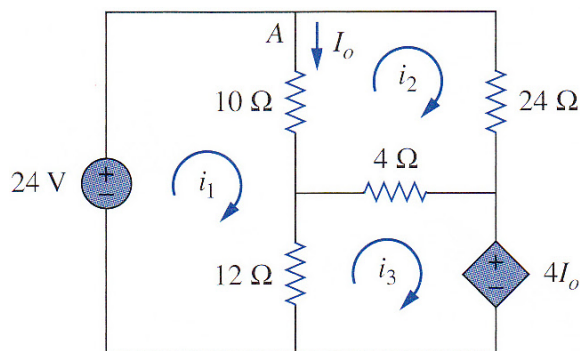
The answers you want are pr_1 , pe_i , pr_2 , pr_3 and pe_d , in **Results**.

The calculator versions wrap this in approx to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

The answer, **{.768,1.92,.2048,.1792,-3.072}**, is correct.

AS2's Example 3.6

Determine the value of I_o in the circuit. My solution:



```
ei,a,0,24
ro,a,b,10
r12,b,0,12
r4,b,c,4
r24,a,c,24
ed,c,0,4*iro
```

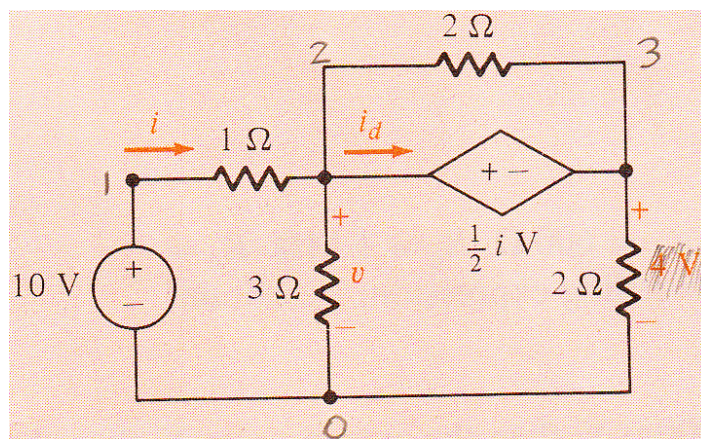
The answer you want is i_{ro} , in **Results**.

The calculator versions wrap this in approx to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with $n = 3$ here.

The answer, **1.5 A**, is correct.

Bo2's Drill Exercise 1.12

Determine i , v and i_d .



We are given an unnecessary piece of information: the 4V drop in the 2Ω resistor.

My solution:

```
ei,1,0,10
r1,1,2,1
r2,2,3,2
r3,2,0,3
r4,3,0,2
ed,2,3,ir1/2
```

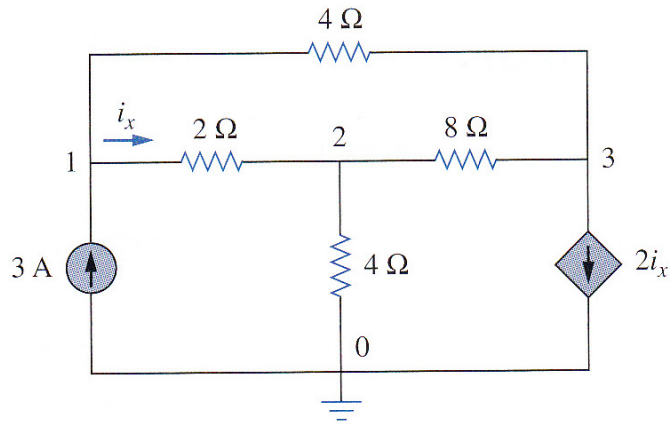
The answers you want are i_{r1} , v_{r3} and i_{ed} , in **Results**.

The answer, **{4,6,1}**, is correct: $i=4$, $v=6$ and $i_d=1$.

AS2's Example 3.2

Determine the voltages at the nodes.

My solution:



```

ji,0,1,3
jd,3,0,2*ir2
r2,1,2,2
r4a,1,3,4
r8,2,3,8
r4b,2,0,4

```

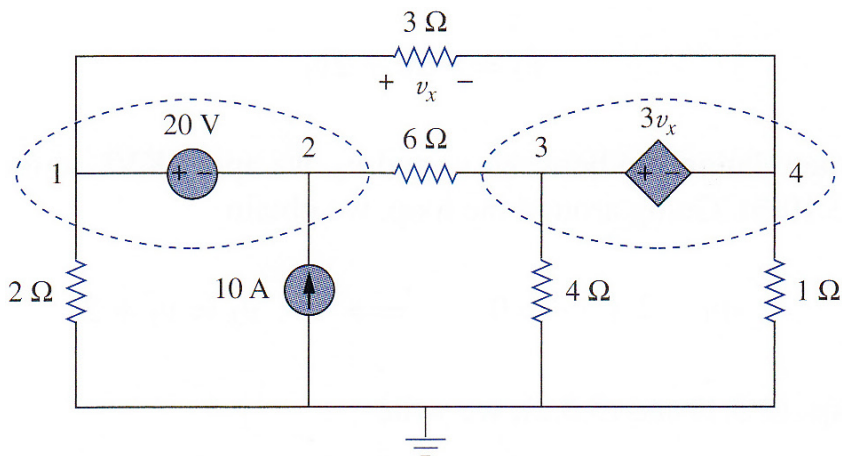
The answers you want are v_1 , v_2 and v_3 , in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with $n = 3$ here.

The answer, **{4.8,2.4,-2.4}**, is correct.

AS2's Example 3.4

Find the node voltages in the circuit. My solution:



```

r2,1,0,2
e1,1,2,20
j,0,2,10
r6,2,3,6
rx,1,4,3
r4,3,0,4
ed,3,4,3*vrx

```



```
r1,4,0,1
```

The answers you want are v_1 , v_2 , v_3 and v_4 , in **Results**.

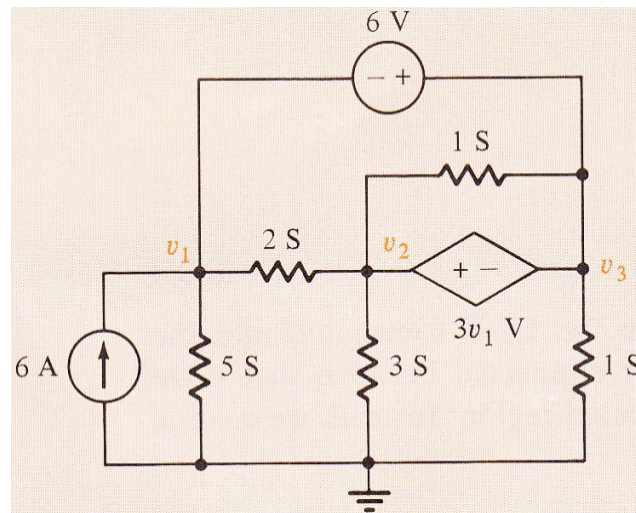
The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with $n = 3$ here.

The answer, **{26.67,6.67,173.33,-46.67}**, is correct.

Bo2's Drill Exercise 2.6

Determine the voltage in each node.

Notice that in the schematic, the resistors' values are given in siemens. Symulator has to be fed the resistors with values in ohms. This means that, when we simulate them, we have to convert them from siemens to ohms by dividing 1 over the siemens value.



My solution:

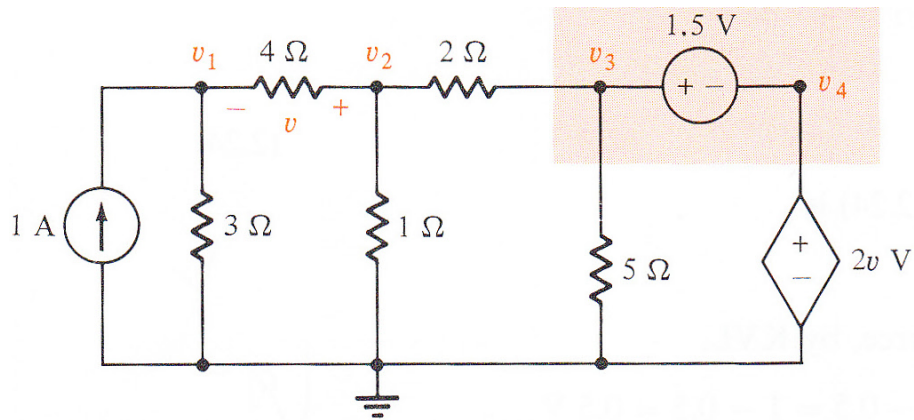
```
j,0,1,6
ei,3,1,6
ed,2,3,3*v1
r5,1,0,1/5
r2,1,2,1/2
r3,2,0,1/3
r1,2,3,1
r4,3,0,1
```

The answers you want are v_1 , v_2 and v_3 , in **Results**.

The answer, **{-1,2,5}**, is correct.

Bo2's Example 2.7

Determine the voltages in all nodes. My solution:



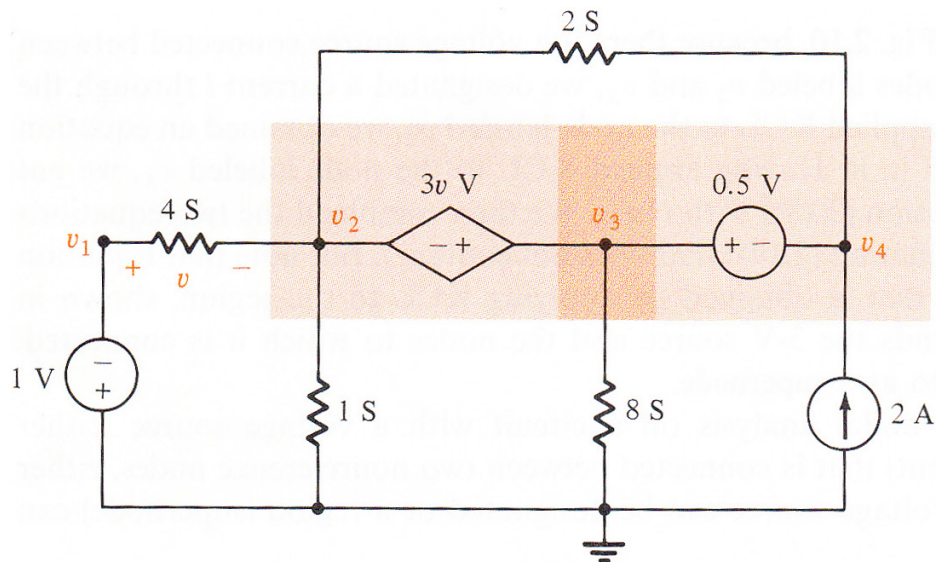
```
j,0,1,1
r3,0,1,3
r4,2,1,4
r1,2,0,1
r2,2,3,2
r5,3,0,5
ei,3,4,1.5
ed,4,0,2*vr4
```

The answers you want are v_1 , v_2 , v_3 and v_4 , in **Results**.

The answer, **{1.5,-.5,-2.5,-4.}**, is correct.

Bo2's Example 2.6

Determine the voltages in all nodes.



My solution:

```
e1,0,1,1
e2,3,4,.5
ed,3,2,3*vr4
j,0,4,2
```

```

r4,1,2,1/4
r1,2,0,1
r8,3,0,1/8
r2,2,4,1/2

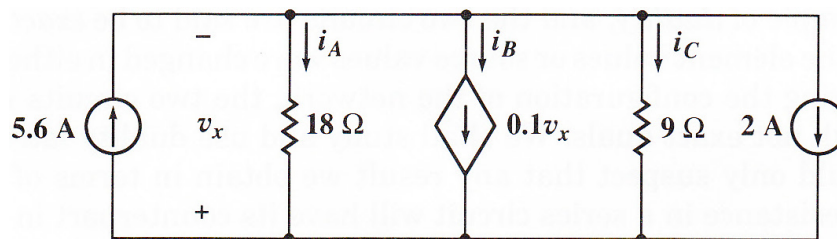
```

The answers you want are v_1 , v_2 , v_3 and v_4 , in **Results**.

The answer, **{-1,-2,1,.5}**, is correct.

HK5's Drill Problem 1-12

Find i_A , i_B and i_C .



My solution:

```

j1,x,0,5.6
ra,0,x,18
jb,0,x,.1*v_x
r9,0,x,9
jr,0,x,2

```

The answers you want are i_{ra} , i_{jb} and i_{r9} , in **Results**.

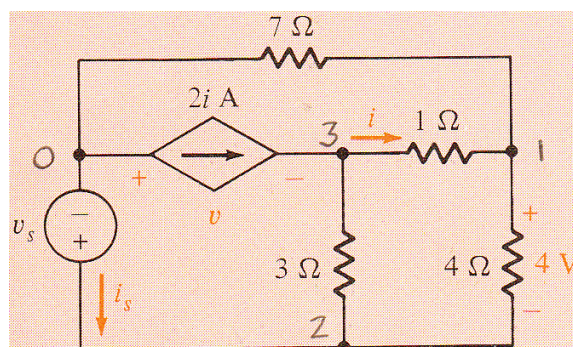
The calculator versions wrap this in approx to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

The answer, **{3.,-5.4,6.}**, is correct.

Numerical-from-symbolic examples

Bo2's Drill Exercise 1.10

Determine i , v , i_s and v_s .



With one unknown value and one known solution, this problem is a job for Expert.

Determine i , v , i_s and v_s .

```
es,2,0,vs
jd,0,3,2*ir1
r7,0,1,7
r1,3,1,1
r3,3,2,3
r4,1,2,4
```

Set **Analysis** to *DC – direct current*. This one needs **Enable Expert Mode** ticked in the **Expert Mode** box; the equations and unknowns go in the fields it reveals.

The answers you want are i_{r1} , v_{jd} and v_s , in **Results**.

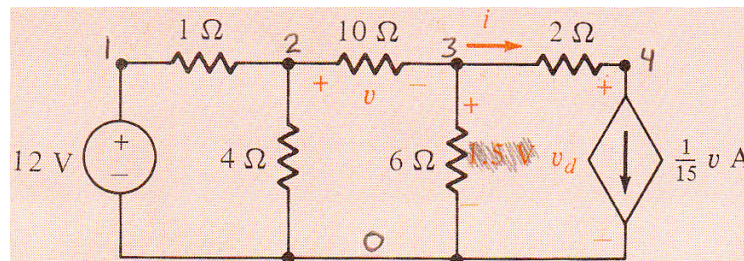
Ask **Evaluate** for:

```
-ies
```

Select DC, Add $v_{r4}=4$ to the equations and v_s to the unknowns. Run the simulation. The answer, **{2,-9,-3,3}**, is correct: $i=2$, $v=-9$, $i_s=-3$ and $v_s=3$.

Bo2's Drill Exercise 1.11

Determine i , v and v_d . (Since all element values are known, the tip we are given by the book – namely, that the voltage drop in the 6Ω resistor is $1.5V$ – is totally superfluous.)



```
e,1,0,12
r1,1,2,1
r4,2,0,4
r10,2,3,10
r6,3,0,6
r2,3,4,2
j,4,0,vr10/15
```

The answers you want are v_{r10} , i_{r2} and v_j , in **Results**.

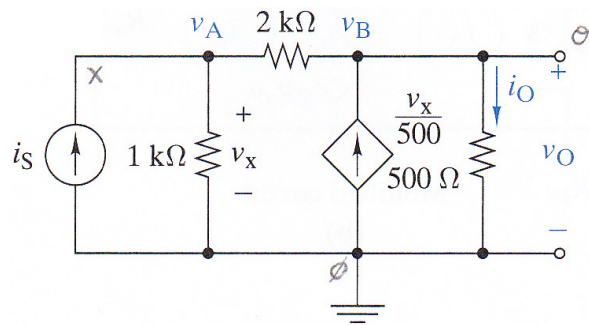
The calculator versions wrap this in approx to get a decimal; version 9 does it through **Rounding** – *approx to n digits*, with $n = 3$ here.

The answer, **{7.5,.5,.5}** is correct: $i=.5$, $v=7.5$ and $v_d=.5$.

Symbolic examples

TR5's Exercise 4.2

Find v_O and i_O in terms of i_S .



This is my solution.

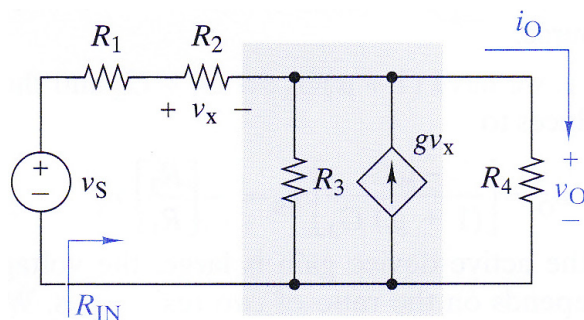
```
ji,0,x,is1
r1,x,0,1'k
r2,x,o,2'k
jd,0,o,vx/500
ro,o,0,500
```

The answers you want are v_O and i_O , in **Results**.

The answer, **{1000*is,2*is}**, is correct: $v_O = 1000 i_S$ and $i_O = 2 i_S$. Version 9's panel names the source's value i_{S1} , so its answers read $1000*i_{S1}$ and $2*i_{S1}$.

TR5's Example 4.4

Find v_O and the equivalent resistance R_{IN} , in terms of v_S , when R_1 is 50Ω , R_2 is $1 \text{ k}\Omega$, R_3 is 100Ω , R_4 is $5 \text{ k}\Omega$ and g is 100 mS (i.e. 100 m).



This is my solution.

```
e,1,0,vs
r1,1,2,50
r2,2,o,1'k
r3,o,0,100
r4,o,0,5'k
j,0,o,100'm*vr2
```

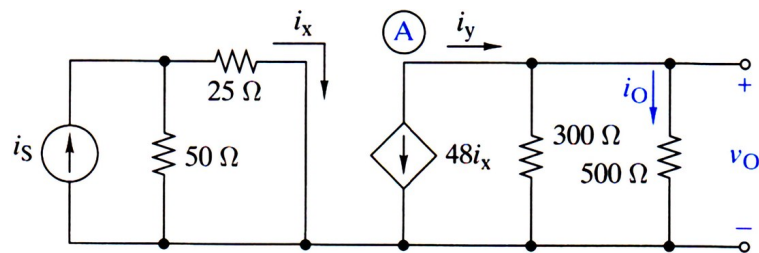
The answers you want are v_O and r_e , in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

The answer, **{.904*vs,10952.}**, is correct: $v_O = .904 v_S$ and $R_{IN} = 10.95 \text{ k}\Omega$.

TR5's Figure 4-4

Find the voltage drop, current, and power consumed by the 500Ω resistor, and the ratio of that power to that delivered by the independent source, all in terms of i_S .



I have not labeled the nodes in the figure, so you can practice doing it. My solution:

```
js,0,1,is1
r50,1,0,50
rx,1,0,25
jd,o,0,48*irx
r3,o,0,300
ro,o,0,500
```

The answers you want are `iro`, `vo` and `pro`, in **Results**.

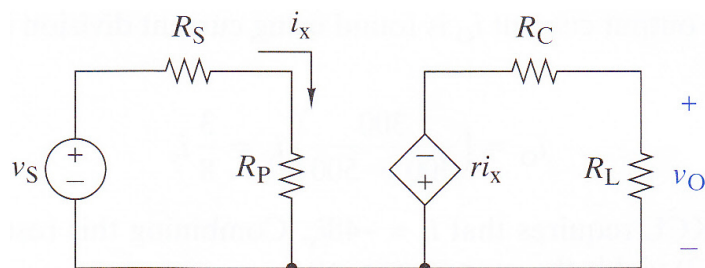
Ask **Evaluate** for:

```
pro/(-pjs)
```

The answers we get are correct: $i_O = -12i_S$, $v_O = -6000i_S$, $p_O = 72000i_S^2$, and $p_O/p_S = 4320$. Version 9's panel names the source's value `is1`, so its answers carry `is1` where these carry `is`.

TR5's Example 4.1 (Symbolic)

Find v_O .



I did not label the nodes, so you can practice.

```

ei,1,0,vs
rs,1,2,rs
rx,2,0,rp
ed,0,3,r*irx
rc,3,0,rc
rl,0,0,rl

```

The answer you want is v_o , in **Results**.

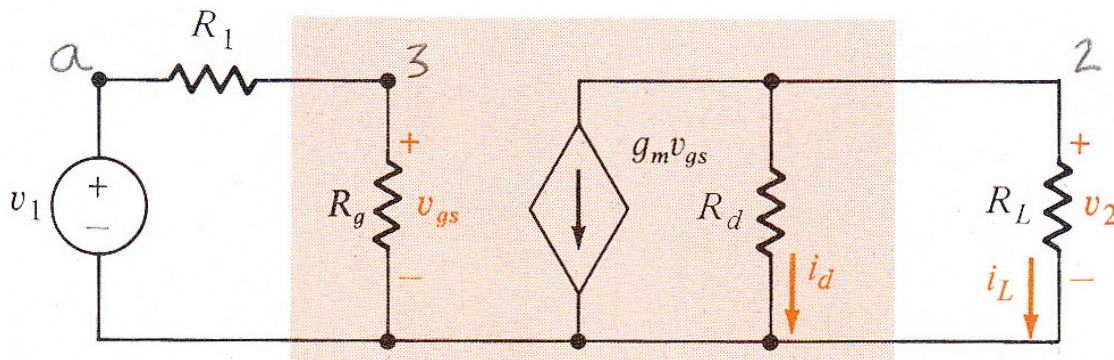
The answer, shown first below, is correct. The textbook's answer follows it.

$$-\frac{r_{rl} v_s}{r_{crp} + r_{crs} + r_{lrp} + r_{lrs}}$$

$$v_o = \left[\frac{-R_L r}{(R_S + R_P)(R_C + R_L)} \right] v_s$$

Bo2's Example 1.11 (Symbolic)

Determine v_2 .



My solution: In my solution I named the value of the source v_1 , to keep it similar to the book. This required avoiding naming any node as **1**: if there was a node 1, Symbolator would store in **v1** the voltage of the node, creating trouble. There is no problem with using **r1** as a value, since nothing will be stored in that **r1** value.

```

e,a,0,v1
r1,a,3,r1
rg,3,0,rg
j,2,0,gm*vrg
rd,2,0,rd
rl,2,0,rl

```

The answer you want is v_2 , in **Results**.

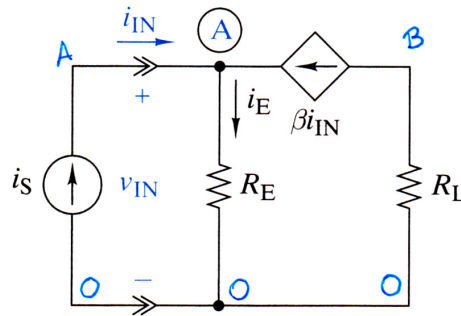
The answer I got is shown first below (the textbook's follows it.)

$$-\frac{g_m r_d r_g r_l v_1}{r_1 r_d + r_1 r_l + r_d r_g + r_g r_l}$$

$$v_2 = \frac{-g_m R_d R_g R_L v_1}{(R_g + R_1)(R_L + R_d)}$$

TR5's Example 4-7 (Symbolic)

Find R_{IN} , i.e. the resistance as seen by the current source. My solution:



```
ji,0,a,is1
re1,a,0,re1
jd,b,a,b*is1
rl,b,0,rl
```

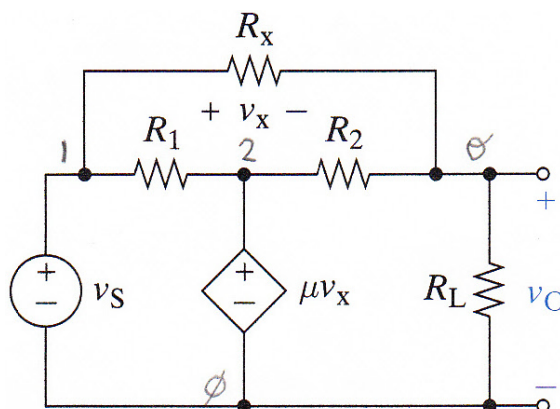
The answer you want is `rji`, in **Results**.

The answer we get $-\text{re1} \cdot (\beta + 1)$, version 9's panel having renamed the element — is correct, as can be seen by comparing it to the textbook's answer.

$$R_{IN} = (\beta + 1)R_E$$

TR5's Exercise 4.3 (Symbolic)

Find v_O , in terms of the value in the circuit. For resistors, use their conductance value.



This is my solution. We use μ as a constant in the dependent source. It is not confused with the SI prefix for micro because the prefix has an apostrophe.


```

ei,1,0,vs
ed,2,0,μ*(vrx)
r1,1,2,1/g1
r2,2,0,1/g2
rx,1,0,1/gx
rl,0,0,1/gl

```

The answer you want is v_o , in **Results**.

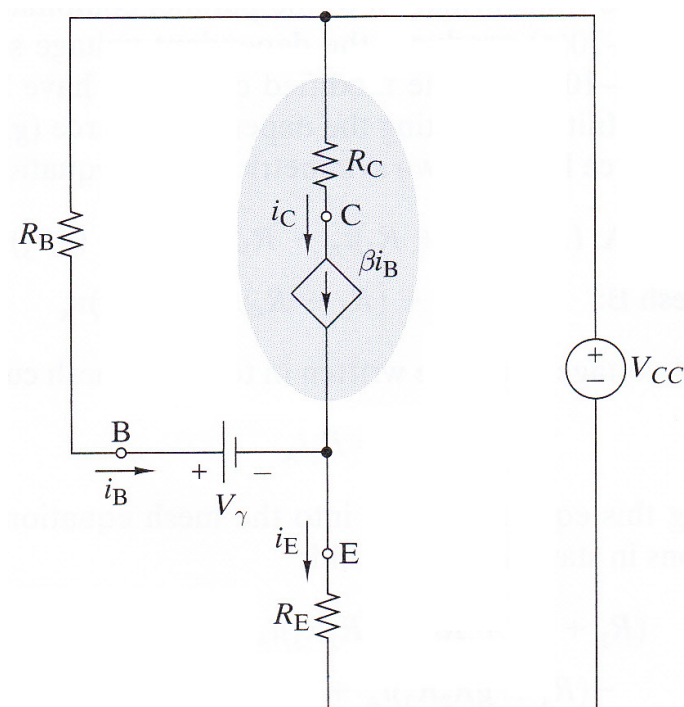
This is the answer we get. It is correct. Compare it with the textbook's answer below it.

$$\frac{v_s (g_2 \mu + g_x)}{g_2 \mu + g_2 + g_l + g_x}$$

$$v_o = \frac{G_x + \mu G_2}{G_x + G_L + (\mu + 1)G_2} v_s$$

TR5's Example 4.5 (Symbolic)

Find i_B .



My solution is shown below.

```

e1,1,0,vcc
rb,1,b,rb
e2,e,b,vγ
re1,e,0,re1
rc,1,c,rc
j,c,e,β*irb

```

The answer you want is i_{rb} , in **Results**.

Compare my answer, first below, to the book's answer beneath it. Note `re1`: as in the previous problem, version 9's panel renamed the element.

$$\frac{vcc + v\gamma}{re1\beta + re1 + rb}$$

$$i_B = \frac{V_{CC} - V_\gamma}{R_B + (\beta + 1)R_E}$$

The last four problems show Symbulator at its DC best. I don't know of any calculator-based program that was able to provide this kind of purely symbolic answer to a circuit simulator back in 1999 when I made Symbulator. As a matter of fact, even today – a quarter of a century later - I know of no other calculator-based simulator that can do this.

LESSON 4

Shorts, equivalent resistance and Thévenin/Norton

Learn how to describe a *short circuit* using the **s** element. Learn how to find *equivalent resistances* using the **er** script, and *Thevenin and Norton equivalents* using the **th** script.

In this lesson you will learn how to describe a *short circuit* with the **s** element, how to find *equivalent resistances* with the **er** script, and *Thévenin and Norton equivalents* with the **th** script.

4.1 How to describe a short circuit

Shorts are used mostly to find a current in a part of the circuit with no element in it already. Otherwise we would just make it a single node.

NOTE

Describing short circuits

In Symbolator, an ideal short circuit is described with three bits of information, separated by commas: the name of the short, which must start with the letter **s**, then the names of its first and second node.

For example, an ideal short circuit called **s1**, between nodes **3** and **5**, is described:

s1,3,5

What answers do you get

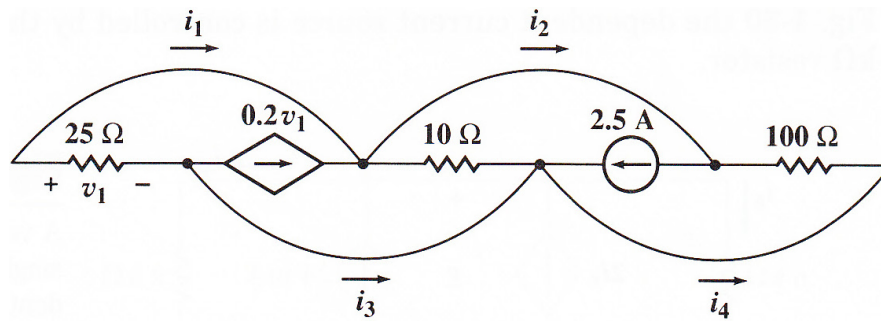
No power is consumed, and no voltage is dropped, in a short circuit. For each short in a circuit, Symbolator stores only the current through it, flowing from the first node towards the second. For a short called **sx**, that is the **current through** line of the **sx** block in **Results**.

HK5's Drill Problem 1-13

Find i_1 , i_2 , i_3 and i_4 .

SOLUTION

My solution: I define the shorts in the same direction as the arrows in the schematic.



HK5's Drill Problem 1-13

```
r1,1,0,25
jd,0,2,0.2*v1
r2,2,3,10
ji,4,3,2.5
r3,4,5,100
s1,1,2
s2,2,4
s3,0,3
s4,3,5
```

Choose *Solve circuit* and *DC*, then run it. Each short has its own block in **Results**, and the number you want is on its **current through** line:

s1 reads **-2** A, s2 **3** A, s3 **-8** A and s4 **-0.5** A.

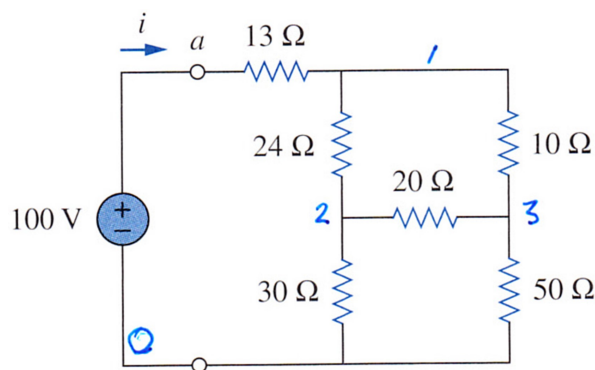
These are correct, and they can only be found using short circuits.

4.2 The equivalent resistance script: er

As we saw, Symbulator gives the equivalent resistance of a circuit as seen from any source. That lets us solve problems like this one.

AS2's Practice Problem 2.15

Find the equivalent resistance as seen by the 100 V source, and the value of current i .



AS2's Practice Problem 2.15

SOLUTION

```
e,a,0,100
r13,a,1,13
r24,1,2,24
r10,1,3,10
r20,2,3,20
r30,2,0,30
r50,3,0,50
```

Solve it in DC. There is no special tool to reach for here: because a source sits between the two nodes we care about, the equivalent resistance is one of the answers Symbolator works out anyway. In the block for e , the **resistance seen** line reads $r_e = 40 \Omega$.

The current i is the current through r_{13} , and its block gives $i_{r13} = 2.5 \text{ A}$.

Both answers came free, because there was a source connected between the two desired nodes. The rest of this lesson is about what to do when there is not.

What if there is no independent source?

But how do we find the equivalent resistance of a **passive circuit**, one with no *independent* source in it? One way is to connect a 1 A current source between the two nodes and read the voltage drop across it. That is the manual way.

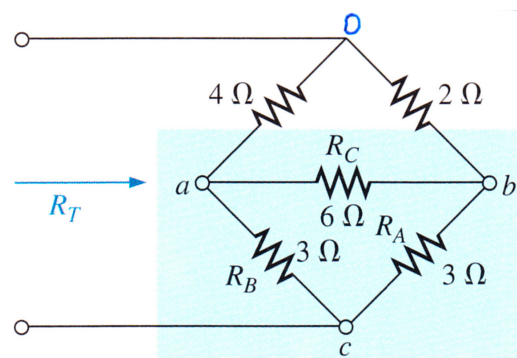
An easier way is to let Symbolator do that for us.

Version 9 puts this on the **Type of analysis** menu. Choose *Find equivalent* instead of *Solve circuit*, and a second menu appears – **Type of equivalent** – whose first entry is *Resistance / impedance*. Two node boxes appear with it, for the pair of terminals you are measuring between.

That is the whole of it: no source to invent, no voltage to divide. Symbolator does the 1 A trick internally and reports the answer.

B11's Example 8.29

Calculate the equivalent resistance of the circuit shown.



B11's Example 8.29

SOLUTION

Let me solve this problem step by step. After I label the nodes, I describe the circuit and store it in a variable.

```
r4,0,a,4
r2,0,b,2
r6,a,b,6
rb,a,c,3
ra,b,c,3
```

Set **Type of analysis** to *Find equivalent*, **Type of equivalent** to *Resistance / impedance*, and the two node boxes to **0** and **c**. Leave **Analysis** on *DC – direct current* and press **Run Symulator**.

Results looks different from a normal solve: instead of the node and element listing there is a single block headed **Equivalent impedance**, with one line in it.

Req = 2.89 Ω

The value is 2.89 Ω . This is correct.

What counts as passive

That example was made of resistors only. The *er* script also handles a second kind of passive circuit: one with resistors and dependent sources, but no independent sources. Such a circuit can only be reduced to an equivalent resistance, not to a Thévenin or Norton equivalent, so *er* is the script to use – in exactly the same way, provided the dependent sources are described properly.

4.3 The Thévenin / Norton script: *th*

Just as a *passive circuit* can be reduced to an equivalent resistance, an **active circuit** – one with independent sources – can be reduced to a Thévenin or Norton equivalent.

One way is to run a first simulation for the voltage between the two nodes where we want the equivalent (the Thévenin voltage, *VTH*), then a second with a short between those nodes for the current through it (the Norton current, *INO*). *REQ* is then *VTH/INO*. That is the manual way.

An easier way is the ***th*** script, which does exactly that automatically. It takes three arguments: the circuit description, the first node and the second node.

RM3's Practice Problem 9-4

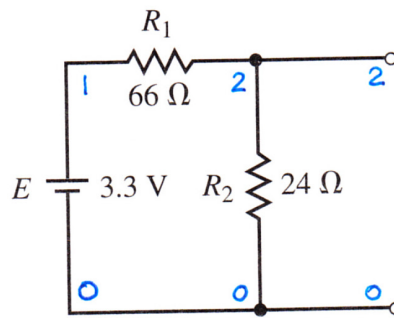
Find the Thévenin and Norton equivalents of the circuit.

SOLUTION

This is my circuit description, and this is how we run the *th* script:

```
e,1,0,3.3
r1,1,2,66
r2,2,0,24
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **2** in the first and **0** in the second – the pair of terminals you are



RM3's Practice Problem 9-4

looking into. Leave **Analysis** on *DC – direct current* and press **Run Symulator**.

The answers appear in **Results** under these names:

- **vth** has the Thévenin voltage
- **ino** has the Norton current
- **req** has the equivalent resistance
- **pmax** has the maximum power that the equivalent can deliver to a hypothetical load

Results replaces the usual node-and-element listing with a single block headed **Thévenin / Norton equivalent**. There are no prompts and no second step; all four answers arrive together:

- **Thevenin voltage**, $v_{th} = 880 \text{ mV}$
- **Norton current**, $i_{no} = 50 \text{ mA}$
- **Equivalent resistance**, $R_{eq} = 17.6 \text{ } \Omega$
- **Maximum deliverable power**, $p_{max} = 11 \text{ mW}$

The first three are what you came for. The fourth is Symulator being generous – the most this circuit could deliver into a matched load.

Problems with a load

One type of problem books and professors like when teaching the Thévenin / Norton equivalents is what I call an RL problem. A typical one goes like this: “First, reduce the circuit, as seen by resistor R_L , to its Thévenin or Norton equivalent. Then, find the value of the voltage drop, current and/or power consumed in the load resistor R_L if its value is (whatever) ohms.”

Symulator 9 does not ask, and it does not carry the load quantities on the results either. **Results** shows four answers and no more: v_{th} , i_{no} , r_{eq} and p_{max} .

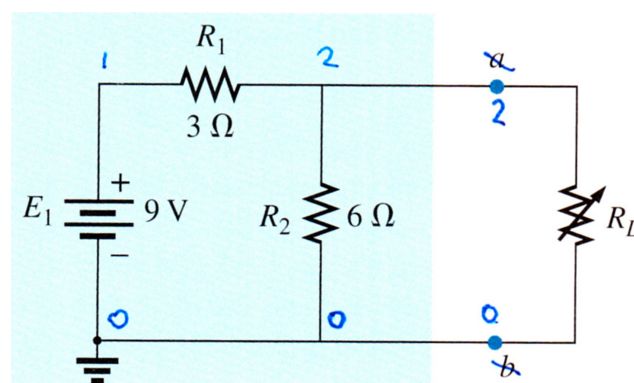
That is no great loss, because each load quantity is one line in **Evaluate**, built from the two answers you already have. Writing R for the load resistance:

To find	Type into Evaluate
the current in the load	$v_{th}/(r_{eq}+R)$
the voltage drop in the load	$v_{th}*R/(r_{eq}+R)$
the power consumed in the load	$v_{th}^2*R/(r_{eq}+R)^2$

Put the load's actual value where R is. The rest of this section reads exactly as it does on the calculator, with those three in place of the calculator's i_{rL} , v_{rL} and p_{rL} .

B11's Example 9.6

Find the Thévenin equivalent circuit for the network in the shaded area. Then find the current through R_L for R_L values of $2\ \Omega$, $10\ \Omega$ and $100\ \Omega$.



B11's Example 9.6

SOLUTION

First we find the Thévenin equivalent.

```
e,1,0,9
r1,1,2,3
r2,2,0,6
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **2** in the first and **0** in the second. Choose *DC* and run it.

Results gives $v_{th} = 6\text{ V}$ and $r_{eq} = 2\ \Omega$.

Correct. Now we find the values of the current in the load for the different values. We can do this in a single push, or separately. Here I find them in one go:

There is no single push here — you ask **Evaluate** three times, once per load, and the answers come back one at a time:

```
vth/(req+2)
```

Then $v_{th}/(r_{eq}+10)$, then $v_{th}/(r_{eq}+100)$.

The three answers come back one at a time: **1.5 A**, **.5 A** and **.059 A**.

The answers are correct.

Power transfer problems

Another problem often associated with the Thévenin / Norton equivalents is power transfer to a load, particularly the maximum possible. Maximum power is transferred when the load R_L equals the R_{EQ} of the equivalent. Symbulator's `th` script gives you the maximum power that can be delivered in `pmax`, and the power transferred to the load as a function of its value in the expression $v_{th}^2 \cdot R / (r_{eq} + R)^2$ derived above.

What if it's more than a load?

The formulas for the current, voltage and power in the load apply only when a load is the only thing connected to the equivalent circuit. If the problem you want to solve includes something more complicated, you will have to run your own simulation.

Symbulator 9 writes no such string for you, and does not need to: the equivalent is three lines, and you already have both numbers on screen. Type them into a fresh **Circuit Description**, putting `ino` and `req` where they belong:

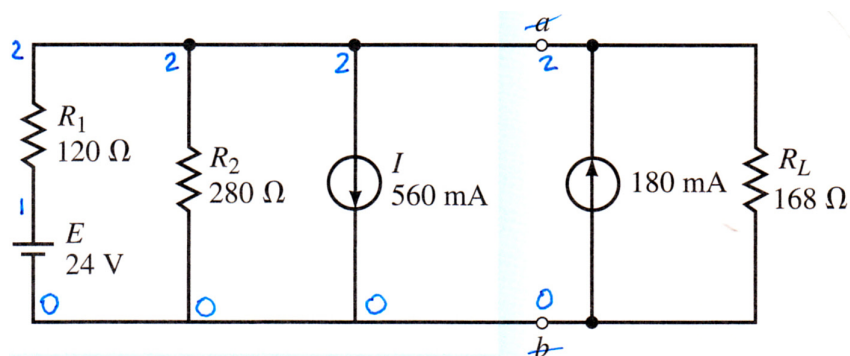
```
jn,0,n,ino  
re1,n,0,req  
r1,n,0,load
```

Replace `ino` and `req` with the numbers **Results** just gave you. The load is called `r1` and its value is the symbol `load`, so the answers come back in terms of it.

You can use it as a starting point for a new simulation.

RM3's Example 9-8

Find the Norton equivalent of the circuit left of a-b; then find the current through R_L .



RM3's Example 9-8

SOLUTION

Let's first find the circuit equivalent:

```
e,1,0,24  
r1,1,2,120  
r2,2,0,280  
j,2,0,560'm
```

Find equivalent, Thévenin / Norton, nodes **2** and **0**, in DC.

Results gives $i_{no} = -0.36$ A and $r_{eq} = 84 \Omega$.

The sign differs from the calculator's, which prints **0.36** A: version 9 reports the Norton current in the direction it actually flows, from the first node to the second. Nothing else changes — carry the sign through and the load current below comes out the same.

Correct. Now to the second part of the question. In order to find the current through R_L , we cannot use the load expressions, because now the load is not the only thing connected to the terminals of the equivalent: there is also a current source. We have to run a new simulation.

The fastest way is to start from the equivalent circuit description:

```
"jN,0,n,iNo:rE,n,0,rEq:rL,n,0,L"
```

We change the value of the load to 168Ω , and add the 180 mA source flowing from node 0 to node n. Then we run a dc simulation and ask for the current in the load:

```
jn,0,n,-0.36  
re1,n,0,84  
r1,n,0,168  
j,0,n,180'm
```

Run it in DC. The $r1$ block's **current through** line reads **-0.06** A.

Correct: there is a current of 60 mA flowing through R_L from 0 to n.

Using the equivalent circuit description is meant to save you time. If you find it confusing to use, just don't use it.

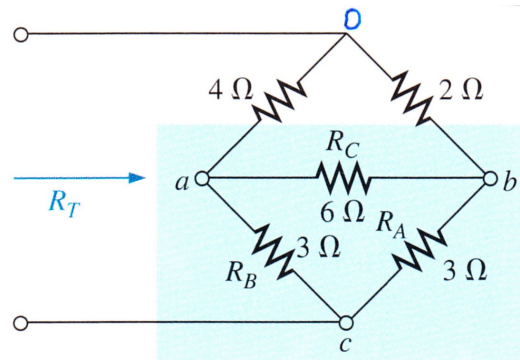
4.4 Instructive solved examples

NOTE

Some of this narration still describes the calculator

Every problem below carries a **Circuit Description** panel you can type straight into Symbulator 9, and the circuits and answers are identical across the versions. What has not all been rewritten is the narration between them: where it says to evaluate a name or press a key, do the version 9 equivalent — the results are already on screen.

Practice problems for resistive circuits



B11's Example 8.29

Calculate the equivalent resistance of the circuit shown.

You saw this circuit solved step by step in the walkthrough above; here it is again, in the compact form the rest of these problems use.

After I label the nodes, I describe the circuit. In this case, I store it in a variable.

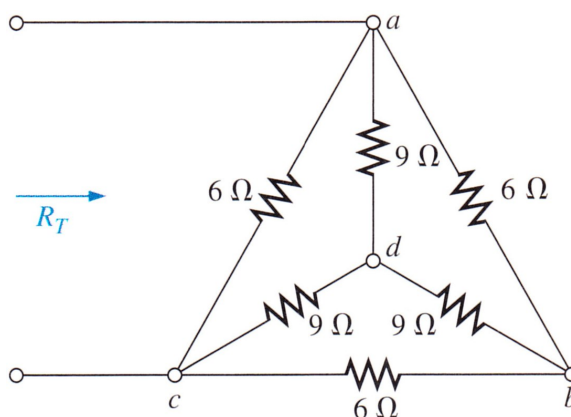
```
r4,0,a,4
r2,0,b,2
r6,a,b,6
rb,a,c,3
ra,b,c,3
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Resistance / impedance*, with nodes **0** and **c**, in DC. The answer is $r_{eq} = 2.889 \Omega$.

This is correct. Below are many practice examples of this type.

B11's Example 8.30

Find the equivalent resistance of the circuit.



```
rac,a,c,6
rad,a,0,9
rab,a,b,6
```

```

rkd,c,0,9
rbc,b,c,6
rbd,b,0,9

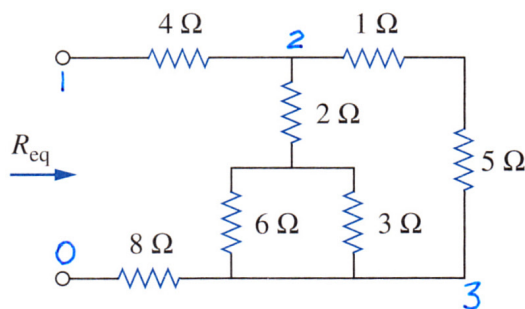
```

Find equivalent, Resistance / impedance, nodes **a** and **c**, in DC: r_{eq} is **3.273 Ω** .

The calculator description names its four nodes a, b, c and d. Version 9 needs one of them to be the reference node, so d is called 0 here instead. Which node you ground makes no difference to the answer in a network with no sources – grounding b gives the same **3.273 Ω** .

AS2's Example 2.9

Find the equivalent resistance of the circuit.



We don't need to run a simulation for this. We can reduce it using `s\pr`.

```

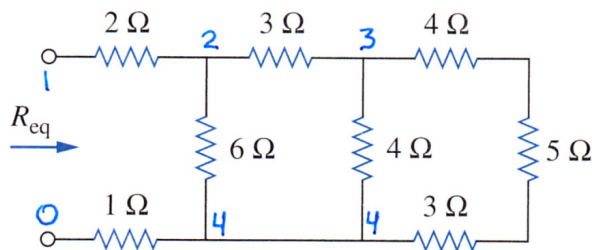
4+pr(1+5,2+pr(6,3))+8

```

Evaluating approximately gives us the equivalent resistance: **14.4 Ω** .

AS2's Practice Problem 2.9

Find the equivalent resistance of the circuit.



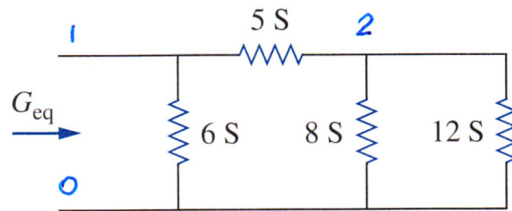
We don't need to run a simulation for this. We can reduce it using `s\pr`.

```

2+pr(6,3+pr(4,4+5+3))+1

```

Evaluating approximately gives us the equivalent resistance: **6 Ω** .



AS2's Example 2.11

Find the equivalent *conductance* of the circuit.

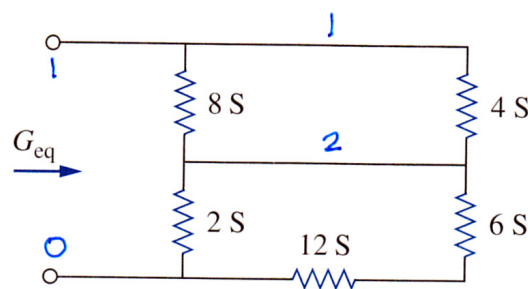
We don't need to run a simulation for this. We can reduce it using `s\pr`.

```
1/(pr(1/6,1/5+pr(1/8,1/12)))
```

Evaluating approximately gives us the equivalent conductance: **10 S**.

AS2's Practice Problem 2.11

Find the equivalent *conductance* of the circuit.



We don't need to run a simulation for this. We can reduce it using `s\pr`.

```
1/(pr(1/8,1/4)+pr(1/2,1/12+1/6))
```

Evaluating approximately gives us the equivalent conductance: **4 S**.

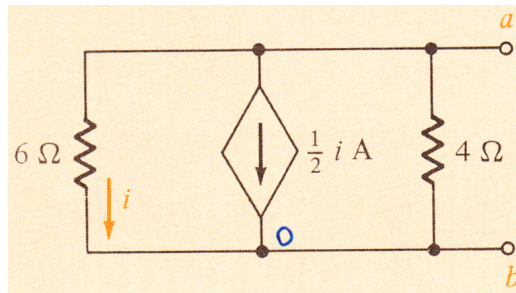
Examples with dependent sources

Bo2's Drill Problem 3.14

Find the equivalent *resistance* of the circuit.

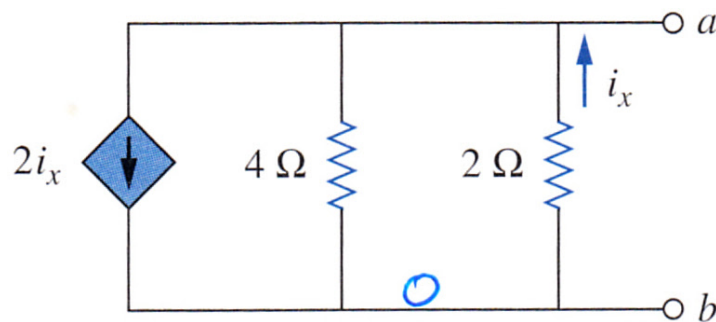
```
r4,a,0,4
ri,a,0,6
ji,a,0,iri/2
```

Find equivalent, Resistance / impedance, nodes **a** and **0**, in DC: r_{eq} is **2 Ω** .



AS2's Example 4.10

Find the equivalent resistance of the circuit.

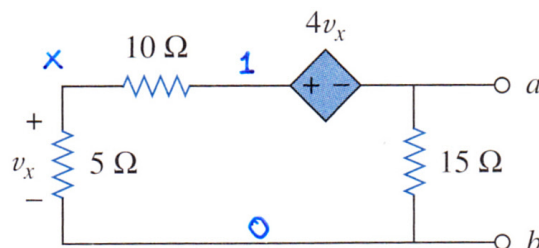


```
r4,a,0,4
rx,0,a,2
j,a,0,2*irx
```

Find equivalent, Resistance / impedance, nodes **a** and **0**, in DC: r_{eq} is **-4 Ω**. It may be surprising to have a negative resistance. This is the result of the dependent sources.

AS2's Practice Problem 4.10

Find the equivalent resistance of the circuit.

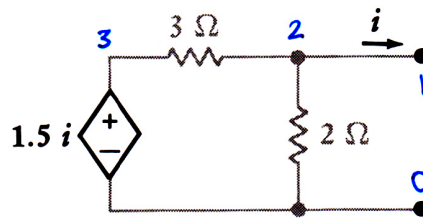


```
r15,a,0,15
e,1,a,4*vr_x
r10,1,x,10
rx,x,0,5
```

Find equivalent, Resistance / impedance, nodes **a** and **0**, in DC: r_{eq} is **-7.500 Ω**.

HK5's Figure 2-29

Find the equivalent resistance of the circuit.

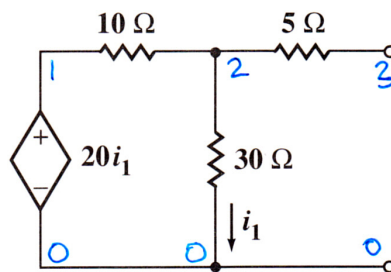


```
e,3,0,1.5*is1
r3,3,2,3
r2,2,0,2
s1,2,1
```

Find equivalent, Resistance / impedance, nodes **1** and **0**, in DC: r_{eq} is **0.6000** Ω . Only the short is renamed, s to $s1$: a short named bare s would have its current spelled is , which already means something in Python, so version 9 refuses that one name and suggests $s1$. The dependent value follows it — $is1$.

HK5's Drill Problem 2-9d

Find the equivalent resistance of the circuit.



```
r10,1,2,10
r5,2,3,5
r1,2,0,30
e,1,0,20*ir1
```

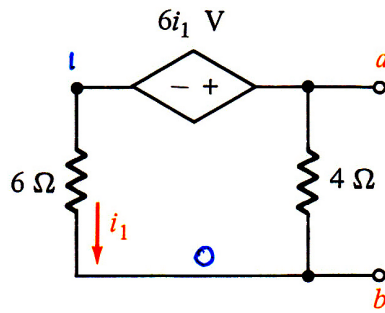
Find equivalent, Resistance / impedance, nodes **3** and **0**, in DC: r_{eq} is **20** Ω .

Bo2's Example 3.12

Find the equivalent resistance of the circuit.

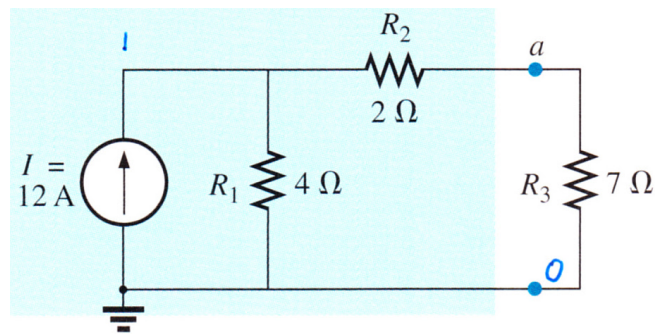
```
r1,1,0,6
r4,a,0,4
e,a,1,6*ir1
```

Find equivalent, Resistance / impedance, nodes **a** and **0**, in DC: r_{eq} is **3** Ω .



B11's Example 9.7

Find the Thévenin equivalent of the circuit, as seen from the R_3 resistor.



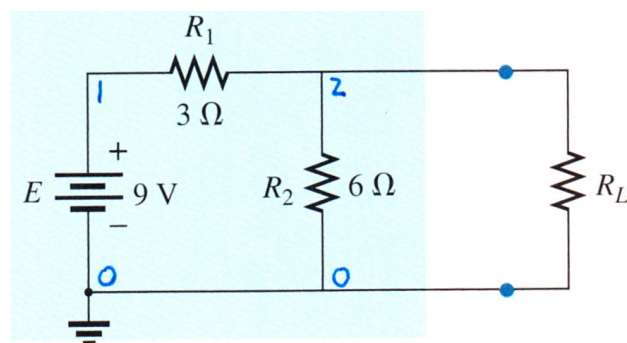
```
j,0,1,12
r1,1,0,4
r2,1,a,2
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **a** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via v_{th} we find $V_{TH} = 48$ V. Via r_{eq} we find $R_{EQ} = 6$ Ω.

B11's Example 9.11

Find the Norton equivalent of the circuit, as seen from the R_L resistor.



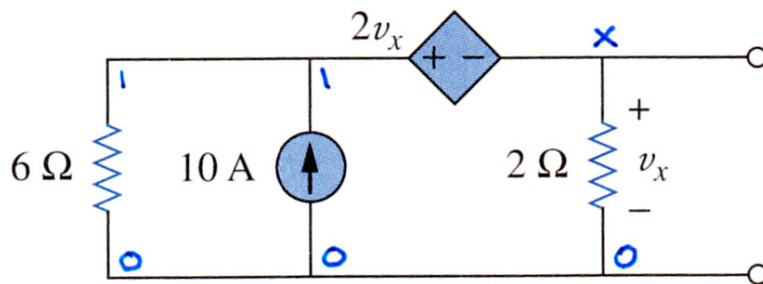

```
e,1,0,9
r1,1,2,3
r2,2,0,6
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **2** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via *ino* we find $I_{NO} = 3 \text{ A}$. Via *req* we find $R_{EQ} = 2 \Omega$.

AS2's Practice Problem 4.12

Find the Norton equivalent of the circuit.

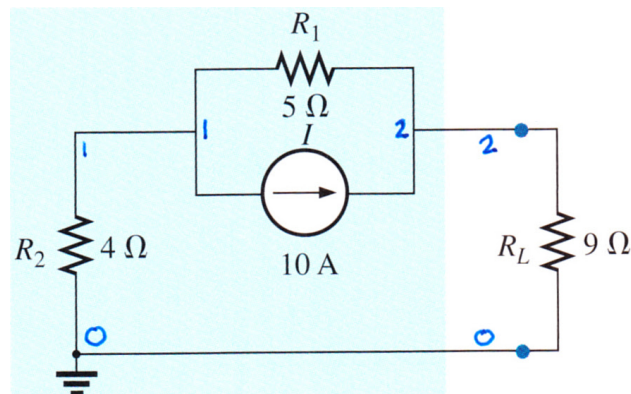


```
r6,1,0,6
j,0,1,10
r2,x,0,2
e,1,x,2*v_x
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **x** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via *ino* we find $I_{NO} = 10 \text{ A}$. Via *req* we find $R_{EQ} = 1 \Omega$.

B11's Example 9.12



Find the Norton equivalent of the circuit, as seen from the R_L resistor.

```

r2,1,0,4
r1,1,2,5
j,1,2,10

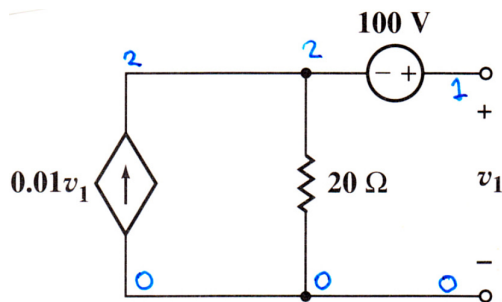
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **2** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via i_{no} we find $I_{NO} = 5.56$ A. Via r_{eq} we find $R_{EQ} = 9 \Omega$.

HK5's Drill Problem 2-8b

Find the Thévenin equivalent of the circuit.



```

j,0,2,0.01*v1
r,0,2,20
e,1,2,100

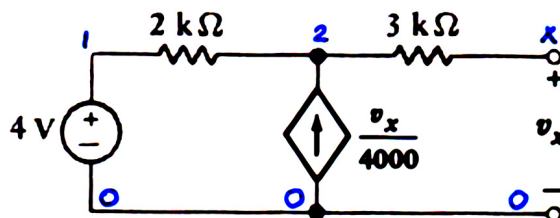
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **1** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via v_{th} we find $V_{TH} = 125$ V. Via r_{eq} we find $R_{EQ} = 25 \Omega$.

HK5's Figure 2-27

Find the Thévenin equivalent of the circuit.



```

e,1,0,4
r2,1,2,2'k
r3,2,x,3'k
j,0,2,vx/4000

```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **x** in the first and **0** in the second — the pair of terminals you are

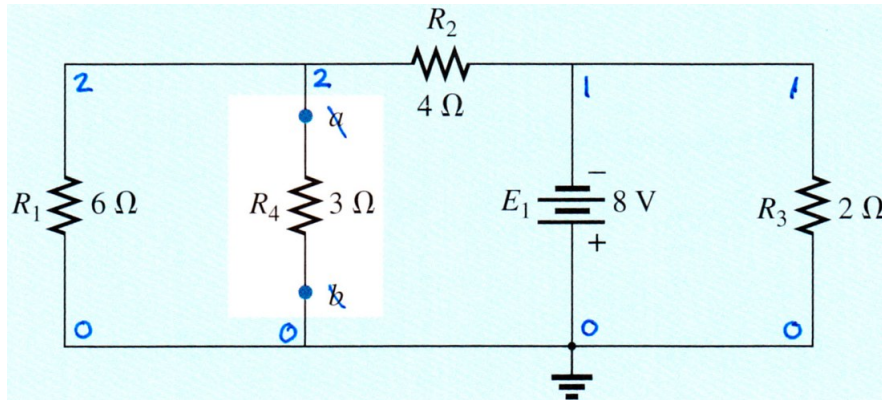
looking into.

Via v_{th} we find $V_{TH} = 8 \text{ V}$. Via r_{eq} we find $R_{EQ} = 10 \text{ k}\Omega$.

B11's Example 9.8

Find the Thévenin equivalent of the circuit, as seen from the R_4 resistor.

I ignore the textbook's decision to call the nodes **a** and **b**, since b is ground anyway.



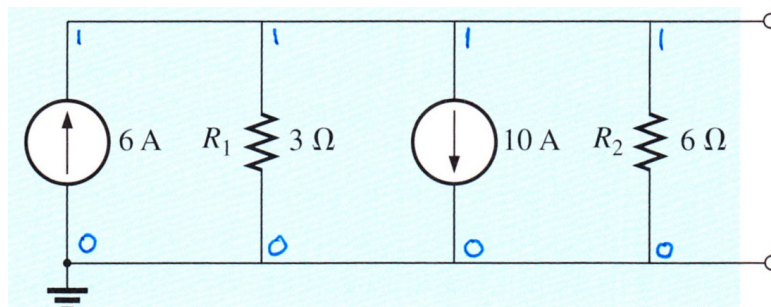
```
r1,2,0,6
r2,2,1,4
r3,1,0,2
e,0,1,8
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **2** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via v_{th} we get $V_{TH} = -4.8 \text{ V}$. Via r_{eq} we get $R_{EQ} = 2.4 \Omega$.

B11's Example 8.6

Find the Norton equivalent of the circuit.



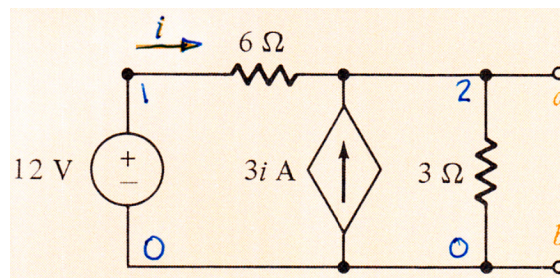
```
j1,0,1,6
r1,1,0,3
j2,1,0,10
r2,1,0,6
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **1** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via i_{no} we find $I_{NO} = -4$ A. Via r_{eq} we find $R_{EQ} = 2 \Omega$.

Bo2's Drill Exercise 3.12

Find the Norton equivalent of the circuit.



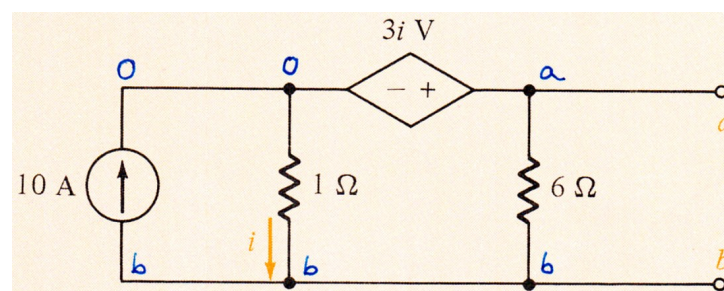
```
e,1,0,12
r6,1,2,6
j,0,2,3*ir6
r3,2,0,3
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **2** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via i_{no} we find $I_{NO} = 8$ A. Via r_{eq} we find $R_{EQ} = 1 \Omega$.

Bo2's Drill Exercise 3.9

Find the Thévenin equivalent of the circuit.



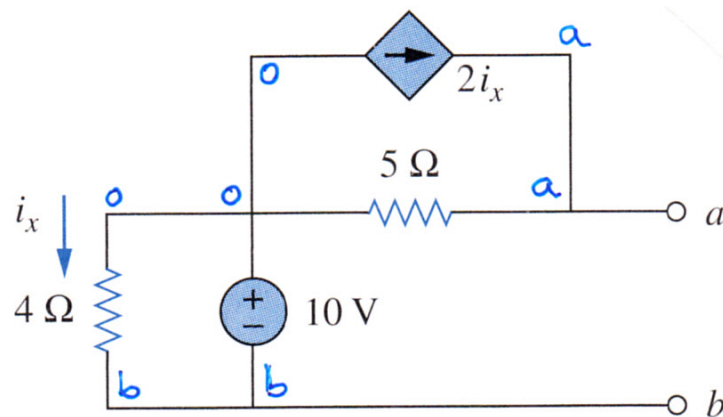
```
j,b,0,10
r1,0,b,1
e,a,0,3*ir1
r6,a,b,6
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **a** in the first and **b** in the second — the pair of terminals you are looking into.

Choose DC. Via v_{th} we find $V_{TH} = 24$ V. Via r_{eq} we find $R_{EQ} = 2.4 \Omega$.

AS2's Example 4.12

Find Norton equivalent of the circuit.



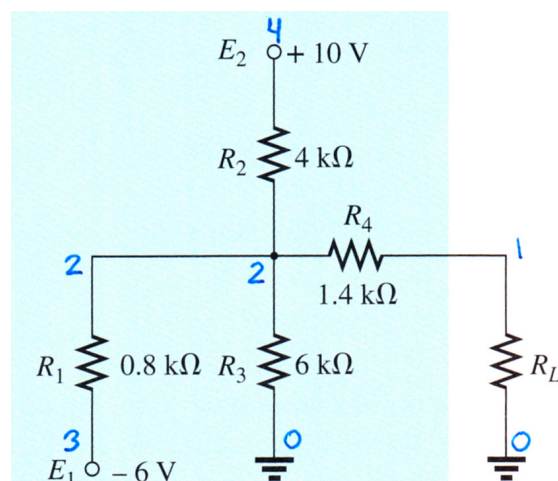
```
rx,0,b,4  
e,0,b,10  
r5,0,a,5  
j,0,a,2*irx
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin/Norton*. Two node boxes appear: put **a** in the first and **b** in the second — the pair of terminals you are looking into.

Choose DC. Via *ino* we find $I_{NO} = 7$ A. Via *req* we find $R_{EQ} = 5 \Omega$.

B11's Example 9.10 (Hidden source)

Find the Thévenin equivalent of the circuit.



My solution is below:

```
e1,3,0,-6  
e2,4,0,10  
r1,2,3,0.8'k
```

```

r2,2,4,4'k
r3,2,0,6'k
r4,2,1,1.4'k

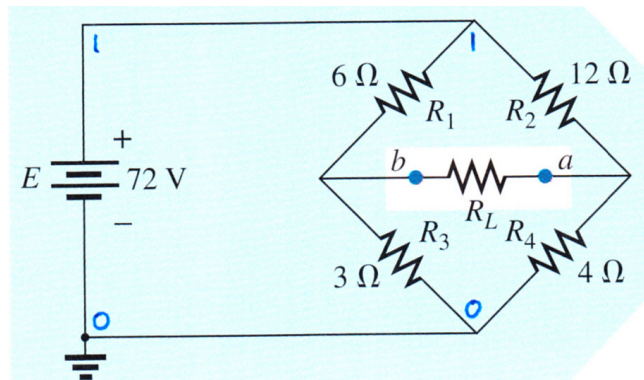
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **1** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via v_{th} we find $V_{TH} = -3$ V. Via r_{eq} we find $R_{EQ} = 2$ k Ω .

B11's Example 9.9

Find the Thévenin equivalent of the circuit, as seen from the R_L resistor.



```

e,1,0,72
r1,1,b,6
r2,1,a,12
r3,0,b,3
r4,0,a,4

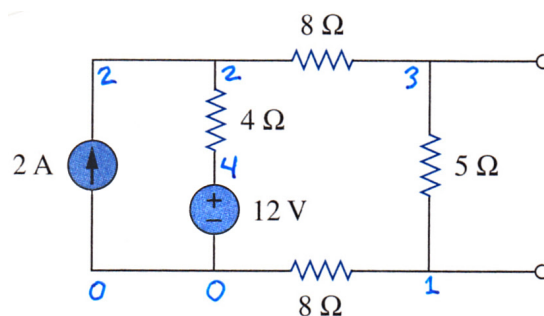
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **b** in the first and **a** in the second — the pair of terminals you are looking into.

Choose DC. Via v_{th} we find $V_{TH} = 6$ V. Via r_{eq} we find $R_{EQ} = 5$ Ω .

AS2's Example 4.11

Find the Norton equivalent of the circuit.



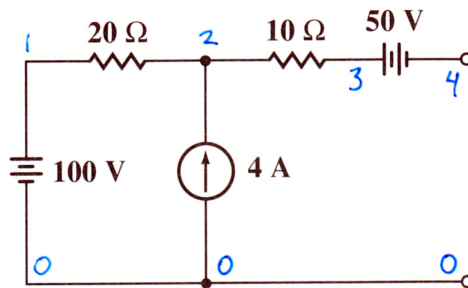

```
j,0,2,2
e,4,0,12
r1,2,4,4
r2,2,3,8
r3,0,1,8
r4,3,1,5
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **3** in the first and **1** in the second — the pair of terminals you are looking into.

Choose DC. Via i_{no} we find $I_{NO} = 1$ A. Via r_{eq} we find $R_{EQ} = 4 \Omega$.

HK5's Drill Problem 2-8a

Find the Thévenin equivalent of the circuit.



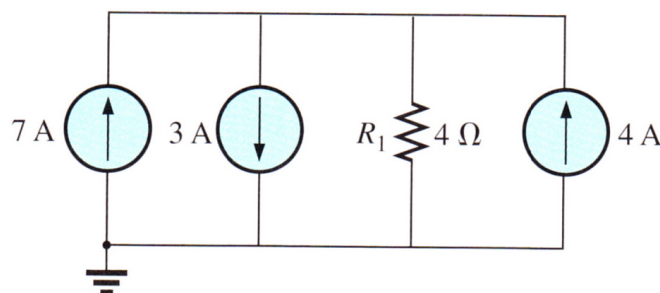
```
e1,1,0,100
r2,1,2,20
j,0,2,4
r1,2,3,10
e5,3,4,50
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **4** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via v_{th} we find $V_{TH} = 130$ V. Via r_{eq} we find $R_{EQ} = 30 \Omega$.

B11's Example 8.7

Find Norton equivalent of the circuit.



```

j7,0,1,7
j3,1,0,3
r1,1,0,4
j4,0,1,4

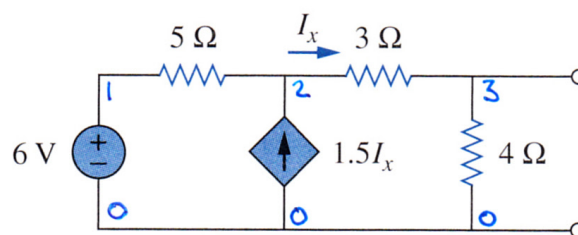
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **1** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via `ino` we find $I_{NO} = 8 \text{ A}$. Via `req` we find $R_{EQ} = 4 \Omega$.

AS2's Practice Problem 4.9

Find the Thévenin equivalent of the circuit.



```

e,1,0,6
r5,1,2,5
rx,2,3,3
j,0,2,1.5*irx
r4,3,0,4

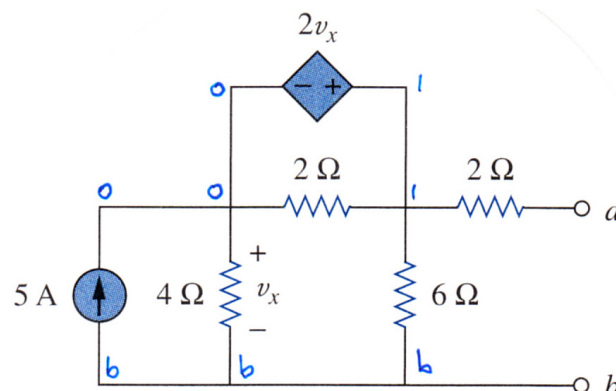
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **3** in the first and **0** in the second — the pair of terminals you are looking into.

Choose DC. Via `vth` we find $V_{TH} = 5.33 \text{ V}$. Via `req` we find $R_{EQ} = 0.44 \Omega$.

AS2's Example 4.9

Find the Thévenin equivalent of the circuit.



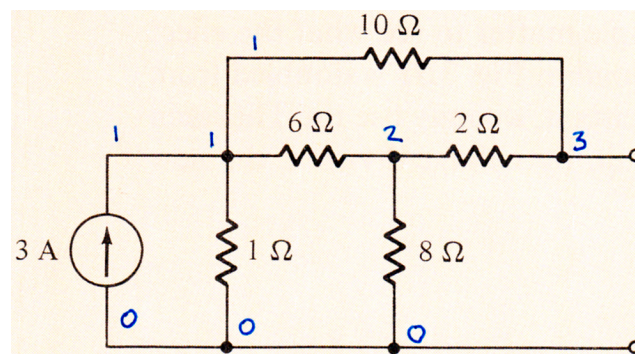

```
j,b,0,5
rx,0,b,4
r1,0,1,2
r2,1,b,6
r3,1,a,2
e,1,0,2*vrx
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **a** in the first and **b** in the second — the pair of terminals you are looking into.

Choose DC. Via v_{th} we find $V_{TH} = 20$ V. Via r_{eq} we find $R_{EQ} = 6$ Ω .

Bo2's Drill Exercise 3.8

Find the Thévenin equivalent of the circuit.



```
j,0,1,3
r1,0,1,1
r6,1,2,6
r10,1,3,10
r8,2,0,8
r2,2,3,2
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **3** in the first and **0** in the second — the pair of terminals you are looking into.

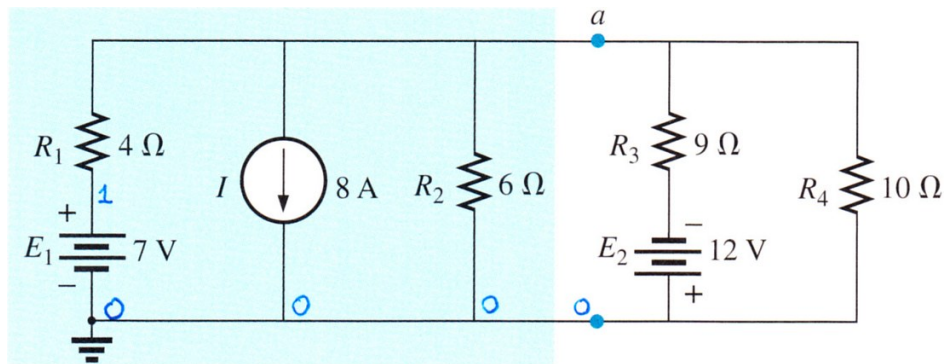
Choose DC. Via v_{th} we find $V_{TH} = 2$ V. Via r_{eq} we find $R_{EQ} = 4$ Ω .

B11's Example 9.13

Find Norton equivalent of the shaded part of the circuit.

```
e,1,0,7
r1,1,a,4
j,a,0,8
r2,a,0,6
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **a** in the first and **0** in the second — the pair of terminals you are looking into.



looking into.

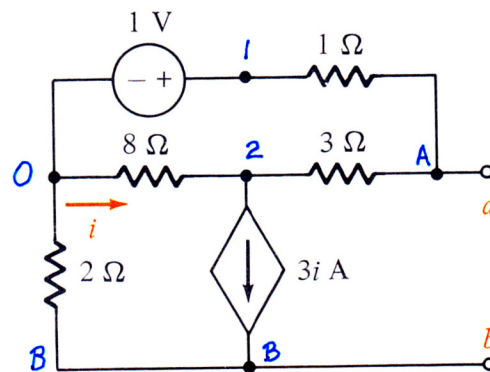
Choose DC. Via `ino` we find $I_{\text{NO}} = -6.25\text{ A}$. Via `req` we find $R_{\text{EQ}} = 2.4\Omega$.

Tricky problems

Some circuit theory books – and some professors – find it entertaining or instructive to surprise unsuspecting students with tricky problems, like the two we solve below.

Bo2's Example 3.11 (Tricky)

Find the Norton equivalent of the circuit.



Since you do not know beforehand that this is a tricky problem, you go for the usual:

```
r2,0,b,2
r8,0,2,8
r3,2,a,3
r1,1,a,1
e,1,0,1
j,2,b,3*ir8
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **a** in the first and **b** in the second – the pair of terminals you are looking into.

Run it in DC and it is refused: *Could not solve the system of equations*. This often means there is a division by zero somewhere.

Symbulator's own advice is to try again with a symbolic value, which is what we do. We

chose to use **x** instead of **3** in the dependent source.

```
r2,0,b,2
r8,0,2,8
r3,2,a,3
r1,1,a,1
e,1,0,1
j,2,b,x*ir8
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **a** in the first and **b** in the second — the pair of terminals you are looking into.

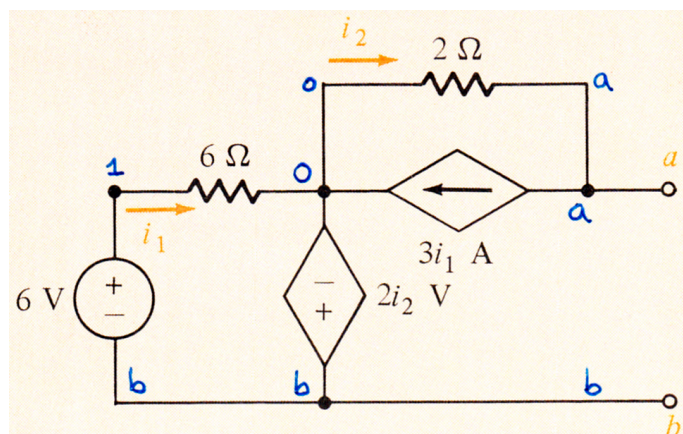
Now it solves. The expression for i_{no} is fine, but the one for r_{eq} , $(9x - 35)/(4(x - 3))$, will divide by zero at $x = 3$ — the very value we replaced.

The equivalent is already on screen, written in terms of x , so there is nothing to run again. Put i_{no} in the **Evaluate** card with $x = 3$ in its **Conditions** box: $I_{NO} = 1$ A. Ask for r_{eq} the same way and the answer is infinite, the denominator being zero at that value.

This means the equivalent resistance is, for practical purposes, infinite. Your idea of fun, right?

Bo2's Drill Exercise 3.13 (Tricky)

Another tricky one. Find the Norton equivalent of the circuit.



As in the previous example, we get an error if we simulate using **2** and **3** in the dependent sources. So, we use **2x** and **3x** instead, where **x** will be later defined as 1.

```
ei,1,b,6
ed,b,0,2*x*ir2
r1,1,0,6
r2,0,a,2
jd,a,0,3*x*ir1
```

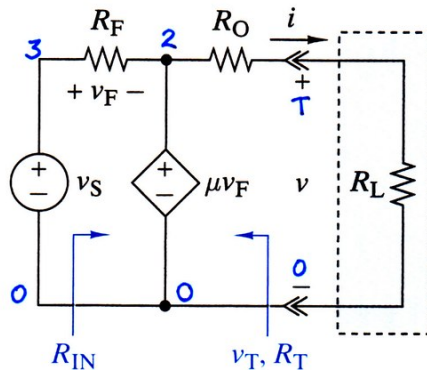
Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **a** in the first and **b** in the second — the pair of terminals you are looking into.

Exploring the answers, we see that the denominator of the expression for r_{eq} , $(x-1)$, results in a division by zero at $x = 1$.

Put i_{no} in the **Evaluate** card with $x = 1$ in its **Conditions** box: $I_{NO} = -3$ A. Ask for r_{eq} the same way and the answer is infinite, so R_{EQ} is undefined at that value.

TR5's Exercise 4-6 (Symbolic)

Find the input resistance and output Thévenin equivalent circuit of the circuit



$$R_{IN} = (1 + \mu)R_F$$

$$v_T = \frac{\mu}{\mu + 1} v_S$$

$$R_T = R_O$$

First let's find the input resistance, R_{IN} , i.e. the resistance as seen by the v_S source. We use μ for the constant in the dependent source.

```
ei,3,0,vs
rf,3,2,rf
ro,2,t,ro
ed,2,0,μ*vrf
```

The answer you want is r_{ei} , in **Results**.

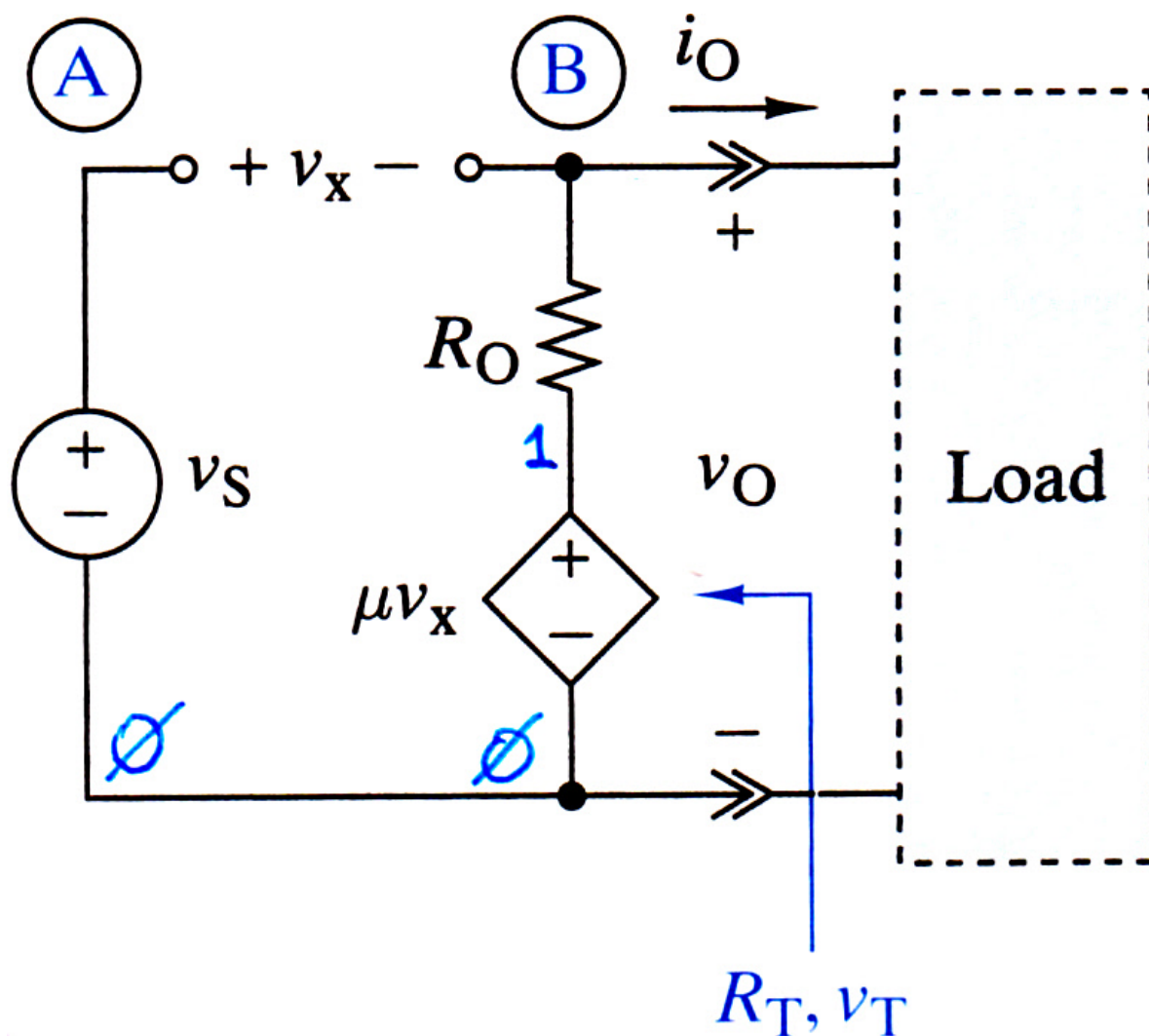
We get $rf*(\mu+1)$, which is correct. The textbook's answers are shown right of the circuit schematic. Now we find the output Thévenin equivalent circuit as seen by R_L .

```
ei,3,0,vs
rf,3,2,rf
ro,2,t,ro
ed,2,0,μ*vrf
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **t** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are v_{th} and r_{eq} , in **Results**.

We get $\{vs*\mu/(\mu+1), ro\}$, which is correct, as can be seen in the textbook's answers for v_T and R_T , shown right of the circuit schematic above.



TR5's Example 4-8 (Symbolic)

Find the Thévenin equivalent as seen by the load.

The dependent source's value is the difference between two node voltages, so version 9 writes that difference straight into the value — $v_a - v_b$, exactly as Lesson 3 (page 39) describes. There is nothing to define beforehand.

Now I run the th script, with this circuit description:

```
ei,a,0,vs
ed,1,0,μ*(va-vb)
ro,b,1,ro
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin/Norton*. Two node boxes appear: put **b** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are v_{th} and r_{eq} , in **Results**.

The answers we get, $\{v_s \mu / (\mu + 1), r_o / (\mu + 1)\}$, are correct, as can be seen by comparing

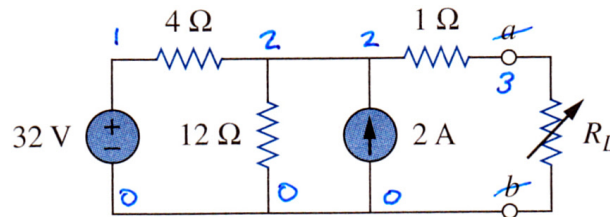
them to those in the book:

$$v_T = \frac{\mu v_s}{\mu + 1} \quad \text{and} \quad R_T = \frac{R_O}{\mu + 1}$$

I am not sure there is any other circuit simulator for calculators that can do this.

AS2's Example 4.8

Find the Thévenin equivalent of the circuit shown to the left of terminals a-b. Then find the current through $R_L = 6, 16$ and 36Ω .



My solution:

```
e,1,0,32.
r4,1,2,4
r12,2,0,12
j,0,2,2
r1,2,3,1
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **3** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are v_{th} and r_{eq} , in **Results**.

Ask **Evaluate** for:

```
vth/(req+6)
vth/(req+16)
vth/(req+36)
```

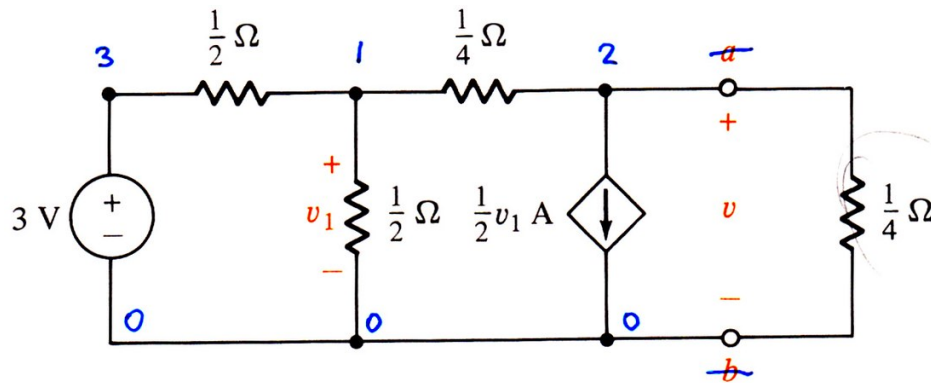
Choose DC.

The answer, **{30.,4.,3.,1.5,.75}**, is correct.

Bo2's Example 3.10

Find the Norton equivalent of the circuit left of the a-b terminals, and then find the voltage drop and the current through the $\frac{1}{4}\Omega$ resistor. My one-line solution:

```
e,3,0,3
r31,3,1,1/2
r10,1,0,1/2
r12,1,2,1/4
j,2,0,v1/2
```



Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **2** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are i_{no} and r_{eq} , in **Results**.

Ask **Evaluate** for:

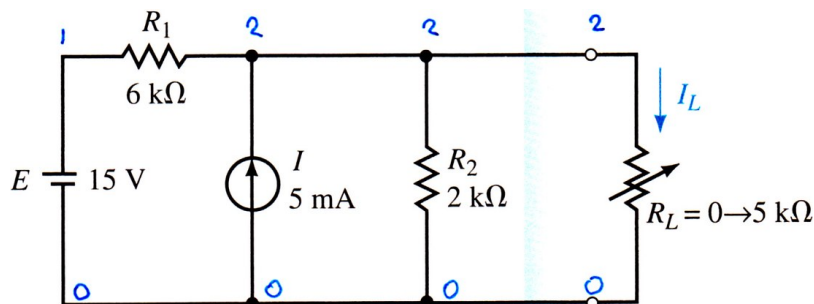
```
vth*(1/4)/(req+1/4)
vth/(req+1/4)
```

Choose DC.

The book gives the answers as fractions. We get it right: **{21/8, 4/9, 21/50, 42/25}**.

RM3's Example 9-7

Find the Norton equivalent of the circuit external to R_L . Then determine the load current I_L when $R_L = 0\ \Omega$, $2\ \text{k}\Omega$ and $5\ \text{k}\Omega$. My one-line solution:



```
e,1,0,15.
r1,1,2,6'k
j,0,2,5'm
r2,2,0,2'k
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **2** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are i_{no} and r_{eq} , in **Results**.

Ask **Evaluate** for:

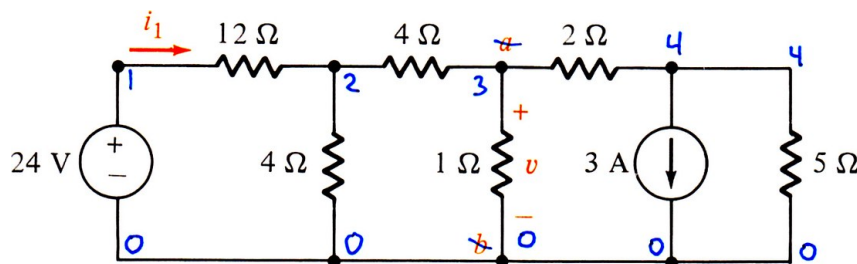

```
vth/(req+0)
vth/(req+2000)
vth/(req+5000)
```

Choose DC.

The answer, **{.0075, 1500., .0075, .00321, .00173}**, is correct.

Bo2's Example 3.5

Find the Norton equivalent of the circuit external to the 1Ω resistor. Then determine the voltage drop across this 1Ω resistor.



My solution:

```
e,1,0,24
r12,1,2,12
r20,2,0,4
r23,2,3,4
r34,3,4,2
j,4,0,3
r40,4,0,5
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **3** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are i_{no} and r_{eq} , in **Results**.

Ask **Evaluate** for:

```
vth*1/(req+1)
```

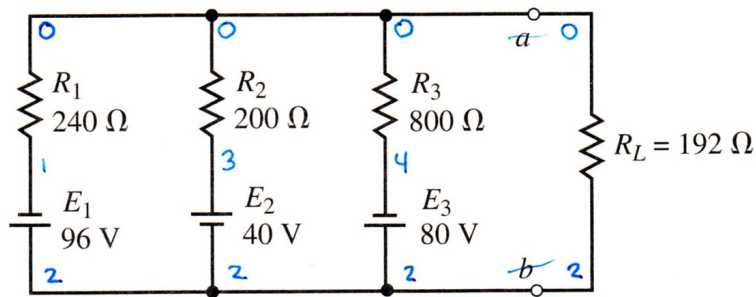
Choose DC.

The answer, **{-9/7, 7/2, -1}**, is correct.

RM3's Example 9-13

Use Millman's Theorem to simplify the circuit left of a-b so that there is only one voltage and one resistor. Then find the current in the load resistor R_L .

I don't know Millman's Theorem, but in my book this is called the Thévenin equivalent.



```
r1,0,1,240.
e1,2,1,96
r2,0,3,200
e2,3,2,40
r3,0,4,800
e3,2,4,80
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin/Norton*. Two node boxes appear: put **2** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are v_{th} and r_{eq} , in **Results**.

Ask **Evaluate** for:

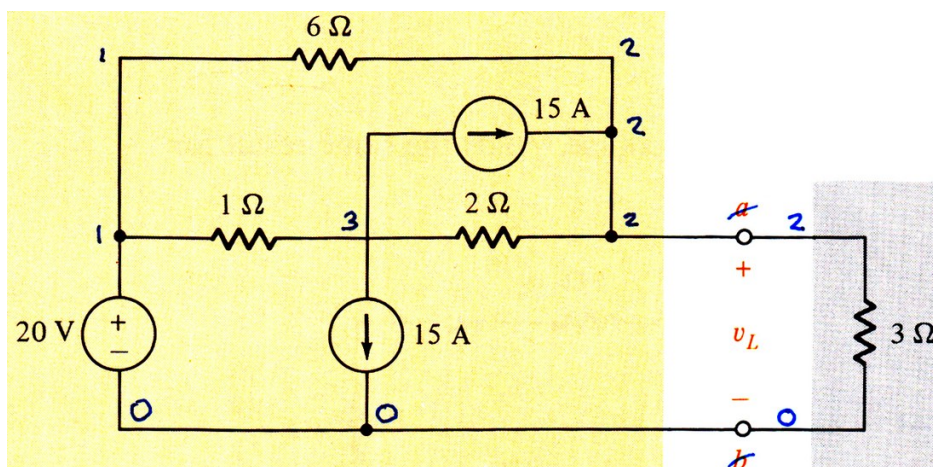
```
vth/(req+192)
```

Choose DC.

The answer, **{28.8,96,.,1}**, is correct.

Bo2's Example 3.7

Find the Thévenin equivalent for the circuit left of a-b. Then find the voltage across the $3\ \Omega$ resistor, and also if it was $6\ \Omega$. My answer:



```
e,1,0,20
r6,1,2,6
```

```

r1,1,3,1
r2,3,2,2
j32,3,2,15
j30,3,0,15

```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **2** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are v_{th} and r_{eq} , in **Results**.

Ask **Evaluate** for:

```

vth*3/(req+3)
vth*6/(req+6.)

```

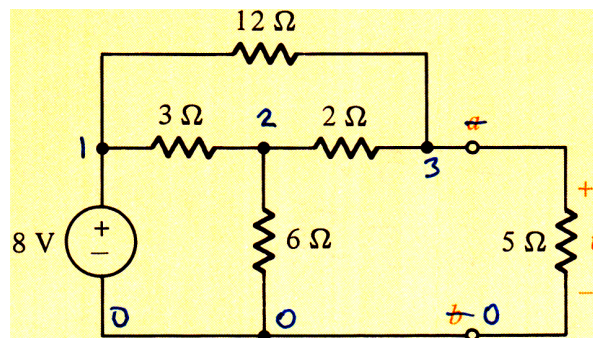
Choose DC.

The answer we find, **{30,2,18,22.5}**, is correct.

Bo2's Drill Exercise 3.7

Find the Thévenin equivalent for the circuit left of a-b. Then find the voltage v .

My answer:



```

e,1,0,8
r3,1,2,3
r12,1,3,12
r6,2,0,6
r2,2,3,2

```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **3** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are v_{th} and r_{eq} , in **Results**.

Ask **Evaluate** for:

```

vth*5/(req+5)

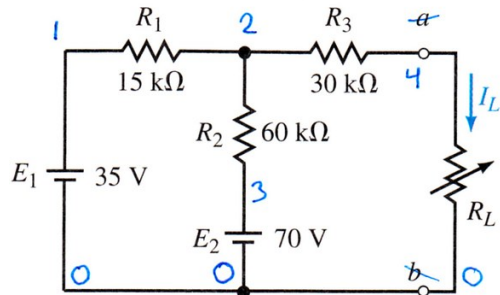
```

Choose DC.

The answer, {6,3,3.75}, is correct.

RM3's Practice Problem 9.5

Find the Norton equivalent external to R_L in the circuit. Solve for the current I_L when $R_L = 0, 10 \text{ k}\Omega, 50 \text{ k}\Omega$, and $100 \text{ k}\Omega$.



Answers

$$R_N = 42 \text{ k}\Omega, I_N = 1.00 \text{ mA}$$

$$\text{For } R_L = 0: I_L = 1.00 \text{ mA}$$

$$\text{For } R_L = 10 \text{ k}\Omega: I_L = 0.808 \text{ mA}$$

$$\text{For } R_L = 50 \text{ k}\Omega: I_L = 0.457 \text{ mA}$$

$$\text{For } R_L = 100 \text{ k}\Omega: I_L = 0.296 \text{ mA}$$

My solution:

```
e1,1,0,35.  
r1,1,2,15'k  
r2,2,3,60'k  
e2,3,0,70  
r3,2,4,30'k
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin/Norton*. Two node boxes appear: put **4** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are i_{no} and r_{eq} , in **Results**.

Ask **Evaluate** for:

```
ino*req/(req+0)  
ino*req/(req+1e4)  
ino*req/(req+5e4)  
ino*req/(req+1e5)
```

One line at a time, as before.

Choose DC.

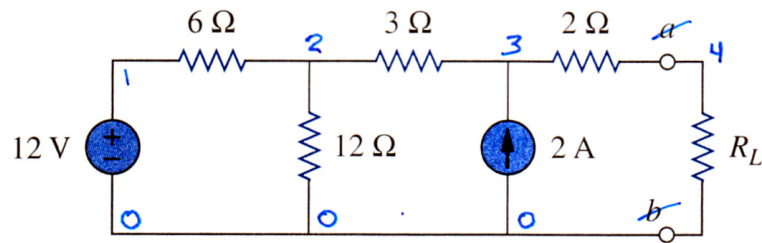
The answer, {,001,42000,,001,,000808,,000457,,000296}, is correct.

Power transfer problems

AS2's Example 4.13

Find the R_L value for maximum power transfer and the maximum power transferred.

R_L for maximum transfer is r_{eq} . The maximum power is in p_{max} . My solution:



```
e,1,0,12
r6,1,2,6
r12,2,0,12
r3,2,3,3
j,0,3,2
r2,3,4,2
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **4** in the first and **0** in the second — the pair of terminals you are looking into.

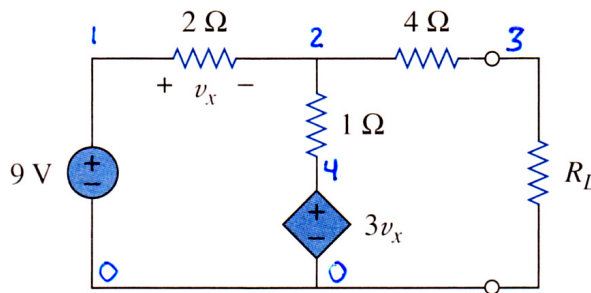
The answers you want are r_{eq} and p_{max} , in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

Choose DC. The answer, **{9.,13.44}**, is correct: the maximum transfer of power occurs when the load is 9Ω . At this point, the power transferred is 13.44W. Now let's solve another one.

AS2's Practice Problem 4.13

Find the R_L value for maximum power transfer and the maximum power transferred.



```
ei,1,0,9
rx,1,2,2
r1,2,4,1
ed,4,0,3*vr_x
r4,2,3,4
```

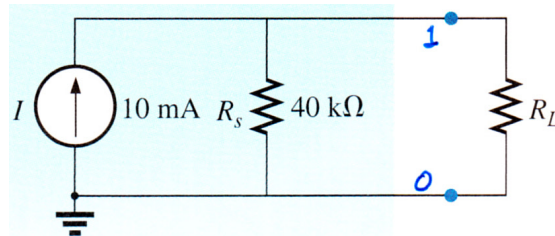
Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **3** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are r_{eq} and p_{max} , in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

Choose DC. The logic of this problem is identical to the previous one. The answer is **{4.22,2.901}**.

B11's Example 9.15



- Find the load resistance that will result in maximum power transfer to the load, and find the maximum power delivered.
- If the load were changed to $68\text{ k}\Omega$, would you expect a fairly high level of power transfer to the load based on the results of part (a)? What would the new power level be? Is your initial assumption verified?
- If the load were changed to $8.2\text{ k}\Omega$, would you expect a fairly high level of power transfer to the load based on the results of part (a)? What would the new power level be? Is your initial assumption verified?

My one-line solution to all three questions is presented below.

```
j,0,1,10'm
rs,1,0,40'k
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **1** in the first and **0** in the second — the pair of terminals you are looking into.

The answers you want are r_{eq} and p_{max} , in **Results**.

Ask **Evaluate** for:

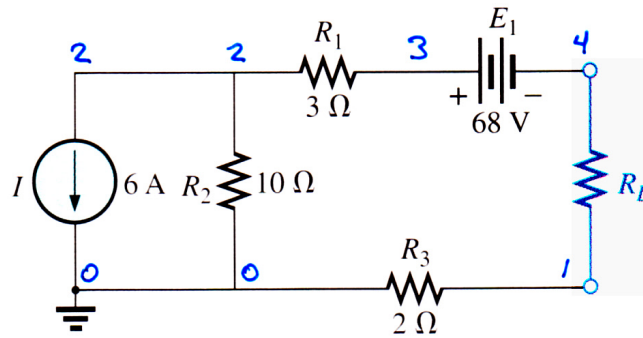
```
vth^2*68000./(req+68000.)^2
vth^2*8200./(req+8200.)^2
```

Choose DC. The answer we obtain, **{40000, 1, .93, .57}**, is correct. Let's deconstruct it.

Part (a) is answered by the first two values: a $40\text{ k}\Omega$ resistor as load would receive 1 W power. Since this is the maximum — this is the most that any load could receive ever.

Part (b) is answered by the third value. We find that a load of $68\text{ k}\Omega$ receives $.93\text{ W}$, which is less than the maximum.

Part (c) is answered in similar manner by the fourth value. A load of $8.2\text{ k}\Omega$ receives $.57\text{ W}$, which is less than the maximum. Any resistance other than $40\text{ k}\Omega$ gets less power.



B11's Example 9.17

Find the R_L value for maximum power transfer and the maximum power transferred.

```
j,2,0,6
r2,2,0,10
r1,2,3,3
r3,0,1,2
e,3,4,68
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **4** in the first and **1** in the second — the pair of terminals you are looking into.

The answers you want are r_{eq} and p_{max} , in **Results**.

The calculator versions wrap this in `approx` to get a decimal; version 9 does it through **Rounding** — *approx to n digits*, with **n** = 3 here.

Choose DC. The answer is **{15., 273.07}**. Let's now see one that is a little different.

LESSON 5

Operational amplifiers

Learn how to describe an ideal *operational amplifier* (Op Amp) using the **o** element. And practice what you learn in a variety of solved op-amp problems, both easy and hard.

In this lesson, you will learn how to describe an ideal *operational amplifier* — also called an Op Amp — using the **o** element. And you will practise what you learn in a variety of solved op-amp problems, both easy and hard.

5.1 How to describe an ideal operational amplifier

There are many types of operational amplifiers. Symbulator can simulate the ideal, linear type.

NOTE

Describing op amps

In Symbulator, an ideal op amp is described using four bits of information, separated by commas: first, the name of the op amp, which must start with the letter **o**; then, the names of the two input nodes, separated by a comma (the polarity doesn't matter, you can give these input nodes in any order); and finally, the name of the output node.

For example, an ideal op amp called o1, with input nodes 2 and 3 and output node 5, is described o1,2,3,5. And one called o, with input nodes p and n and output node o, is o,p,n,o

What answers do you get

For each ideal op amp in a circuit, Symbulator will store the following answers:

- The current through the output node, flowing from the output node outwards. For an op amp called o, that is i_o .
- The power consumed by the visible part of the op amp. For an op amp called o2, p_{o2} ; the power delivered is the negative of that.
- And, as with every element's nodes, the voltage of each of its nodes with reference to ground. For node o, v_o .

Livin' on the edge

Since I was born to be bad, I like to play with node names to get the answers as close to the book as possible. So do not be surprised if I call one op amp “o” and name its output node “o” as well. That way I can ask for v_o and i_o and get the output voltage and current. But realise this: v_o asks for the voltage in node o, while in i_o the o stands for the element o.

In Symbulator 9 the collision is impossible: **Results** lists node voltages and element quantities in separate sections, each labelled by its own node or element, so a node called o and an op amp called o never contend for the same name.

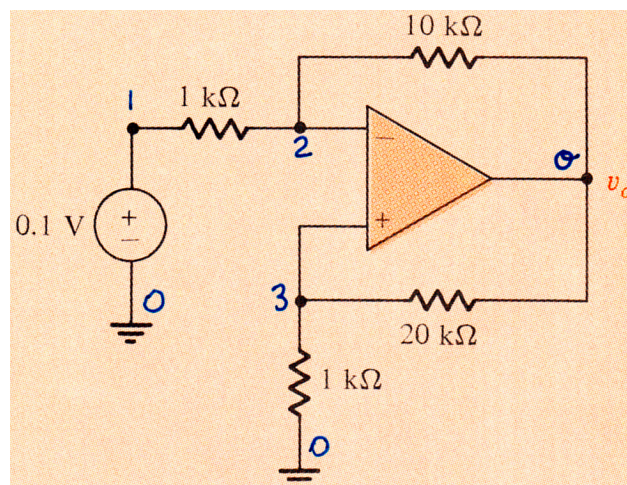
⚠ CAUTION

Never describe a source as e,#,0,v_#

If you describe a source as e1,1,0,v1, you are declaring the voltage of node 1 to be the voltage of node 1. The resulting equation, $v1 = v1$, is trivially true and carries no information, which leaves the system one equation short.

Bo2's Drill Exercise 3.2

For the op-amp circuit shown, find v_o and the power absorbed by the op amp.



Bo2's Drill Exercise 3.2

SOLUTION

My solution:

```
e,1,0,.1
r12,1,2,1'k
r2o,2,o,10'k
r3o,3,0,1'k
r3o,3,o,20'k
o,3,2,o
```

We look in the results and see v_o and p_o :

- $v_o = -2.1$ V

- $p_o = -0.00063 \text{ W}$

Which are correct. The voltage in node o is -2.1 V , and the op amp is absorbing -0.63 mW , which is to say it is delivering 0.63 mW of power to the circuit.

5.2 Instructive Op Amp examples

NOTE

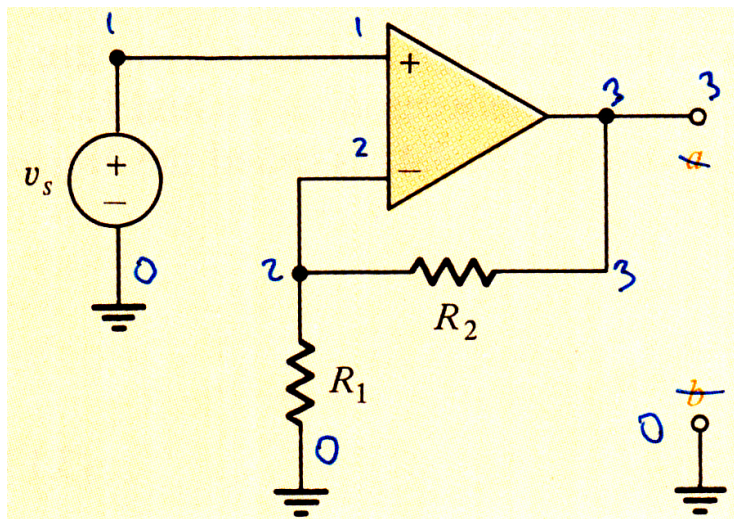
Some of this narration still describes the calculator

Every problem below carries a **Circuit Description** panel you can type straight into Symbolator 9, and the circuits and answers are identical across the versions. What has not all been rewritten is the narration between them: where it says to evaluate a name or press a key, do the version 9 equivalent — the results are already on screen.

Solved Op Amp problems

Bo2's Drill Exercise 3.11 (Thévenin)

Find the Thévenin equivalent.



```
e,1,0,vs
r1,2,0,r1
r2,2,3,r2
o,1,2,3
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin / Norton*. Two node boxes appear: put **3** in the first and **0** in the second — the pair of terminals you are looking into.

This is the one that catches people out, and Symbolator says so plainly:

- $v_{th} = \frac{v_s (r_1 + r_2)}{r_1}$
- $i_{no} = \infty$
- $r_{eq} = 0$
- $p_{max} = \infty$

with a note beneath the answers explaining that the short-circuit current is unbounded, so the equivalent is a voltage source with no impedance in series with it.

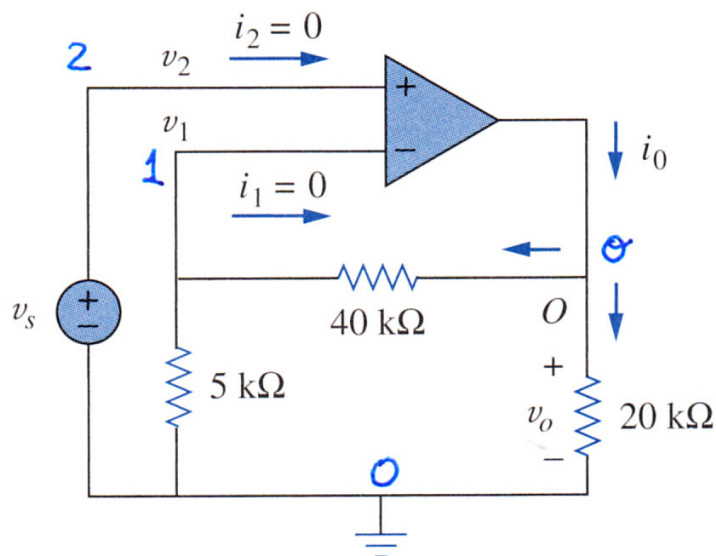
None of that is a failure to solve, and none of it is guesswork. An ideal op amp holds its output voltage whatever current is drawn from it, so asking what flows through a short across that output has no answer as an equation — the two rounds Symbolator runs to build a Thévenin equivalent leave the second one unsolvable. What it does instead is put a resistance across the terminals rather than a short, and let that resistance fall to zero, which is what a short is. The current grows without bound, and an unbounded current through a short is precisely a source with nothing in the way of it: $R_{EQ} = 0\Omega$.

So V_{TH} is correct, as can be seen by comparing it to the book's answer — and the Thévenin resistance is not merely asserted here, it is the answer Symbolator gives.

$$\left(1 + \frac{R_2}{R_1}\right) v_s$$

AS2's Example 5.2

Find v_o and i_o .



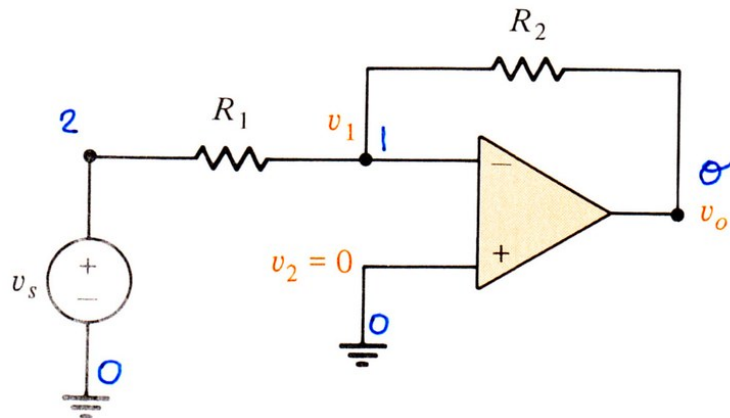
```
e,2,0,1.
r5,1,0,5'k
r4,1,0,40'k
r2,0,0,20'k
o,2,1,0
```

The answers you want are v_o and i_o , in **Results**.

The answer, {9,.,00065}, is correct: v_o is 9V and i_o is 0.65mA

Bo2's Figure 3.3 (Inverting)

Find the gain of the overall circuit, v_o/v_s .



$$\frac{v_o}{v_s} = -\frac{R_2}{R_1}$$

```
e,2,0,vs
r1,2,1,r1
r2,1,o,r2
o,1,0,o
```

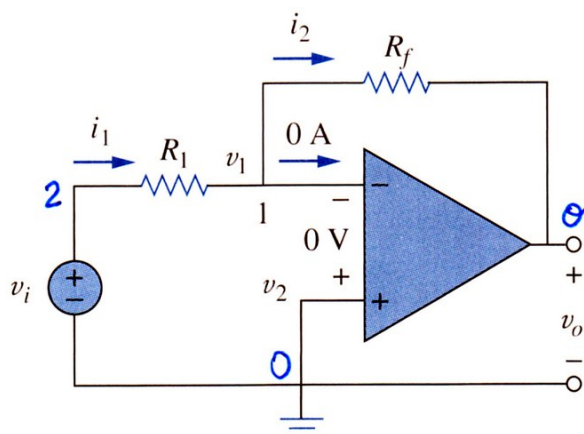
Ask **Evaluate** for:

```
vo/vs
```

We get **-r2/r1**, which is correct, as can be seen in the book's answer above.

AS2's Figure 5.10 (Inverting)

Find v_o .



$$v_o = -\frac{R_f}{R_1} v_i$$

```
e,2,0,vi
r1,2,1,r1
rf,1,o,rf
o,0,1,o
```

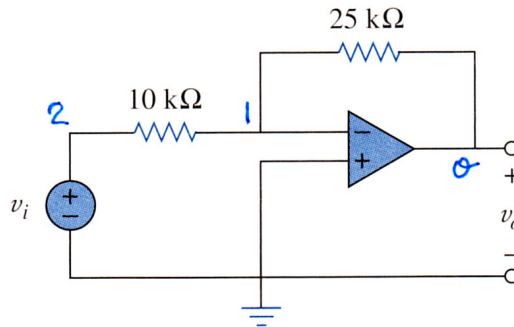
The answer you want is v_o , in **Results**.

This problem is almost identical to the one above. The answer we get is correct:

$$-(r_f/r_1)v_i$$

AS2's Example 5.3 (Inverting)

If v_i is 0.5V, calculate the output voltage v_o and the current in the 10 k Ω resistor.



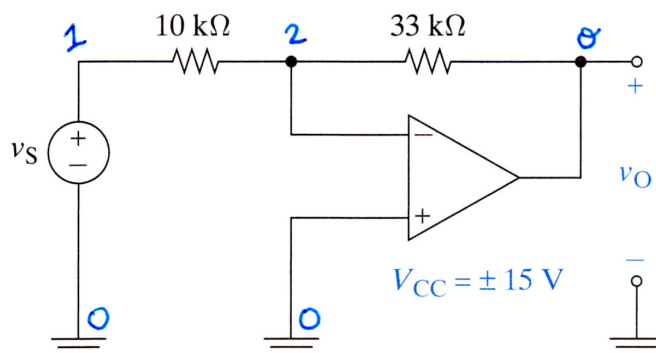
```
e,2,0,.5
r1,2,1,10'k
rf,1,o,25'k
o,0,1,o
```

The answers you want are v_o and i_{r1} , in **Results**.

The answer, **{-1.25, 5e-5}**, is correct.

TR5's Exercise 4-11 (Inverting)

Find v_o when v_s is 2V, -4V and 6V. Notice the output of the op amp is limited to ± 15 V.



This may be the only non-linear problem you will see in this book, because I solved it before I realized it included the ± 15 V constraint. But anyway, here it goes.

```
e,1,0,vs
r1,1,2,10'k
r2,2,o,33'k
o,0,2,o
```

The calculator's | says "given"; version 9 says the same thing in the **Evaluate** card's own

Conditions box. Put the answer you want in the first and the value you are giving it in the second:

v_o

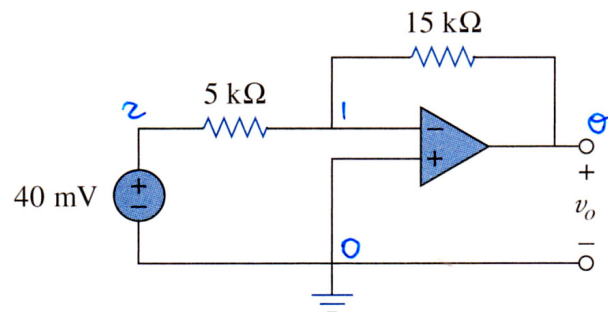
$v_s = 2$

Then run it again with $v_s = -4$ and once more with $v_s = 6$.

The answer, **{-6.6,13.2,-19.8}**, is correct within the linear realm, but since the output is constrained to no more than 15V or less than -15V, the answer is $v_o = -15V$ for $v_s = 6V$.

AS2's Practice Problem 5.3 (Inverting)

Find the output voltage of the op amp (i.e. v_o) and calculate the current through the feedback resistor (i.e. the 15 k Ω resistor).



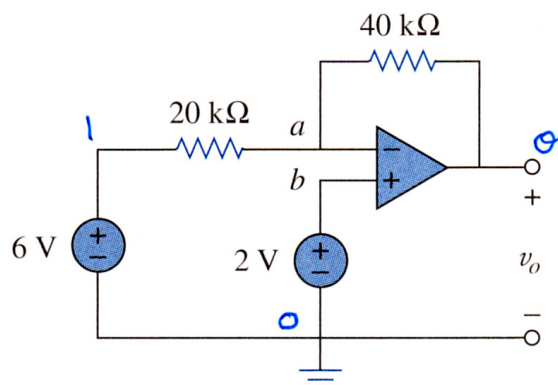
My answer: Notice we used the m for milli in the value of the voltage source.

```
e,2,0,40'm
r1,2,1,5'k
rf,1,0,15'k
o,0,1,o
```

The answer, **{-.12, 8e-6}**, is correct.

AS2's Example 5.4 (Inverting)

Determine v_o .



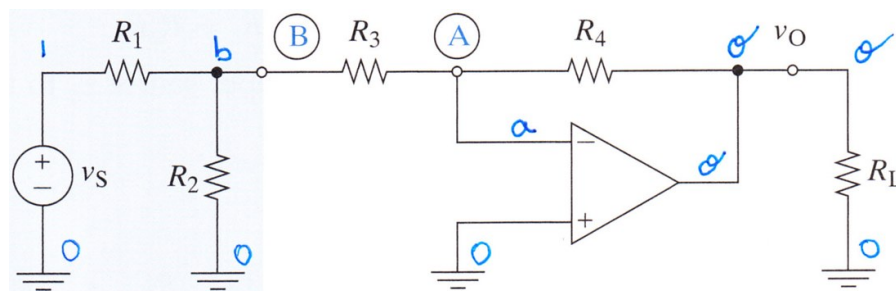
```
e6,1,0,6
r1,1,a,20'k
e2,b,0,2
rf,a,o,40'k
o,a,b,o
```

The answer you want is v_o , in **Results**.

The answer, **-6**, is correct.

TR5's Example 4-14 (Inverting)

Find the input-output relationship of the circuit.



The 'formal' way to *symbolate* this circuit would be as described below.

```
e,1,0,vs
r1,1,b,r1
r2,b,0,r2
r3,b,a,r3
r4,a,o,r4
o,0,a,o
r1,o,0,r1
```

However, since we are only interested in the ratio of the input to the output, a smarter way to *symbolate* it is this:

```
e,1,0,1
r1,1,b,r1
r2,b,0,r2
r3,b,a,r3
r4,a,o,r4
o,0,a,o
r1,o,0,1
```

When we evaluate v_o/v_1 , both approaches get the same answer:

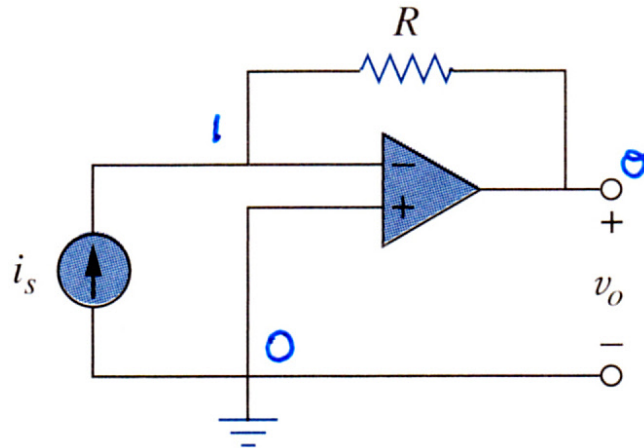
$$-\frac{r_2 r_4}{r_1 r_2 + r_1 r_3 + r_2 r_3}$$

which is correct, as can be seen by comparing it to the book's answer.

$$-\frac{R_2 R_4}{R_1 R_2 + R_1 R_3 + R_2 R_3}$$

AS2's Practice Problem 5.4a (Transresistance)

This is a current-to-voltage converter, also called a *transresistance amplifier*. Find v_o/i_s .



```
j,0,1,is1
r,1,0,r
o,0,1,o
```

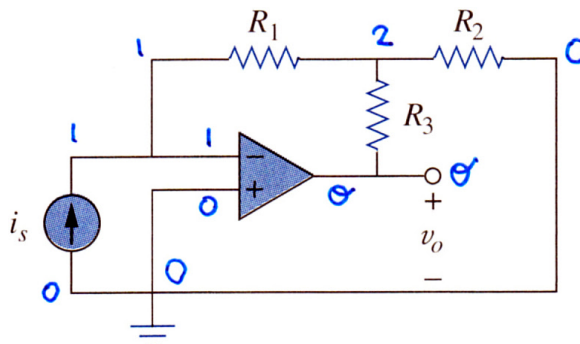
Ask **Evaluate** for:

```
vo/is1
```

The answer, **-r**, is correct.

AS2's Practice Problem 5.4b (Transresistance)

This is another *transresistance amplifier*. Again, find v_o/i_s .



```
j,0,1,is1
r1,1,2,r1
r2,2,0,r2
r3,2,0,r3
```

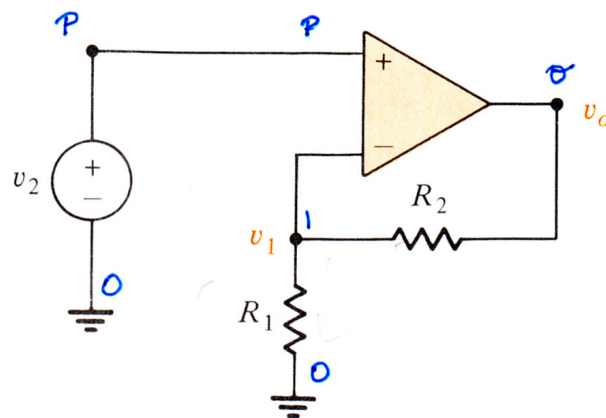
o,0,1,o

Ask **Evaluate** for v_o/i_s1 . It gives $-r1 - r1*r3/r2 - r3$, which is equivalent to the book's answer — the same expression, gathered differently.

$$\frac{v_o}{i_s} = -R_1 \left(1 + \frac{R_3}{R_1} + \frac{R_3}{R_2} \right)$$

Bo2's Example 3.1 (Non-Inverting Amplifier)

Find v_o/v_1 .



e,p,0,v2
r1,1,0,r1
r2,1,o,r2
o,p,1,o

Ask **Evaluate** for:

v_o/v_1

We get $1+r2/r1$, which is correct.

AS2's Figure 5.16 (Non-Inverting Amplifier)

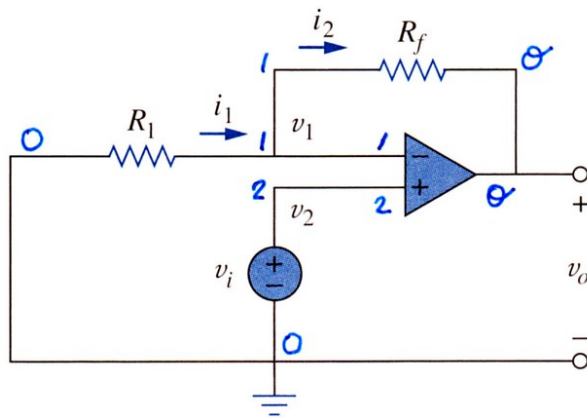
Find v_o .

e,2,0,vi
r1,0,1,r1
rf,1,o,rf
o,2,1,o

The answer you want is v_o , in **Results**.

We get the right answer (below), an expression equivalent to the book's answer.

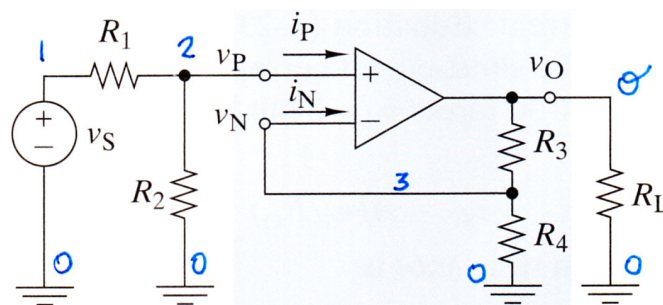
$$\left(\frac{r1 + rf}{r1} \right) vi$$



$$v_o = \left(1 + \frac{R_f}{R_1}\right) v_i$$

TR5's Example 4-13 (Non-Inverting Amplifier)

Find V_o/V_s .



We can solve this problem in one simulation, as shown below.

```
e,1,0,vs
r1,1,2,r1
r2,2,0,r2
o,2,3,o
r3,o,3,r3
r4,3,0,r4
```

Ask **Evaluate** for:

```
vo/vs
```

We get

$$\frac{r2(r3 + r4)}{(r1 + r2)r4}$$

which is correct, as can be seen by comparing it to the book's answer.

$$\left[\frac{R_2}{R_1 + R_2} \right] \left[\frac{R_3 + R_4}{R_4} \right]$$

We can also solve it in stages, as shown below. First, simulate the left half.

e,1,0,vs
r1,1,2,r1
r2,2,0,r2

Ask **Evaluate** for:

v2/vs

We get

$$\frac{r2}{r1 + r2}$$

which is correct for this part. Then, simulate the right half.

e,2,0,1
o,2,3,o
r3,o,3,r3
r4,3,0,r4

Ask **Evaluate** for:

vo/v2

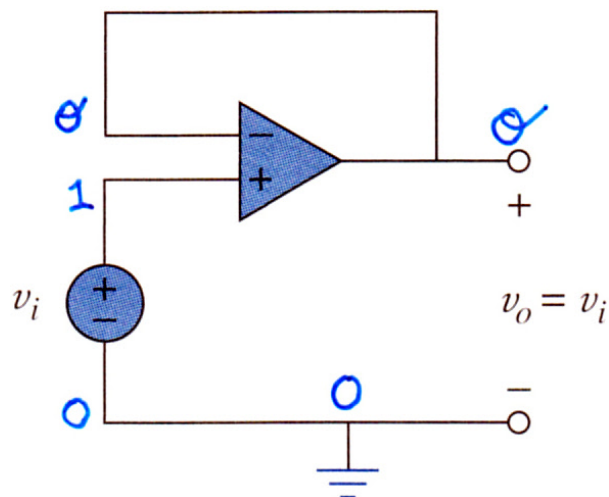
We get

$$\frac{r3 + r4}{r4}$$

which is correct for this part. The product of these two partial answers produces the same expression shown above after the big simulation, and is the right answer.

AS2's Figure 5.17 (Voltage Follower)

Find v_o .



```
e,1,0,vi  
o,1,o,o
```

The answer you want is v_o , in **Results**.

The answer, **vi**, is correct.

TR5's Figure 4-32 (Voltage Follower)

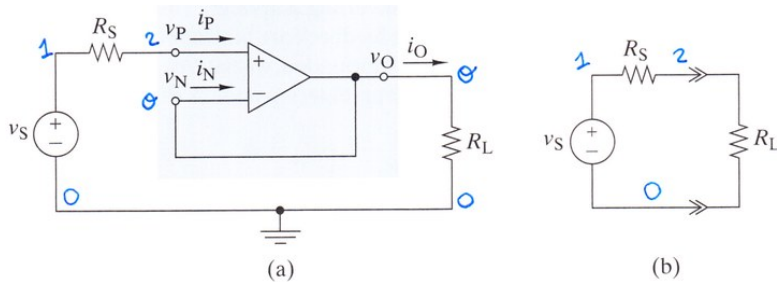


FIGURE 4-32 (a) Source-load interface with a voltage follower. (b) Interface without the voltage follower.

The circuits in Figure 4-32 have $v_S = 1.5$ V, $R_S = 2$ k Ω , and $R_L = 1$ k Ω . Compute the maximum power available from the source. Compute the power absorbed by the load resistor in the direct connection in Figure 4-32(b) and in the voltage follower circuit in Figure 4-32(a). Discuss any differences.

Answers: 281 μ W; 250 μ W; 2250 μ W

Alright, so first we simulate the (b) circuit, to find its maximum power and load power:

```
e,1,0,1.5  
rs,1,2,2'k
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Thévenin/Norton*. Two node boxes appear: put **2** in the first and **0** in the second — the pair of terminals you are looking into.

The answer you want is p_{max} , in **Results**.

The power in a 1 k Ω load is the expression Lesson 4 (page 71) derives, with 1000 where R goes:

```
vth^2*1000/(req+1000)^2
```

The answers we get, **{2.8125e-4,2.5e-4}**, are correct. Now we simulate the (a) circuit.

```
e,1,0,1.5  
rs,1,2,2'k  
o,2,o,o  
rl,o,0,1'k
```

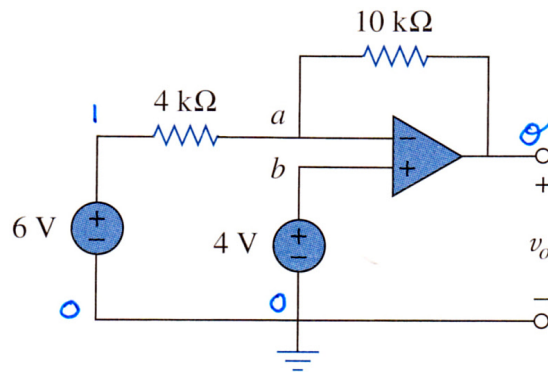
The answer you want is p_{rl} , in **Results** — the power consumed by the load resistor r_l .

The answer, **.00225**, is correct. The apparent paradox — the load in (a) drawing more power than the source in (b) seems able to provide — evaporates once we remember

that the ideal op amp in the schematic is only part of the truth: the real one has its own source of power, which makes up the difference.

AS2's Example 5.5 (Inverting)

Find v_o .



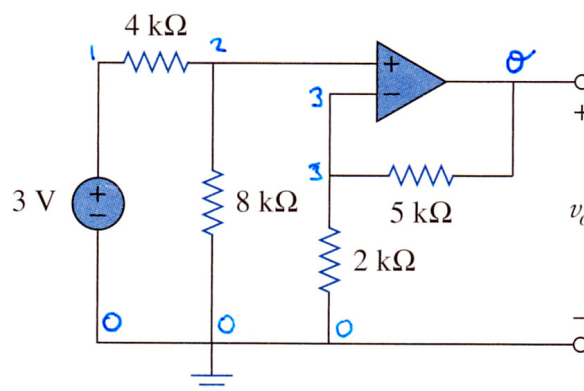
```
e1,1,0,6
r4,1,a,4'k
r10,a,o,10'k
e4,b,0,4
o,b,a,o
```

The answer you want is v_o , in **Results**.

The answer, **-1**, is correct.

AS2's Practice Problem 5.5 (Non-Inverting)

Calculate v_o .

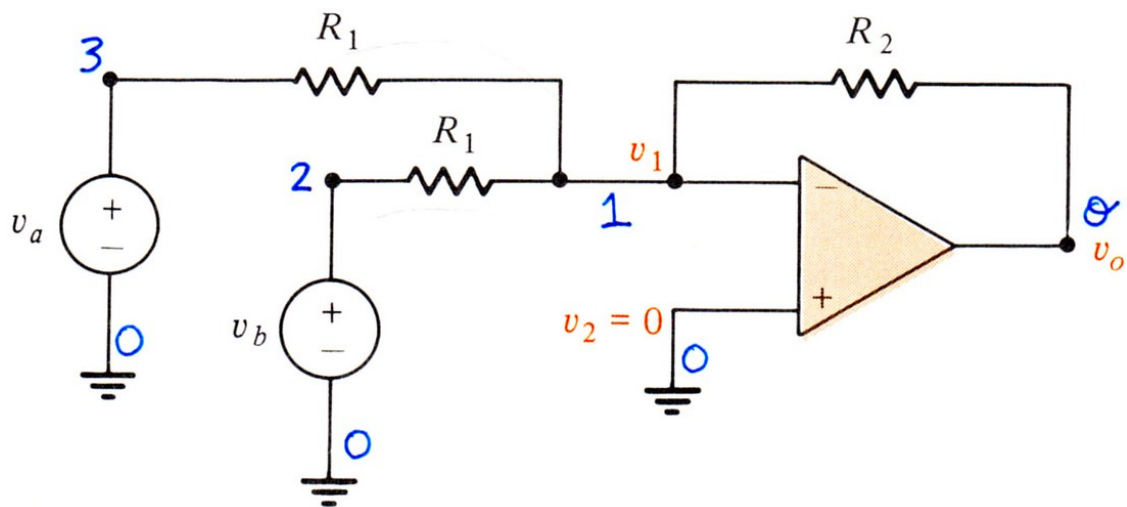


```
e,1,0,3
r4,1,2,4'k
r8,2,0,8'k
r2,3,0,2'k
r5,3,o,5'k
o,2,3,o
```

The answer for v_o , **7**, is correct.

Bo2's Example 3.2 (Adder or Summing)

Find v_o .



```
ea,3,0,va
eb,2,0,vb
r31,3,1,r1
r21,2,1,r1
r1o,1,o,r2
o,0,1,o
```

The answer you want is v_o , in **Results**.

$-(r_2/r_1)(v_a + v_b)$

which is correct, as can be seen by comparing it to the book's answer.

$$v_o = -\frac{R_2}{R_1} (v_a + v_b)$$

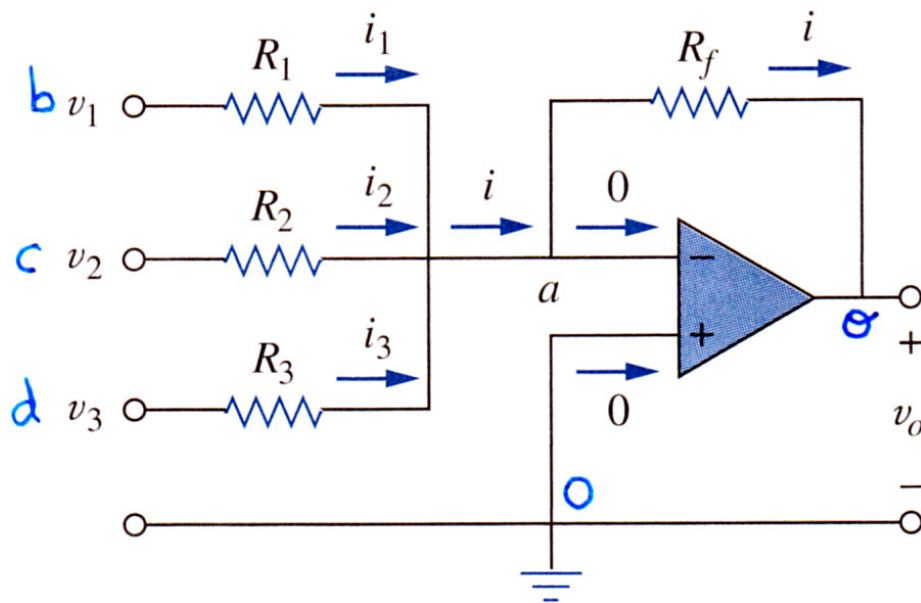
AS2's Figure 5.21 (Adder or Summing)

Find v_o .

```
e1,b,0,v1
e2,c,0,v2
e3,d,0,v3
r1,b,a,r1
r2,c,a,r2
r3,d,a,r3
rf,a,o,rf
o,0,a,o
```

Ask **Evaluate** for `expand(vo)`, exactly as on the calculator.

$-r_f v_1/r_1 - r_f v_2/r_2 - r_f v_3/r_3$

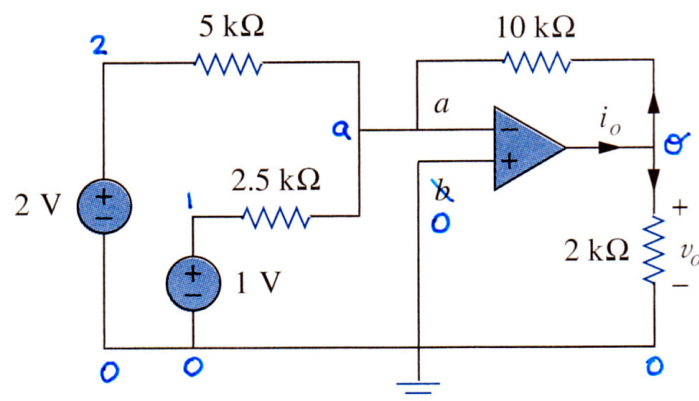


which is correct, as can be seen by comparing it to the book's answer.

$$v_o = - \left(\frac{R_f}{R_1} v_1 + \frac{R_f}{R_2} v_2 + \frac{R_f}{R_3} v_3 \right)$$

AS2's Example 5.6 (Adder or Summing)

Find v_o and i_o .



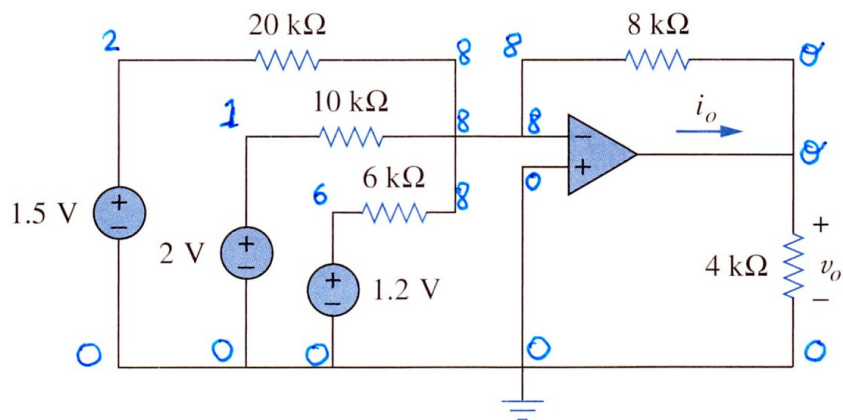
```
e1,1,0,1
e2,2,0,2
r1,2,a,5'k
r2,1,a,2.5'k
r3,a,o,10'k
r4,o,0,2'k
o,0,a,o
```

The answers you want are v_o and i_o , in **Results**.

The answer, **{-8.,-.0048}**, is correct. Notice a current of 4.8mA is going into the op amp.

AS2's Practice Problem 5.6 (Adder or Summing)

Find v_o and i_o .

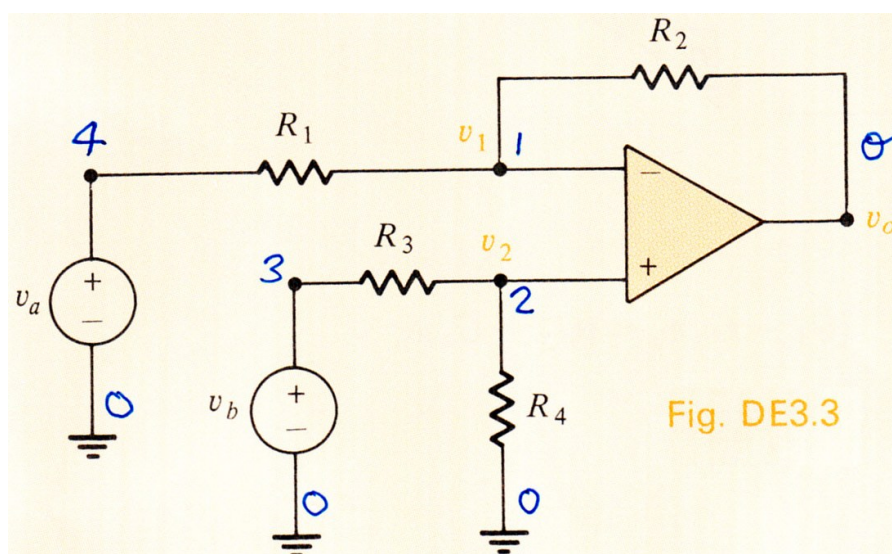


```
e2,2,0,1.5
e1,1,0,2
e6,6,0,1.2
r2,2,8,20'k
r1,1,8,10'k
r6,6,8,6'k
r8,8,0,8'k
r4,0,0,4'k
o,0,8,0
```

The answers you want are v_o and i_o , in **Results**.

The answer, **{-3.8,-.001425}**, is correct. Again, the current is going into the op amp.

Bo2's Drill Exercise 3.3 (Difference or Differential)



Find v_o for the **difference-amplifier** circuit shown in Fig. DE3.3 when $R_3 = R_1$ and $R_4 = R_2$.

Answer: $\frac{R_2}{R_1} (v_b - v_a)$

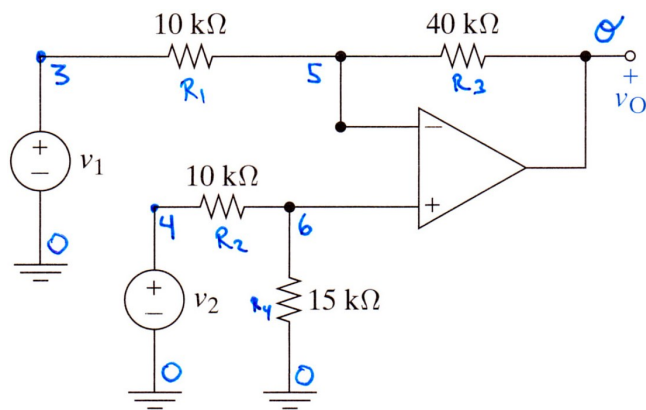
ea,4,0,va
 eb,3,0,vb
 r1,4,1,r1
 r2,1,0,r2
 r3,3,2,r1
 r4,2,0,r2
 o,2,1,o

v_o is the correct answer, equivalent to the book's answer above.

$((v_b - v_a) r_2) / r_1$

TR5's Exercise 4-13 (Difference or Differential)

Find v_o .



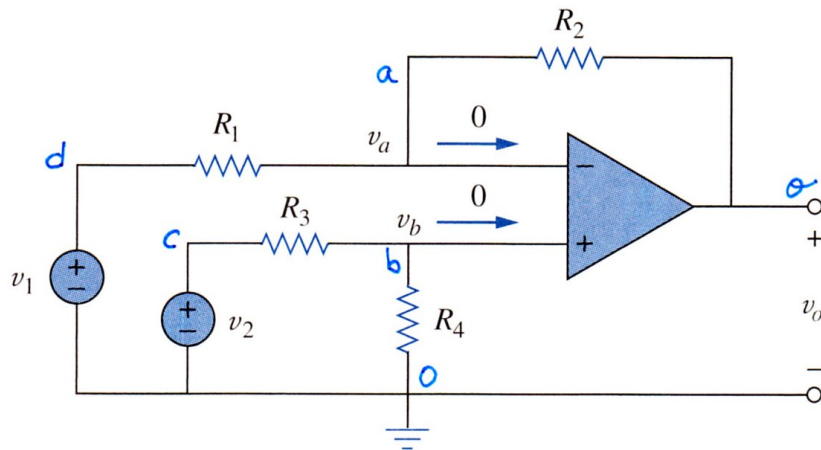
e1,3,0,v1
 e2,4,0,v2
 r1,3,5,10'k
 r2,4,6,10'k
 r3,5,0,40'k
 r4,6,0,15'k
 o,6,5,o

v_o is $3 v_2 - 4 v_1$, which is the correct answer.

AS2's Figure 5.24 (Difference or Differential)

Find v_o . (And keep it in the memory, for you will use it in the next three problems.)

e1,d,0,v1
 e2,c,0,v2
 r1,d,a,r1
 r3,c,b,r3



```
r2,a,o,r2
r4,b,o,r4
o,b,a,o
```

The answer you want is v_o , in **Results**.

$$\frac{r1 r4 v2 - r2 (r3 v1 + r4 (v1 - v2))}{r1 (r3 + r4)}$$

This expression is equivalent to the book's answer.

$$v_o = \frac{R_2 (1 + R_1/R_2)}{R_1 (1 + R_3/R_4)} v_2 - \frac{R_2}{R_1} v_1$$

AS2's Figure 5.24 (Subtractor)

For the same circuit of the previous problem, find v_o when $R_1=R_2$ and $R_3=R_4$.

Since we already have the expression for v_o stored in the memory, we only do this:

The calculator's **|** is Evaluate's **Conditions** box. Ask **Evaluate** for `expand(vo)` and give it both equalities:

```
r2 = r1
r3 = r4
```

The answer we get, **v2-v1**, is correct.

AS2's Example 5.7 (Difference or Differential)

Design an op amp circuit with inputs v_1 and v_2 such that $v_o = -5v_1 + 3v_2$.

My solution follows. The problem statement is a fancy way of saying: for the circuit of the previous problem, find the resistor values that give an output $v_o = -5v_1 + 3v_2$. Not strictly a Symbolator problem, but it shows how Symbolator fits into design problems.

First we take that part of v_o that is a factor of v_1 , and make it equal to -5. Thus:

Conditions again, this time on the inputs rather than the resistors: $v_1 = 1$ and $v_2 = 0$. Then v_o is

$$-\frac{r_2}{r_1}$$

which is equivalent to the book's expression, and it is that which has to equal -5.

Version 9 will not always arrange an expression the way the book does. If you want it another way, the **Evaluate** card takes `simplify()`, `collect()`, `expand()`, `factor()` and `apart()`.

We get **-r2/r1=-5**. Now make that part of v_o that is a factor of v_2 equal to 3. Thus:

With $v_1 = 0$ and $v_2 = 1$ instead, v_o is

$$\frac{r_4(r_1 + r_2)}{r_1(r_3 + r_4)}$$

which is equivalent to the book's expression, and it is that which has to equal 3.

We get **r2*r4/(r1*(r3+r4))+r4/(r3+r4)=3** Now, since you have two equations, you can solve for two unknowns. Of the four resistors you get to choose, two can be whatever you want. The book recommends $R_1 = 10 \text{ k}\Omega$ and $R_3 = 20 \text{ k}\Omega$. Now let's find R_2 and R_4 .

There is no `ans(1)`, so write the two equations out. This is what the **Solve** card is for – it solves a system that is not a circuit:

```
-r2/r1 = -5  
r4*(r1 + r2)/(r1*(r3 + r4)) = 3
```

with r_2 , r_4 as the unknowns. The answer is $r_2 = 5r_1$ and $r_4 = r_3$: a design rule rather than a pair of numbers. Put the book's $r_1 = 10 \text{ k}\Omega$ and $r_3 = 20 \text{ k}\Omega$ into it and you get $50 \text{ k}\Omega$ and $20 \text{ k}\Omega$.

We get **r2=50000 and r4=20000**. This is correct.

Now, if this problem was part of a test, I'd like to verify that the answer is correct. To confirm this, simulate the circuit using the four values given above for the resistors.

```
e1,d,0,v1  
e2,c,0,v2  
r1,d,a,10'k  
r3,c,b,20'k  
r2,a,o,50'k
```

```
r4,b,0,20'k
o,b,a,o
```

v_o is the desired output, $3*v_2 - 5*v_1$. The resistor values are correct.

AS2's Practice Problem 5.7 (Difference or Differential)

Design a difference amplifier with gain 4.

My solution follows. Again the statement is a fancy way of saying: for the circuit of the previous problem, find the resistor values that give an output $v_o = 4(v_2 - v_1)$, that is, $-4v_1 + 4v_2$. Same as before.

Exactly as in the previous problem: solve the circuit twice with conditions on the inputs — $v_1 = 1, v_2 = 0$, then $v_1 = 0, v_2 = 1$ — and read v_o each time. The two factors are the same as before, so the two design equations are the same shape with 4 in place of 5 and 3.

This time the book asks that you use $R_1 = 10 \text{ k}\Omega$ and $R_3 = 10 \text{ k}\Omega$. So we do that.

In the **Solve** card:

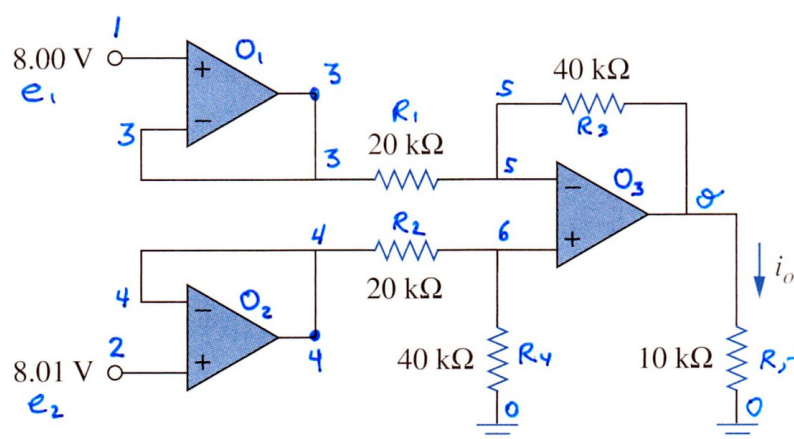
```
-r2/r1 = -4
r4*(r1 + r2)/(r1*(r3 + r4)) = 4
```

with r_2, r_4 as the unknowns. It answers $r_2 = 4r_1$ and $r_4 = 4r_3$ — at $10 \text{ k}\Omega$ each, **40000** Ω and **40000** Ω .

We get **$r_2=40000$ and $r_4=40000$** , the correct values for the remaining resistors.

AS2's Practice Problem 5.8 (Instrumentation)

Find i_o .



```
e1,1,0,8.
e2,2,0,8.01
o1,1,3,3
o2,2,4,4
r1,3,5,20'k
```

```

r2,4,6,20'k
r3,5,0,40'k
r4,6,0,40'k
o3,6,5,o
r5,0,0,10'k

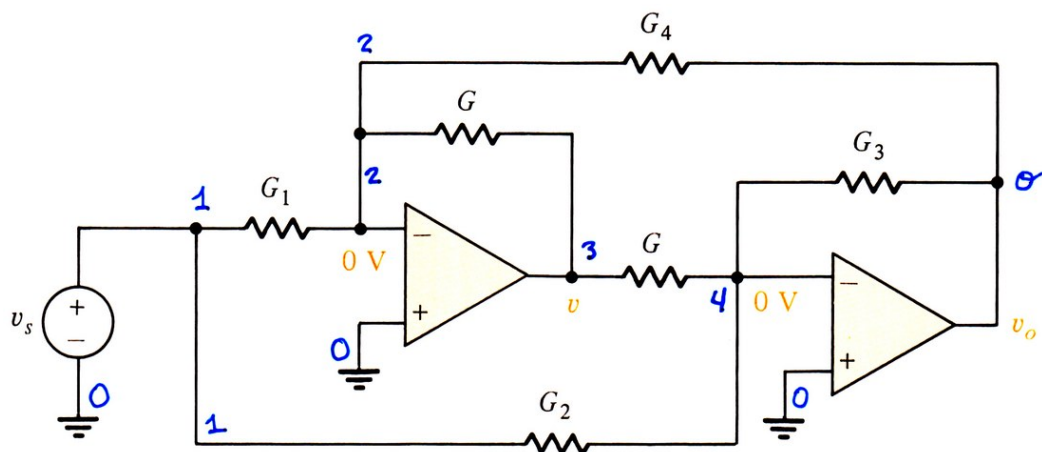
```

The answer you want is `ir5`, in **Results**.

In the schematic, the current i_o corresponds to `ir5`. The answer, **2e-6**, is correct.

Bo2's Example 3.3 (Cascade)

Find v_o in terms of the conductances and the applied voltage v_s .



```

e,1,0,vs
r12,1,2,1/g1
r14,1,4,1/g2
r4o,4,o,1/g3
r2o,2,o,1/g4
r23,2,3,1/g
r34,3,4,1/g
o1,0,2,3
o2,0,4,o

```

v_o is:

$$((g_1 - g_2) v_s) / (g_3 - g_4)$$

which is correct, as can be seen by comparing it to the book's answer.

$$v_o = \frac{G_1 - G_2}{G_3 - G_4} v_s$$

Bo2's Drill Exercise 3.4 (Cascade)

For the op-amp circuit shown in Fig. DE3.4, suppose that $G_1 = 1$ S, $G_2 = 2$ S, $G_3 = 3$ S, $G_4 = 4$ S, and $G_5 = 5$ S. Find v_o in terms of v_s . Answer: $-0.75v_s$

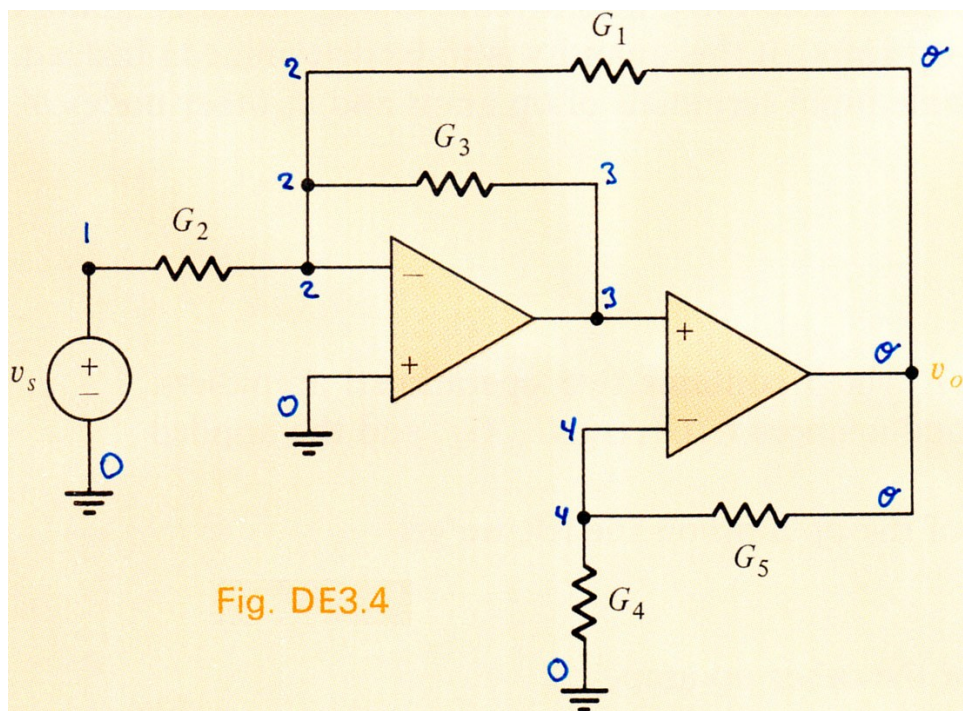


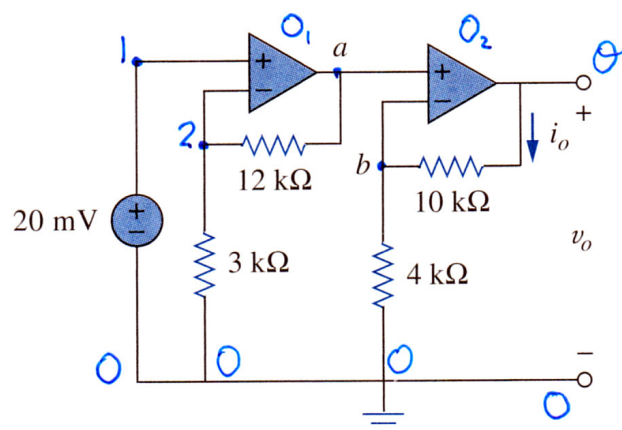
Fig. DE3.4

```
e,1,0,vs
r1,2,o,1/1
r2,1,2,1/2
r3,2,3,1/3
r4,4,0,1/4
r5,4,o,1/5
o1,0,2,3
o2,3,4,o
```

v_o is: **-.75 vs**, which is correct.

AS2's Example 5.9 (Cascade)

Find v_o and i_o .



```
e,1,0,20'm
o1,1,2,a
o2,a,b,o
```

```

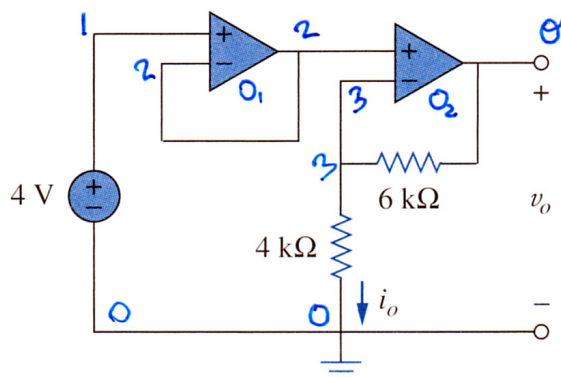
ro,o,b,10'k
r4,b,0,4'k
r2,a,2,12'k
r3,2,0,3'k

```

v_o is **.35** and i_o is **2.5e-5**. This is correct.

AS2's Practice Problem 5.9 (Cascade)

Determine v_o and i_o .



```

e,1,0,4
o1,1,2,2
o2,2,3,o
ro,3,0,4'k
r6,3,o,6'k

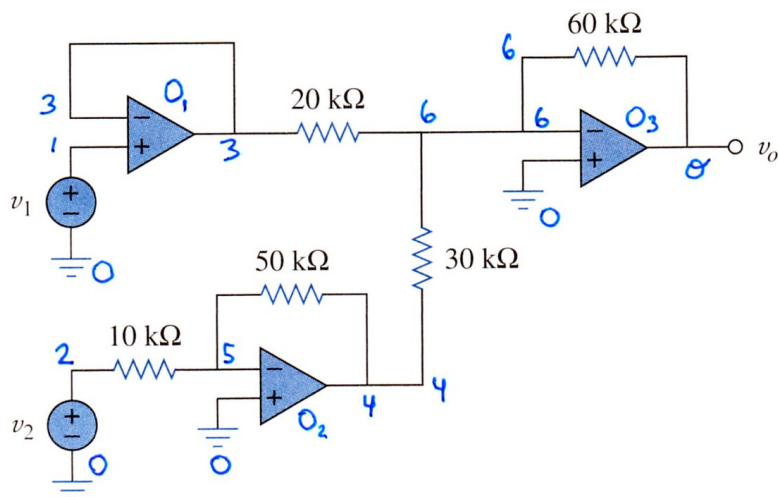
```

The answers you want are v_o and i_o , in **Results**.

The answer, **{10,1/1000}**, is correct. To say a 1/1000 A current is the same as 1mA.

AS2's Practice Problem 5.10 (Cascade)

If $v_1 = 2V$ and $v_2 = 1.5V$, find v_o in the circuit.



```

e1,1,0,2
e2,2,0,1.5
o1,1,3,3
r1,2,5,10'k
r2,3,6,20'k
r5,5,4,50'k
o2,0,5,4
r3,6,4,30'k
r6,6,0,60'k
o3,0,6,0

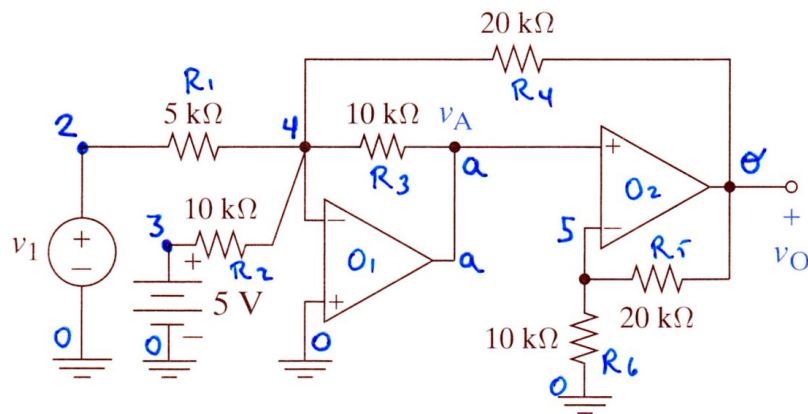
```

The answer you want is v_o , in **Results**.

The answer, **9**, is correct.

TR5's Example 4-16 (Cascade)

Derive an expression for v_o in terms of the two inputs.



```

e1,2,0,v1
e5,3,0,5
r1,2,4,5'k
r2,3,4,10'k
o1,0,4,a
r3,4,a,10'k
r4,4,0,20'k
o2,a,5,o
r5,5,0,20'k
r6,5,0,10'k

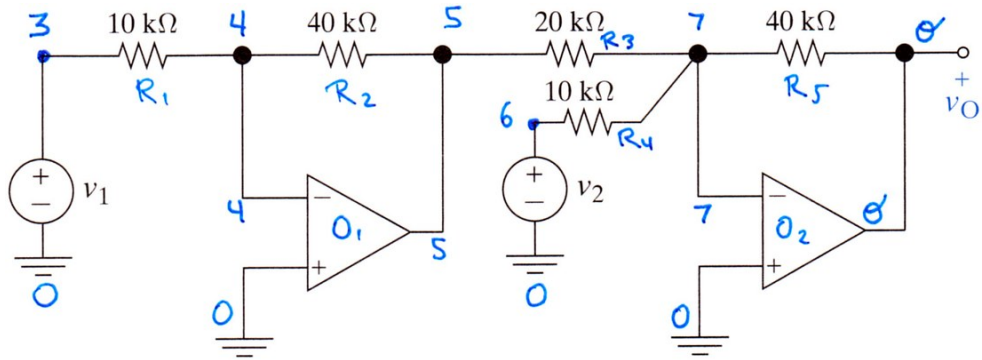
```

Ask **Evaluate** for `expand(vo)` with **Rounding** set to *approx (full precision)* in **Settings** — that is what the calculator's `expand(approx(vo))` is doing. It is arranged differently from the book's, which is a matter of presentation rather than of arithmetic.

The answer, **-2.4 v₁ - 6**, is correct.

TR5's Exercise 4-14 (Cascade)

Derive an expression for v_o in terms of the inputs v_1 and v_2 .



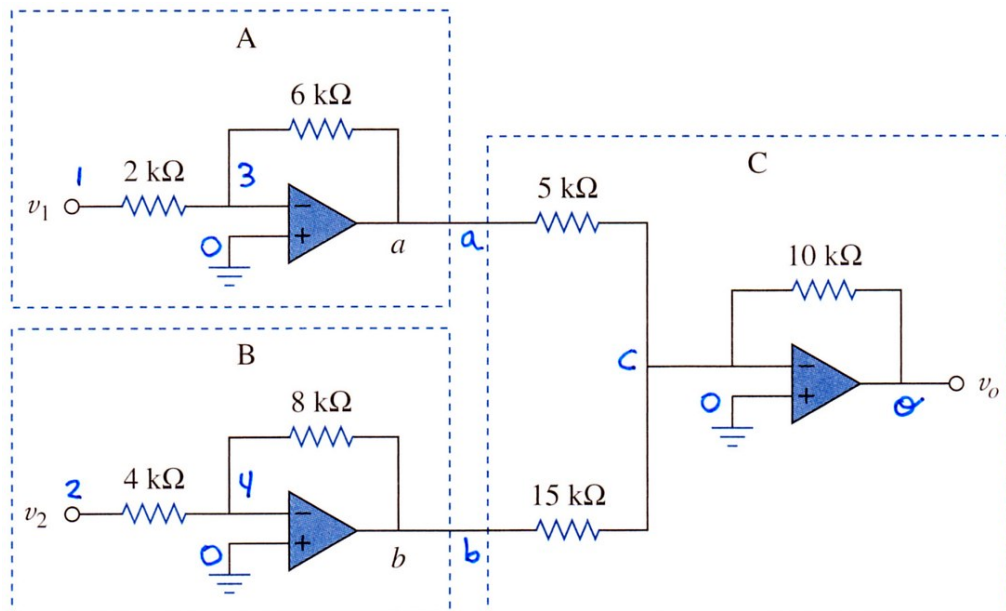
```
e1,3,0,v1
r1,3,4,10'k
o1,0,4,5
r2,4,5,40'k
r3,5,7,20'k
e2,6,0,v2
r4,6,7,10'k
r5,7,o,40'k
o2,0,7,o
```

Ask **Evaluate** for $\text{expand}(v_o)$, as on the calculator.

The answer, **$8 v_1 - 4 v_2$** , is correct.

AS2's Example 5.10 (Cascade)

If $v_1 = 1\text{V}$ and $v_2 = 2\text{V}$, find v_o in the circuit.



```
e1,1,0,1
e2,2,0,2
r2,1,3,2'k
r4,2,4,4'k
```



```

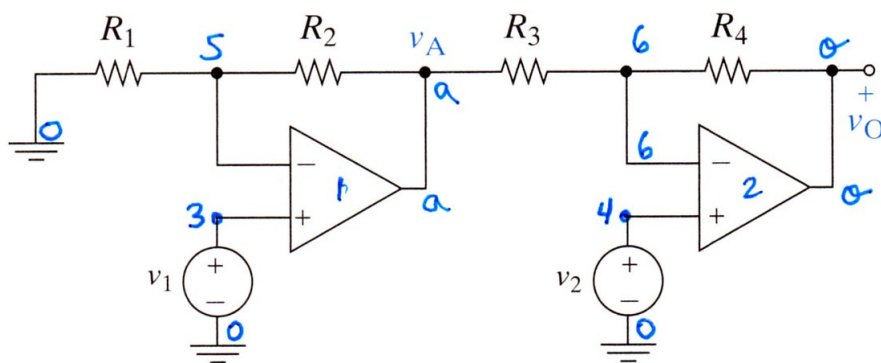
r6,3,a,6'k
r8,4,b,8'k
r5,a,c,5'k
r15,b,c,15'k
r10,c,o,10'k
oa,0,3,a
ob,0,4,b
oc,0,c,o

```

The answer, **8.667**, is correct.

TR5's Example 4-17 (Cascade)

Derive an expression for v_o in terms of the inputs v_1 and v_2 .



```

r1,5,0,r1
r2,5,a,r2
r3,a,6,r3
r4,6,o,r4
e1,3,0,v1
e2,4,0,v2
o1,3,5,a
o2,4,6,o

```

The answer you want is v_o , in **Results**.

We get the right answer. Let's compare it with the book's answer.

$$-\left[\frac{R_4}{R_3}\right]\left[\frac{R_1 + R_2}{R_1}\right]v_1 + \left[\frac{R_3 + R_4}{R_3}\right]v_2$$

Our answer, expanded via `expand(vo)`, is shown below:

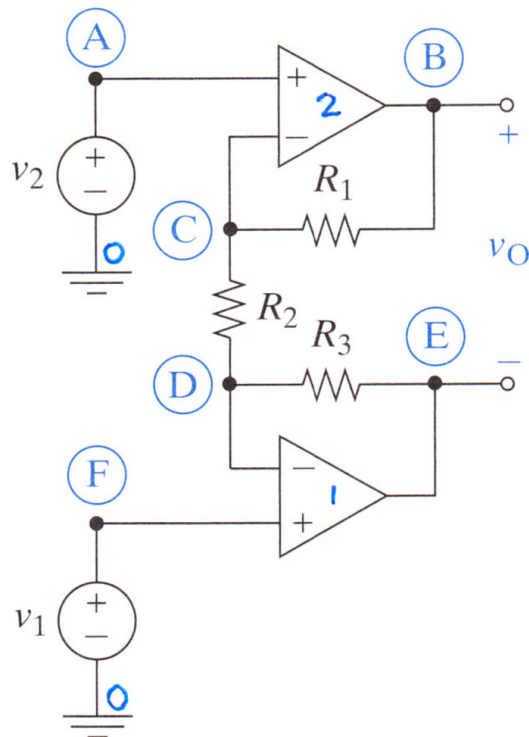
$$-\frac{r2r4v1}{r1r3} - \frac{r4v1}{r3} + \frac{r4v2}{r3} + v2$$

Playing with it by hand we get a form that, in my opinion, is *prettier* ;-) than the book's:

$$v2\left(\frac{r4}{r3} + 1\right) - v1\left(\frac{r4}{r3}\right)\left(\frac{r2}{r1} + 1\right)$$

TR5's Example 4-18 (Multiple)

Derive an expression for v_o in terms of the inputs v_1 and v_2 .



```
e2,a,0,v2
e1,f,0,v1
o2,a,c,b
o1,f,d,e
r1,b,c,r1
r2,c,d,r2
r3,d,e,r3
```

To find v_o , we ask for $v_b - v_e$. We get an expression that can easily be rearranged to look like this:

$$- \left(\frac{r1 + r2 + r3}{r2} \right) (v1 - v2)$$

This is exactly the answer from the book:

$$v_o = v_B - v_E = \left[\frac{R_1 + R_2 + R_3}{R_2} \right] (v_2 - v_1)$$

If you are ever in doubt whether two expressions are the same, subtract one from the other in **Evaluate**. If the answer is 0, they are the same — an equality sign would only be read as a comparison to solve, not as a question about sameness.

LESSON 6

Transient analysis

Learn to run a *transient* time-domain analysis with **tr**. Learn to describe *capacitors* with **c** and *inductors* with **l**. Save time with the **only** tool. Plot expressions in time with the **plot** tool.

Symulator can simulate two energy-storing elements, capacitors and inductors, in three kinds of analysis: transient time-domain (TR), alternating current (AC), and complex frequency-domain (FD). This lesson teaches the transient analysis of a circuit, and how to describe capacitors and inductors for it. AC and FD come in later lessons.

6.1 How to describe a capacitor

NOTE

Describing capacitors

When describing an ideal capacitor for the purpose of a transient analysis, we need five pieces of information, separated by commas: a unique name to identify the capacitor (must start with the letter **c**), the names of its first and second nodes, its value in farads (F), and its initial condition in volts, understood as the voltage in the first node minus the voltage in the second.

For example, an ideal capacitor called **ca**, connected between two nodes called **1** and **2**, with a capacitance of **3 F** and an initial condition of 0.1 V, would be described as follows: `ca,1,2,3,0.1`

Values can use SI prefixes

The values of the capacitance and the initial condition of the capacitor can use prefixes of the International System.

What answers do you get

For each capacitor, a transient analysis will provide two answers: the voltage drop in the capacitor, in volts, and the current through it, flowing from the first node towards the second, in amperes.

NOTE

Note on capacitors in DC

In a DC simulation, the current through a capacitor is defined as 0. It basically becomes an open circuit.

6.2 How to describe an inductor

NOTE

Describing inductors

When describing an ideal inductor for the purpose of a transient analysis, we need five pieces of information, separated by commas: a unique name to identify the inductor (must start with the letter **I**), the names of its first and second nodes, its value in henries (H), and its initial condition in amperes, understood as the current flowing through it from the first node towards the second.

For example, an ideal inductor called **I1**, connected between two nodes called **1** and **2**, with an inductance of **0.1** H and an initial condition of 0.2 A, would be described as follows: `I1,1,2,0.1,0.2`

For each inductor, a transient analysis will provide the voltage drop across it and the current through it, with the same conventions as for capacitors.

NOTE

Note on inductors in DC

In a DC simulation, the voltage drop in an inductor is defined as 0. It basically becomes a short circuit.

6.3 Running a transient analysis

You will learn the subtleties of transient simulations through a series of examples. For now, to run one we use the **tr** function, which takes the circuit description and, optionally, the list of quantities you want back.

For transient simulations, Symbulator 9 uses SymPy's own Laplace machinery. There is nothing extra to install.

Bo2's Example 4.15

SOLUTION

For $t \leq 0$ s, since the current source is inactive, there is nothing going on. Without doing a simulation, we know the voltage v is 0 volts.

For $0 < t \leq 2$ s, the source is active. From the graph, its value in this interval is t : the current in amperes equals the time in seconds. We describe the circuit and run the transient analysis:

EXAMPLE 4.15

For the circuit shown in Fig. 4.37(a), let us determine v given that i is described by Fig. 4.37(b). In this case, let us consider three intervals of time.

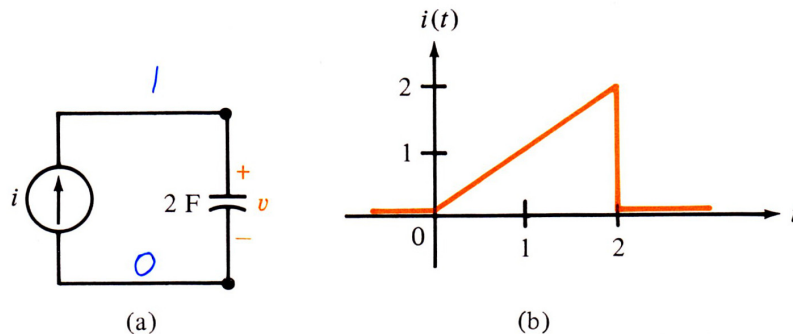


Fig. 4.37 (a) Circuit with a capacitor. (b) Applied current.

Bo2's Example 4.15

```
j,0,1,t  
c,1,0,2,0
```

The source has four terms: name (starting with j), first node, second node, and value — here t . The capacitor has five: name, first node, second node, value in farads, and initial condition in volts — here 0 volts.

Once the simulation is complete, ask for the value of the voltage, v_c . We get this value, which is correct:

```
t^2/4
```

For $2\text{ s} < t$, the voltage is equal to whatever value it had at time 2 s:

The calculator's $|$ says “given”; version 9 says the same thing in the **Evaluate** card's **Conditions** box:

```
v_c
```

```
t = 2
```

We get a value of 1 volt, which is correct.

Describing source values

For the purpose of a transient analysis, Symbolator accepts many types of source values. The only thing you must do is describe these values properly.

Step values: symbolic. A source with an unknown step value starting at $t=0$ is described with a variable and the step function $u(t)$. For example, a voltage source $e1$ between nodes 1 and 0, of V volts starting at $t=0$, is $e1,1,0,V*u(t)$.

Step values: numerical. A step source with a known numerical value can be described the same way, $e1,1,0,12*u(t)$. To save typing, you can skip the $u(t)$ when the value is

numerical, and Symbulator assumes a step: $e1, 1, 0, 12$. The results are the same.

Impulse values. A source with an impulse value at $t=0$ is described with its value, symbolic or numerical, and the delta function $\delta(t)$. For example, a current source $j1$ between nodes 0 and 1, with an impulse of i amperes at $t=0$, is $j1, 0, 1, i*\delta(t)$.

Values as functions of time. A source whose value is a function of time, such as a ramp, a sinusoid or an exponential, is described by writing the value as an expression in terms of t , as in $j, 0, 1, t$.

Dependent values. Sources with dependent values are described as we have seen before, for example $j, 0, 1, 3*vr1$.

A word on intervals

In a transient analysis, every switch opening or closing ends one time interval and begins another, and a simulation covers one interval only. When we run a TR simulation and get answers in terms of t , this variable must be understood as the time elapsed since the start of *that* interval in particular.

This is a distinction without a difference as long as the interval we simulate starts at $t=0$, which is most often the case. However, sometimes problems have switch changes at other times — Bo2's Example 5.6, where a switch closes at $t=1$ s, and Bo2's Drill Exercise 5.6, where one closes at $t=2$ s, are examples in the solved problems.

6.4 Two useful tools

The only tool

Unless you say otherwise, a Symbulator simulation gives you the whole set of answers: voltages in all nodes, and voltage drops, currents and power consumed in all elements. When the expressions are complicated, storing them all takes time.

In case you don't want all the answers, but only a chosen few, tick **Do you want to limit the results to save time?** — it appears under the analysis menus once you choose TR. A box opens asking **What results are you after?** List them there, separated by commas:

$v2$

This matters most in TR analysis, where each answer costs an inverse Laplace transform. Asking for one instead of a dozen is the single easiest speed-up available to you.

The plot tool

Symbulator 9 draws the plot for you, in the **Plot** card below the results. Solve the circuit first — the plot is of an answer, so there has to be one — then open **Plot** and fill in four things:

- **Plot type:** *Plot a function of time (TR)*
- **Variable to plot:** the answer you want, such as $v2$

- **Start time (s)** and **End time (s)**: the window, for example 0 and 0.005
- **Points**: how finely to sample it; 300 is the default and is usually plenty

Press **Run** and the curve appears under the card.

The second field is worth a moment. It takes an answer's name, not an expression, and it must be one this circuit actually has — v_2 and i_{r1} rather than v_c or a formula of your own. If you are unsure of the spelling, **Results** lists every name above.

WARNING

A curve the inverse Laplace could not find

Some circuits have an answer the inverse Laplace transform cannot close into an expression. Symbolator samples the s-domain answer numerically for the plot instead, so the picture still appears even when **Results** shows nothing for that variable. If a plot is blank where you expected a curve, check that the variable is spelled the way **Results** spells it.

6.5 Instructive solved examples

NOTE

Some of this narration still describes the calculator

Every problem below carries a **Circuit Description** panel you can type straight into Symbolator 9, and the circuits and answers are identical across the versions. What has not all been rewritten is the narration between them: where it says to evaluate a name or press a key, do the version 9 equivalent — the results are already on screen.

Transient analysis of RC circuit

Bo2's Example 5.1

EXAMPLE 5.1

Let us begin this example by considering the circuit in Fig. 5.4, in which there is a switch that is closed for time $t < 0$, is opened at time $t = 0$, and stays open for all time $t > 0$. For this circuit, let us determine $v_C(t)$ for all values of t .

Solution. Here we have two intervals. In the first, $t < 0$ s, we assume things have been steady for a long time and any transient has passed, so we run a DC simulation to find the initial conditions for the second.

```
e,1,0,V
r1,1,2,r1
c,2,0,c
r,2,0,r
```

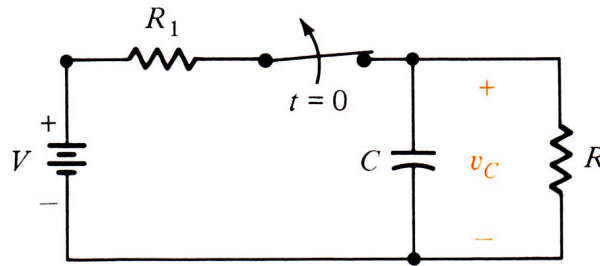


Fig. 5.4 RC circuit with a switch.

When the simulation is *Done*, we ask for v_C , the voltage in the capacitor **c**. We get the expression shown below, which is correct.

For the second interval, when $0s \leq t$, we do a transient simulation. We give the capacitor as initial condition its voltage from the previous interval.

```
c,2,0,c,r*v/(r+r1)
r,2,0,r
```

When the simulation is *Done*, we ask for v_C again. We get the expression below, which is correct.

$$e^{-t/cr}(rv)/(r+r1)$$

Bo2's Drill Exercise 5.1

DRILL EXERCISE 5.1

For the circuit shown in Fig. DE5.1, find a single expression for all t for (a) $v_C(t)$ and (b) $i_C(t)$.

Answer: (a) $6 - 6(1 - e^{-4t})u(t)$ V; (b) $-2e^{-4t}$ A

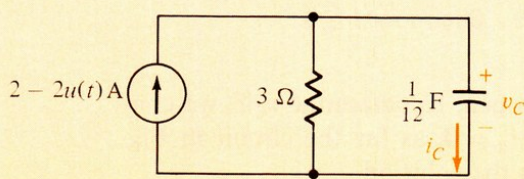


Fig. DE5.1

Solution. Again, we have two intervals: the first for DC, the second for TR.

For the first interval, when $t < 0s$, the current source is 2A. Simulate in DC.

```
j,0,1,2
r,1,0,3
c,1,0,1/12
```

The answers you want are v_C and i_C , in **Results**.

{ 6 , 0 }

So the voltage in the capacitor is 6V. We store this value in a variable called $\mathbf{vc_0}$, to use it

as initial condition of the capacitor for the second interval.

For the second interval, when $0s \leq t$, we do a transient simulation. The current source is 0A, and the capacitor has an initial condition: voltage v_{C0} .

```
j,0,1,0
r,1,0,3
c,1,0,1/12,vc0
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_c and i_c , in **Results**.

When the simulation is *Done*, we get the expressions below. They are right.

$$\{6e^{-4t}, -2e^{-4t}\}$$

Bo2's p224 5.2

EXAMPLE 5.2

For the circuit shown in Fig. 5.7, let us determine $v_c(t)$, $i_c(t)$, and $v(t)$ for all time.

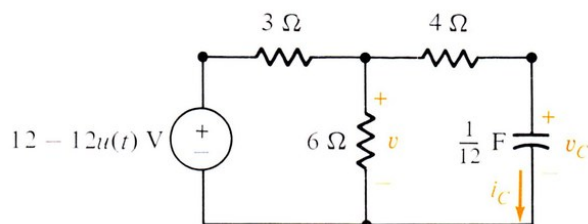


Fig. 5.7 Circuit for Example 5.2.

Solution. Again, we have two intervals: the first for DC, the second for TR.

For the interval when $t < 0s$, the voltage source is 12 V. Simulate in DC.

```
e,1,0,12
r3,1,2,3
r6,2,0,6
r4,2,3,4
c,3,0,1/12
```

The answers you want are v_c , i_c and v_2 , in **Results**.

$$\{8, 0, 8\}$$

For the interval when $0s \leq t$, the voltage source is 0V. Simulate in TR, giving the capacitor as initial condition its voltage from the previous interval.

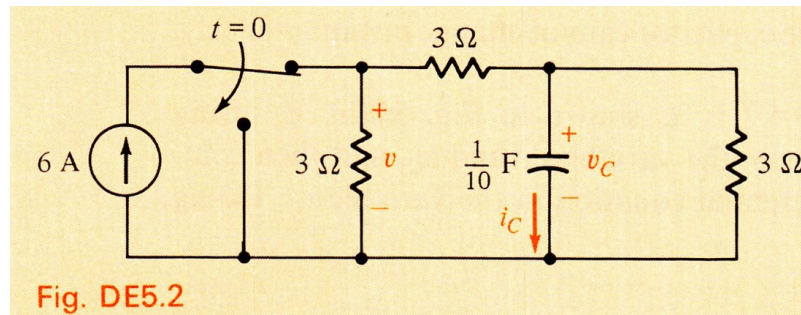
```
e,1,0,0
r3,1,2,3
r6,2,0,6
r4,2,3,4
c,3,0,1/12,8
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_C , i_C and v_2 , in **Results**.

$$\{8e^{-2t}, (-4/3)e^{-2t}, (8/3)e^{-2t}\}$$

Bo2's Drill Exercise 5.2



DRILL EXERCISE 5.2

For the circuit shown in Fig. DE5.2, find an expression for all t for (a) $v_C(t)$, (b) $i_C(t)$, and (c) $v(t)$.

Answer: (a) $6 - 6(1 - e^{-5t})u(t)$ V; (b) $-3e^{-5t}u(t)$ A; (c) $12 - 12u(t) + 3e^{-5t}u(t)$ V

Solution. For $t < 0$ s, the 6A source is part of the circuit. Simulate in DC.

```
j,0,1,6
r10,1,0,3
r12,1,2,3
r20,2,0,3
c,2,0,1/10
```

The answers you want are v_C , i_C and v_1 , in **Results**.

$\{6, 0, 12\}$

For $0\text{s} \leq t$, the source is no longer relevant and can be excluded from the circuit. Simulate in TR, giving the capacitor its proper initial condition.

```
r10,1,0,3
r12,1,2,3
r20,2,0,3
c,2,0,1/10,6
```

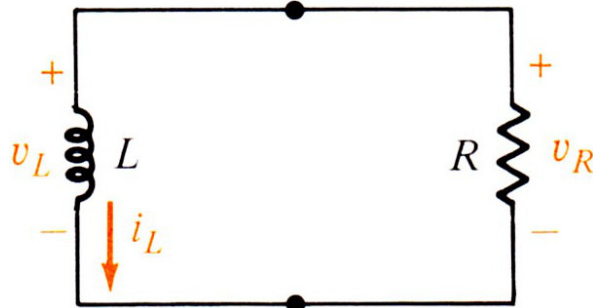
Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_C , i_C and v_1 , in **Results**.

$$\{6e^{-5t}, -3e^{-5t}, 3e^{-5t}\}$$

Bo2's Figure 5.10a

Look at the simple resistor-inductor (RL) circuit shown, where at time $t=0$ the inductor current is $i_L(0)$. Determine $v_L(t)$, $i_L(t)$ and $v_R(t)$ for $t \geq 0$.



Since we are already given the initial condition, we only run the transient simulation for $t \geq 0$. In our circuit description below, the initial condition is **il0**.

```
l,1,0,l,il0
r,1,0,r
```

Set **Analysis** to *TR – transient / time domain*.

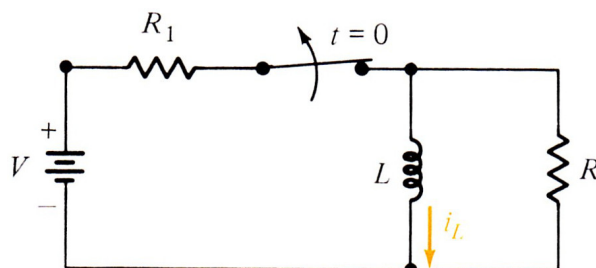
The answers you want are v_l , i_l and v_r , in **Results**.

We get the right answers:

$$\{-il0re^{(-rt)/l}, il0e^{(-rt)/l}, -il0re^{(-rt)/l}\}$$

Bo2's Example 5.3

Determine $i_L(t)$ for all t .



Solution. For $t < 0s$, simulate in DC.

```
e,1,0,v
r1,1,2,r1
l,2,0,l
r2,2,0,r
```

The answer you want is i_l , in **Results**.

For $t \geq 0s$, simulate in TR, giving the inductor its initial condition.

```
l,2,0,1,v/r1
r2,2,0,r
```

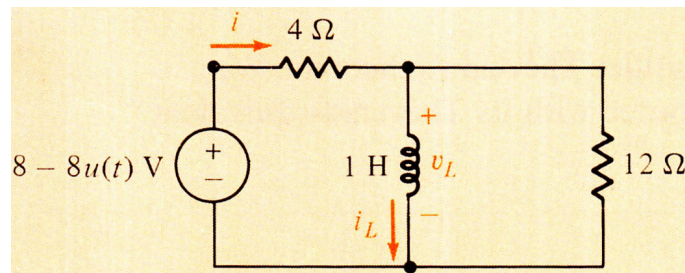
Set **Analysis** to *TR – transient / time domain*.

The answer you want is *i1*, in **Results**.

$$e^{(-rt)/l}v/r1$$

Bo2's Drill Exercise 5.3

Find $i_L(t)$, $v_L(t)$ and $i(t)$ for all t .



Solution. For $t < 0s$, simulate in DC.

```
e,1,0,8
r4,1,2,4
l,2,0,1
r12,2,0,12
```

The answers you want are *i1*, *v1* and *ir4*, in **Results**.

{ 2 , 0 , 2 }

For $t \geq 0s$, simulate in *TR*, giving the inductor its initial condition of 2A.

```
e,1,0,0
r4,1,2,4
l,2,0,1,2
r12,2,0,12
```

Set **Analysis** to *TR – transient / time domain*.

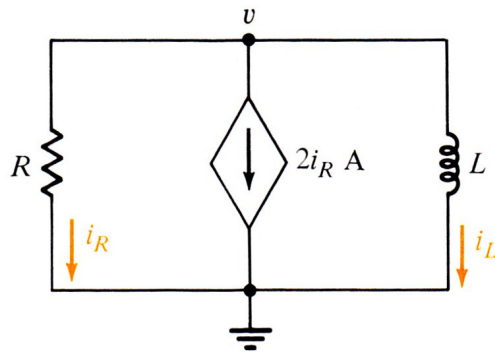
The answers you want are *i1*, *v1* and *ir4*, in **Results**.

$$\{2e^{-3t}, -6e^{-3t}, (3/2)e^{-3t}\}$$

Bo2's p230 (Dependent source)

Find $i_L(t)$ for $t \geq 0$, given that $i_L(0) = 5A$.

We only run the transient simulation for $t \geq 0$, with initial condition 5A.



```
r,v,0,r
j,v,0,2*ir
l,v,0,l,5
```

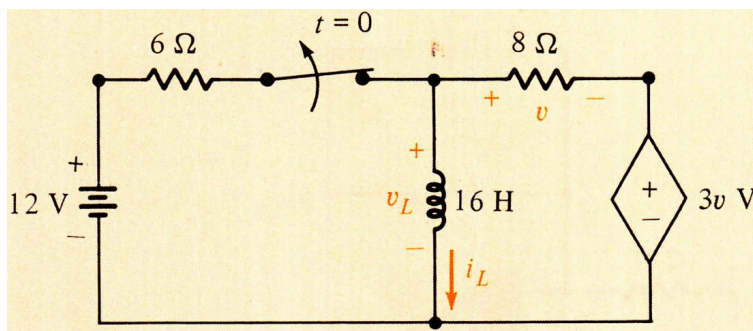
Set **Analysis** to *TR* – transient / time domain.

The answer you want is **i_L**, in **Results**.

$$5e^{((-Rt)/3L)}$$

Bo2's Drill Exercise 5.4 (Dependent source)

Find $i_L(t)$ and $v_L(t)$ for all t .



Solution. For $t < 0$ s, simulate in DC. This gives us the initial condition.

```
ei,1,0,12
r6,1,2,6
l,2,0,16
r8,2,3,8
ed,3,0,3*vr8
```

The answers you want are **i_L** and **v_L**, in **Results**.

{ 2, 0 }

For $t \geq 0$ s, simulate in *TR*, giving the inductor its initial condition.

```
l,2,0,16,2
r8,2,3,8
ed,3,0,3*vr8
```

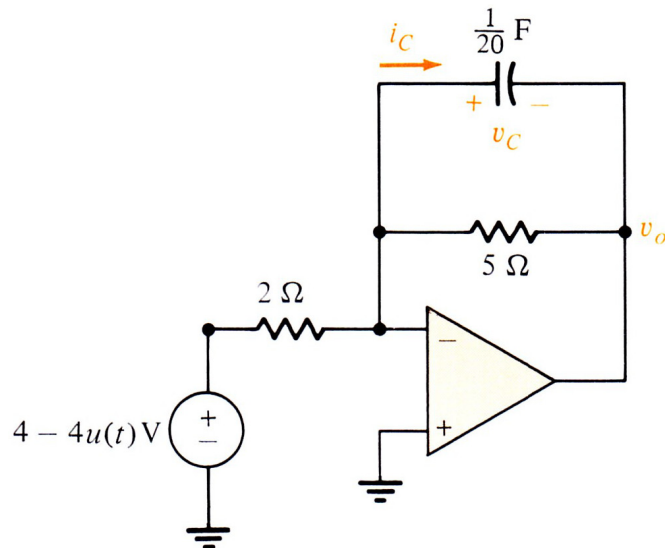
Set **Analysis** to *TR – transient / time domain*.

The answers you want are i_1 and v_1 , in **Results**.

$$\{2e^{-2t}, -64e^{-2t}\}$$

Bo2's Example 5.5 (Op Amp)

Determine $v_C(t)$, $i_C(t)$ and $v_o(t)$ for all t .



Solution. For $t < 0s$, simulate in DC. This gives us the initial condition.

```
e,3,0,4
r2,3,1,2
r5,1,0,5
c,1,0,1/20
o,0,1,0
```

The answers you want are v_c , i_c and v_o , in **Results**.

$$\{10, 0, -10\}$$

For $t \geq 0s$, simulate in TR, giving the capacitor its initial condition.

```
r2,0,1,2
r5,1,0,5
c,1,0,1/20,10
o,0,1,0
```

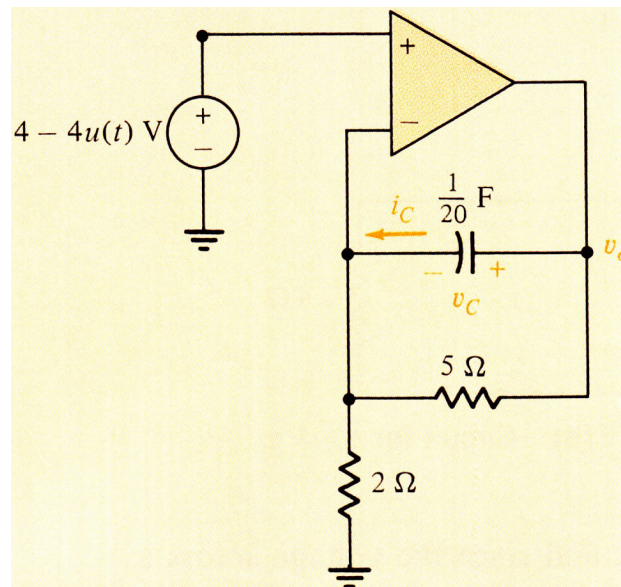
Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_c , i_c and v_o , in **Results**.

$$\{10e^{-4t}, -2e^{-4t}, -10e^{-4t}\}$$

Bo2's Drill Exercise 5.5 (Op Amp)

Determine $v_C(t)$, $i_C(t)$ and $v_o(t)$ for all t .



Solution. For $t < 0s$, simulate in DC. This gives us the initial condition.

```
e,2,0,4
o,2,1,o
c,o,1,1/20
r5,o,1,5
r2,1,0,2
```

The answers you want are v_C , i_C and v_o , in **Results**.

{ 10, 0, 14 }

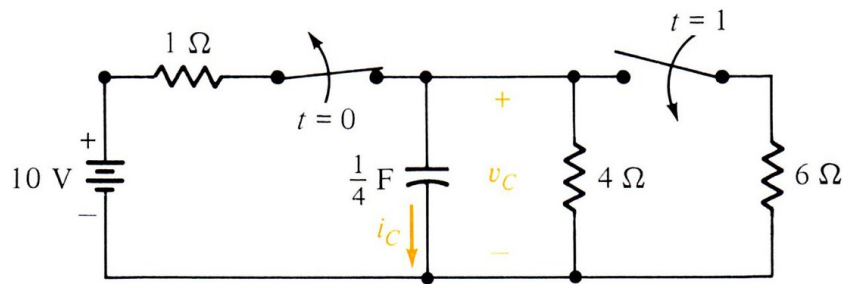
For $t \geq 0s$, simulate in TR, giving the capacitor its initial condition.

```
o,0,1,o
c,o,1,1/20,10
r5,o,1,5
r2,1,0,2
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_C , i_C and v_o , in **Results**.

$$\{10e^{-4t}, -2e^{-4t}, 10e^{-4t}\}$$



Bo2's Example 5.6

In the circuit shown, there are two switches: one that opens at time $t=0$ and one that closes at time $t=1$ second. Determine $v_C(t)$ and $i_C(t)$ for all t .

As we mentioned before, every time a switch moves marks the end of one interval and the beginning of another interval for Symbolator.

For the first interval, which corresponds to $t < 0$, we simulate in DC.

```
e,2,0,10
r1,2,1,1
c,1,0,1/4
r4,1,0,4
```

The answers you want are v_C and i_C , in **Results**.

$\{8, 0\}$

The voltage in the capacitor at the end of this interval will serve as the initial condition of the capacitor for the next interval. The second interval runs between 0 and 1 second, i.e. $0 < t \leq 1$ second. We simulate in TR.

```
c,1,0,1/4,8
r4,1,0,4
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_C and i_C , in **Results**.

$$\{8e^{-t}, -2e^{-t}\}$$

The capacitor's voltage at the end of this second interval is the initial condition for the third. We can use its exact value, , but the textbook prefers using its approximate value, i.e. 2.943.

The third interval corresponds to $t > 1$ second. We simulate in TR. To make things easier for Symbolator, we replace the resistors by their equivalent.

```
c,1,0,1/4,2.943
re,1,0,[4,6]
```

Ask for $\{v_C, i_C\}$. The expressions we get are equivalent to:

$$\{2.943e^{(-5(t-1))/3}, -1.226e^{(-5(t-1))/3}\}$$

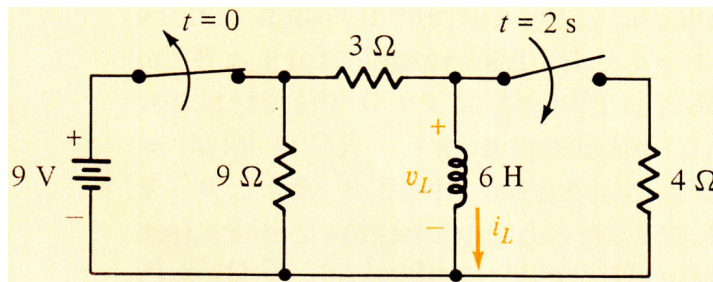
These are the right answers, as can be seen by checking the book's answers.

$$v_C(t) = \begin{cases} 8 \text{ V} & \text{for } t < 0 \\ 8e^{-t} \text{ V} & \text{for } 0 \leq t < 1 \text{ s} \\ 2.943e^{-5(t-1)/3} \text{ V} & \text{for } t \geq 1 \text{ s} \end{cases}$$

$$i_C(t) = \begin{cases} 0 \text{ A} & \text{for } t < 0 \\ -2e^{-t} \text{ A} & \text{for } 0 \leq t < 1 \text{ s} \\ -1.226e^{-5(t-1)/3} \text{ A} & \text{for } t \geq 1 \text{ s} \end{cases}$$

Bo2's Drill Exercise 5.6

Let's see another example where a TR simulation is done for an interval that starts at a time other than $t=0$. Determine $i_L(t)$ and $v_L(t)$ for all t .



For the first interval, which corresponds to $t < 0$, we simulate in DC.

```
e,1,0,9
r9,1,0,9
r3,1,2,3
l,2,0,6
```

The answers you want are i_l and v_l , in **Results**.

$\{3, 0\}$

The 3A current in the inductor is its initial condition for the second interval, which goes from 0 to 2 seconds, i.e. $0 < t \leq 2$ seconds. We simulate in TR.

```
r,1,0,9+3
l,1,0,6,3
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are i_l and v_l , in **Results**.

$$\{3e^{-2t}, -36e^{-2t}\}$$

The inductor's current at the end of this second interval is the initial condition for the third. Find its approximate value thus:

.055

The third interval corresponds to $t > 2$ second. We simulate in TR. To make things easier for Symbulator, we replace the resistors by their equivalent.

```
r,1,0,[9+3,4]
1,1,0,6,.055
```

Set **Analysis** to *TR – transient / time domain*.

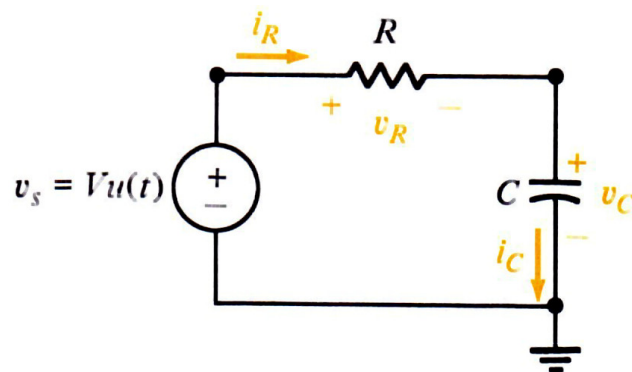
The answers you want are i_1 and v_1 , in **Results**.

$$\{.055e^{-(t-2)/2}, -.165e^{-(t-2)/2}\}$$

These are the kind of expressions your book or professor are looking for.

Bo2's Figure 5.19

In the following circuit, assume that all the initial conditions are zero. The source is a step function $u(t)$ with value V volts. Find i_R , v_R , i_C and v_C .



As discussed before, a step source with symbolic value requires that we use the $u(t)$ nomenclature to describe it. Since the value is V volts, we describe the source as $V \cdot u(t)$.

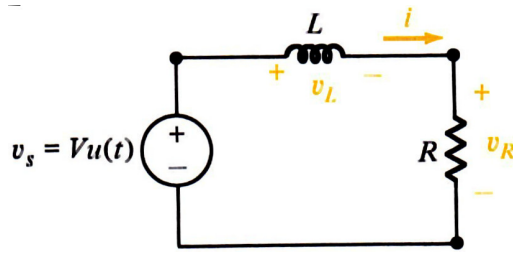
```
e,1,0,V*u(t)
r,1,2,r
c,2,0,c,0
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_c , v_r and i_c , in **Results**.

There is no need to ask for i_R , since $i_R = i_C$. We get the following expressions:

$$\{v - ve^{(-t/(cr))}, ve^{(-t/(cr))}, (v/r)e^{(-t/(cr))}\}$$



Bo2's Figure 5.24

In the following circuit, assume that all the initial conditions are zero. The source is a step function $\mathbf{u(t)}$ with value $\mathbf{V}$ volts. Find i_R , v_R , i_L and v_L .

```
e,1,0,V*u(t)
1,1,2,1,0
r,2,0,r
```

Set **Analysis** to *TR – transient / time domain*.

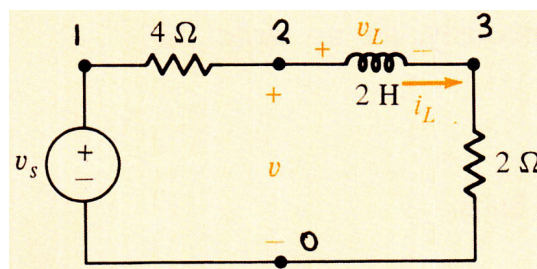
The answers you want are v_L , v_R and i_L , in **Results**.

There is no need to ask for i_R , since in a series circuit it will be identical to i_L .

$$\{ve^{(-rt)/l}, v(1 - e^{(-rt)/l}), (v/r)(1 - e^{(-rt)/l})\}$$

Bo2's Drill Exercise 5.7

For the circuit, with a step source of 12 volts, find i_L , v_L , i_L and v .



As discussed before, a step source with numerical value can be described without the $u(t)$ nomenclature. Thus, we describe the source as 12.

```
e,1,0,12
r4,1,2,4
1,2,3,2,0
r2,3,0,2
```

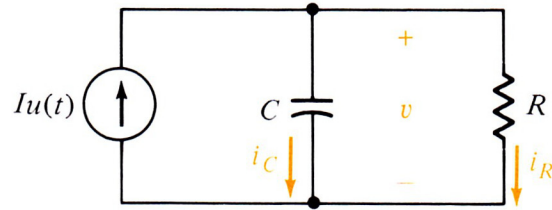
Set **Analysis** to *TR – transient / time domain*.

The answers you want are i_L , v_L and v_2 , in **Results**.

$$\{2 - 2e^{-3t}, 12e^{-3t}, 8e^{-3t} + 4\}$$

Bo2's Figure 5.26

In the following circuit, assume that all the initial conditions are zero. The source is a step function $\mathbf{u(t)}$ with value $\mathbf{I}$ amperes. Find i_C , i_R , and v .



As discussed before, a step source with symbolic value requires that we use the $u(t)$ nomenclature to describe it. Since the value is I amperes, we describe the source as $I \star u(t)$. Thus:

```
j,0,1,i*u(t)
c,1,0,c,0
r,1,0,r
```

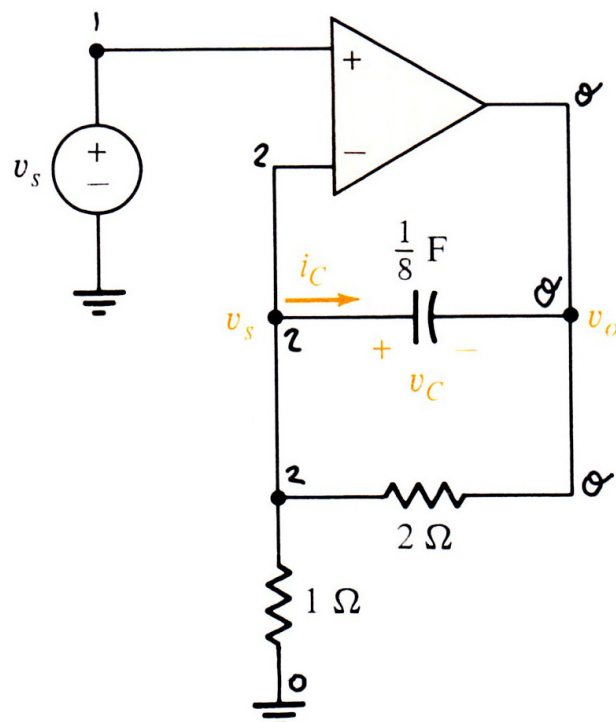
Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_1 , i_r and i_c , in **Results**.

$$\{(i - ie^{-t/(cr)}) \cdot r, i - ie^{-t/(cr)}, ie^{-t/(cr)}\}$$

Bo2's Example 5.7 (Op Amp)

Suppose that $v_s(t)=u(t)$. Find v_C , i_C , and v_o .



This source is a step source with a value of 1 volt. Since its value is numerical, the $u(t)$ nomenclature is not needed. We describe the source plainly as 1.

```
e,1,0,u(t)
o,1,2,o
c,2,o,1/8,0
r2,2,o,2
r1,2,0,1
```

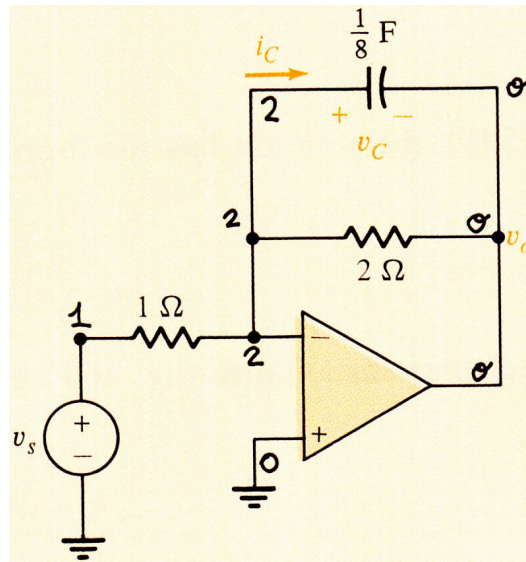
Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_C , i_C and v_O , in **Results**.

$$\{2e^{-4t} - 2, -e^{-4t}, 3 - 2e^{-4t}\}$$

Bo2's Drill Exercise 5.8 (Op Amp)

Suppose that $v_s(t) = u(t)$. Find v_C , i_C , and v_O .



Again, following the same logic as above, we describe the source plainly as 1.

```
e,1,0,u(t)
r1,1,2,1
r2,2,o,2
c,2,o,1/8,0
o,0,2,o
```

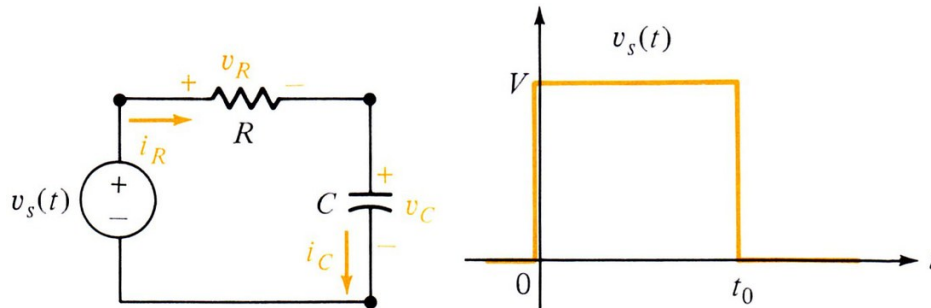
Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_C , i_C and v_O , in **Results**.

$$\{2 - 2e^{-4t}, e^{-4t}, 2e^{-4t} - 2\}$$

Bo2's p249 F5.28

For the circuit left, the source is $v_s(t) = V u(t) - V u(t-t_0)$. This is shown in the plot to the right. Given this, find the voltage drop in the capacitor for all t .



Because the source has two steps, this problem has to be analyzed in two intervals. For the first interval, the initial conditions are zero and the source value is $V \cdot u(t)$. Our circuit description for the first interval is given below.

```
e,1,0,V*u(t)
r,1,2,r
c,2,0,c,0
```

Set **Analysis** to *TR – transient / time domain*.

The answer you want is v_C , in **Results**.

The expression for the capacitor's voltage drop in the first interval is:

$$v - v e^{(-t/(cr))}$$

The second interval starts at t_0 . In it, the initial condition of the capacitor is given by the value of the expression above when $t=t_0$. Here is how we find it:

Put v_C in the **Evaluate** card and write $t = t_0$ in its **Conditions** box. That is what the calculator's $|$ does: evaluate this expression under this condition. The answer comes back $V - V e^{-\frac{t_0}{cr}}$.

The circuit description for the second interval uses that expression as the capacitor's initial condition. The source is now 0 volts – another way of saying it becomes a short – so it need not appear in the description.

$$(v - v e^{(-t_0/(cr))}) e^{((-t+t_0)/(cr))}$$

```
r,0,2,r
c,2,0,c,v-e^(-t_0/(c*r))*v
```

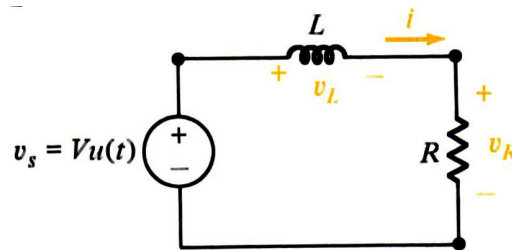
Set **Analysis** to *TR – transient / time domain*.

The answer you want is v_C , in **Results**.

This is the expression for the capacitor's voltage drop in the second interval.

Bo2's Drill Exercise 5.9

Find the current through the inductor if the source is $v_s(t) = V u(t) - V u(t-t_0)$.



Because the source has two steps, this problem has to be analyzed in two intervals. For the first interval, the initial conditions are zero and the source value is $V^*u(t)$. Our circuit description for the first interval is given below.

```
e,1,0,V*u(t)
1,1,2,1,0
r,2,0,r
```

Set **Analysis** to *TR – transient / time domain*.

The answer you want is **i1**, in **Results**.

The expression for the inductor's current in the first interval is:

$$(v/r)(1 - e^{((-rt)/l)})$$

The second interval starts at t_0 . In it, the initial condition of the inductor is given by the value of the expression above when $t=t_0$. Here is how we find it:

Put **i1** in the **Evaluate** card and write $t = t_0$ in its **Conditions** box. The answer comes back $\frac{V}{r} - \frac{Ve^{-\frac{rt_0}{l}}}{r}$, which is the same expression gathered differently.

The circuit description for the second interval uses that expression as the inductor's initial condition. The source becomes a short, so it need not appear in the description.

$$(v/r)(1 - e^{((-tor)/l)})e^{((-rt)/l)}$$

```
1,0,2,1,v/r-e^(-to*r/l)*v/r
r,2,0,r
```

Set **Analysis** to *TR – transient / time domain*.

The answer you want is **i1**, in **Results**.

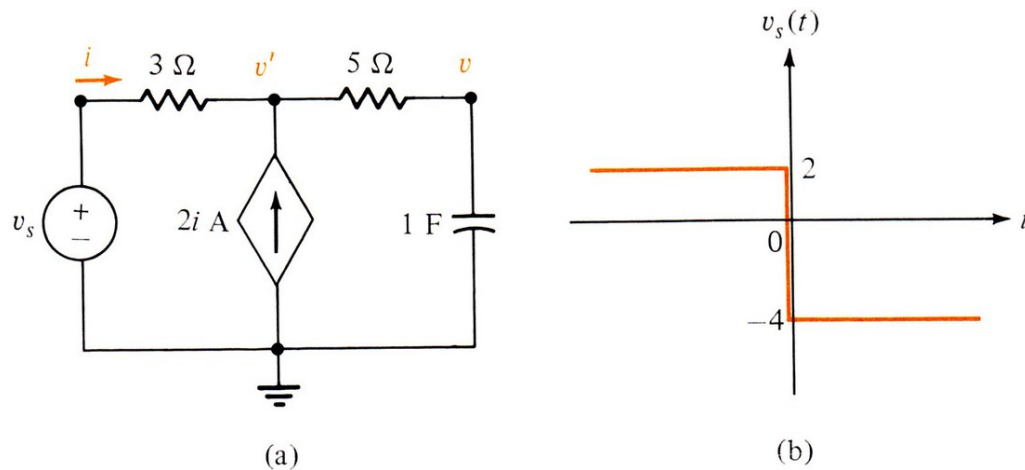
To put this in terms of the same t as the first interval, we replace t with $t-t_0$:

$$(v/r)(1 - e^{((-tor)/l)})e^{((-r(t-t_0))/l)}$$

This is the expression for the inductor's current in the second interval.

Bo2's Figure 5.36

For the circuit shown in (a), find the voltage drop in the capacitor, given that the voltage source is as shown in (b).



This problem requires two intervals. In the first interval, a DC analysis of the circuit with a source of 2 V gives us the voltage in the capacitor.

```
e,1,0,2
r3,1,2,3
r5,2,3,5
c,3,0,1
j,0,2,2*ir3
```

The answer you want is v_c , in **Results**.

2

This is the initial condition for the capacitor in the second interval, which we analyze using TR and a source value of -4 V.

```
e,1,0,-4*u(t)
r3,1,2,3
r5,2,3,5
c,3,0,1,2
j,0,2,2*ir3
```

Set **Analysis** to *TR – transient / time domain*.

The answer you want is v_c , in **Results**.

$$6e^{(-t/6)} - 4$$

This is the expression for the voltage drop in the capacitor after $t=0$.

The only tool

Bo2's Drill Exercise 5.11

For the circuit of Bo2's Example 5.7, change the value of the capacitor to $\frac{1}{4}$ F. Find $v_C(t)$ for the case that the source is $v_S(t)=1$ V for $t<0$ and 3V for $t\geq 0$.

We let Symbulator know that we are only interested in one variable: v_C .

This is optional: Symbulator 9 solves quickly enough that limiting the results rarely saves you anything worth having. If you want to anyway, tick **Do you want to limit the results to save time?** — it sits under the analysis menus once TR is chosen — and list v_C in the field it reveals.

Then we run the simulation for the first interval, just as we did before, but with the new capacitor and source values.

```
e,1,0,1
o,1,2,o
c,2,o,1/4
r2,2,o,2
r1,2,0,1
```

The answer you want is v_C , in **Results**.

-2

This is the initial condition for the next interval. Before we run the simulation for the second interval, we let Symbulator know, again, that we are only interested in one variable: v_C .

This is optional: Symbulator 9 solves quickly enough that limiting the results rarely saves you anything worth having. If you want to anyway, tick **Do you want to limit the results to save time?** — it sits under the analysis menus once TR is chosen — and list v_C in the field it reveals.

Then we run the simulation for the second interval, just as we did before, but with the new capacitor and source values.

```
e,1,0,3*u(t)
o,1,2,o
c,2,o,1/4,-2
r2,2,o,2
r1,2,0,1
```

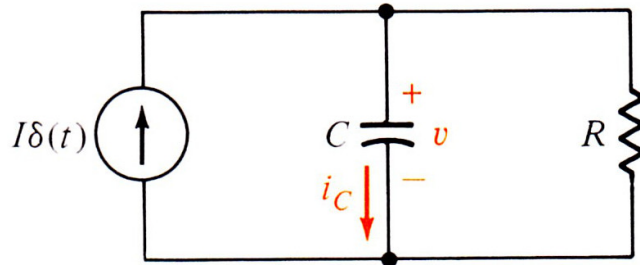
Set **Analysis** to *TR — transient / time domain*.

The answer you want is v_C , in **Results**.

$$4e^{(-2t)} - 6$$

Bo2's Figure 5.38 (Impulse)

Find the voltage drop and current in the capacitor, given a source of $I \delta(t)$ A, where $\delta(t)$ is the impulse function and I is a constant.



As we mentioned before, impulse sources must be described using the $\delta(t)$ nomenclature. Typing `delta(t)` spells it without the Greek.

```
j,0,1,i*delta(t)
c,1,0,c,0
r,1,0,r
```

Once the simulation completes, we ask for the variables of interest:

$$(i/c)e^{(-t/(cr))}$$

$$(-i/(cr))e^{(-t/(cr))}$$

They are already in **Results**: v_c is $\frac{ie^{-\frac{t}{cr}}}{c}$ and i_c is $i\delta(t) - \frac{ie^{-\frac{t}{cr}}}{cr}$.

The impulse survives in the capacitor's current, which is right — the charge arrives all at once, and the delta is that instant.

Bo2's Drill Exercise 5.13 (Impulse)

For the circuit of Bo2's Example 5.7, find $v_c(t)$, $i_c(t)$ and $v_o(t)$ due to an impulse voltage source of $v_s(t) = \delta(t)$ (that is to say, a 1V impulse in $t=0$.)

Nothing new here. Again we use the $\delta(t)$ nomenclature for the source.

```
e,1,0,delta(t)
o,1,2,o
c,2,0,1/8,0
r2,2,o,2
r1,2,0,1
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_c , i_c and v_o , in **Results**.

$$\{-8e^{-4t}, 4e^{-4t}, 8e^{-4t}\}$$

Bo2's Figure 5.39 (Ramp)

Find $i_L(t)$ and $v_L(t)$ for a circuit that has in parallel a resistor R , an inductor L and a current source with a ramp value $I r(t)$. Initial conditions are zero.

A ramp value $I r(t)$ means a value of $I t$. We describe this source simply as $i \star t$.

```
j,0,1,i*t
r,1,0,r
l,1,0,1,0
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are i_l and v_l , in **Results**.

$$\{((i_l)/r)(e^{((-rt)/l)} - 1) + it, i_l(1 - e^{((-rt)/l)})\}$$

Bo2's Drill Exercise 5.14 (Ramp)

For the circuit of Bo2's Example 5.7, find $v_C(t)$ and $i_C(t)$ due to a ramp input voltage of $v_S(t) = r(t)$.

Nothing new here. The command below should be clear to you by now.

```
e,1,0,t
o,1,2,o
c,2,o,1/8,0
r2,2,o,2
r1,2,0,1
```

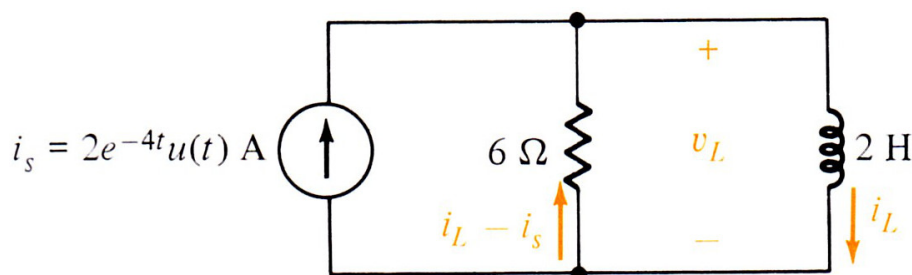
Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_c and i_c , in **Results**.

$$\{(1/2)(1 - e^{-4t}) - 2t, (1/4)(e^{-4t} - 1)\}$$

Bo2's Example 5.12 (Exponential)

Assume initial conditions zero. Find $i_L(t)$ and $v_L(t)$, given that $i_S(t) = 2e^{-4t}$ A.



Describe the source as given to you by the problem, using an exponential.

```
j,0,1,2e^(-4t)
r,0,1,6
l,1,0,2,0
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are *i1* and *v1*, in **Results**.

$$\{6e^{-3t} - 6e^{-4t}, 48e^{-4t} - 36e^{-3t}\}$$

Bo2's p265 (Exponential)

Repeat Bo2's Example 5.12 if the current source is now $i_s(t) = 2 e^{-3t}$ A.

All we do is change one digit in the circuit description: replace 4 with 3.

```
j,0,1,2e^(-3t)
r,0,1,6
l,1,0,2,0
```

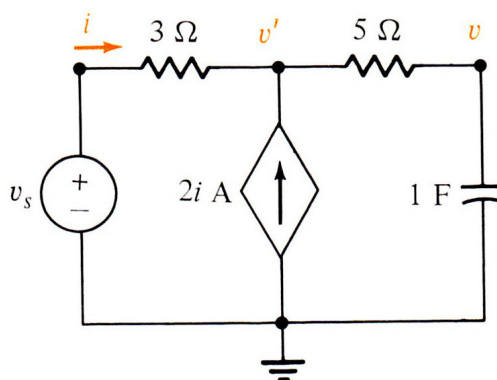
Set **Analysis** to *TR – transient / time domain*.

The answers you want are *i1* and *v1*, in **Results**.

$$\{6te^{-3t}, (12 - 36t)e^{-3t}\}$$

Bo2's Example 5.13 (Exponential)

Find the voltage drop in the capacitor and the current in the $3\ \Omega$ resistor, if the source value is $v_s(t) = 18e^{-t/2}$.



```
e,1,0,18e^(-t/2)
r3,1,2,3
r5,2,3,5
c,3,0,1,0
j,0,2,2*ir3
```

Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_c and i_{r3} , in **Results**.

$$\{9e^{(-t/6)} - 9e^{(-t/2)}, (3/2)e^{(-t/2)} - (1/2)e^{(-t/6)}\}$$

Bo2's Drill Exercise 5.16 (Exponential)

For the circuit of Bo2's Example 5.7, find $v_c(t)$ and $i_c(t)$ for the case when the source's value is $v_s(t) = 2e^{-4t}$. As before, assume initial conditions are zero.

```
e,1,0,2e^(-4t)
o,1,2,o
c,2,o,1/8,0
r2,2,o,2
r1,2,0,1
```

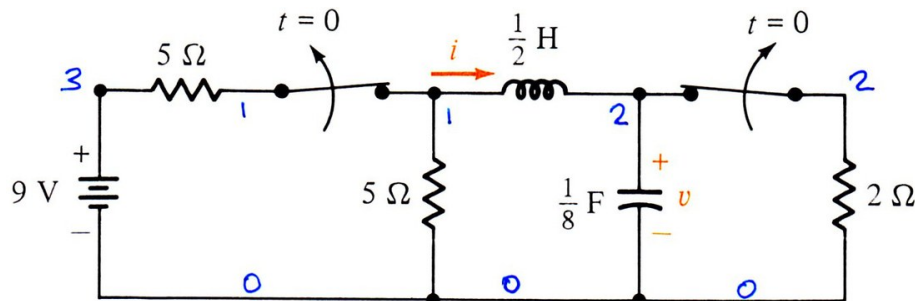
Set **Analysis** to *TR – transient / time domain*.

The answers you want are v_c and i_c , in **Results**.

$$\{-16te^{-4t}, (8t - 2)e^{-4t}\}$$

Bo2's Example 6.1

Find the current in the inductor $i(t)$ and the voltage drop in the capacitor $v(t)$.



Since we have switches changing in time $t=0$, this is a two interval problem. The first interval is for $t < 0$. We can analyze the first interval using DC.

```
e,3,0,9
r2,3,1,5
r1,1,0,5
l,1,2,1/2
c,2,0,1/8
r3,2,0,2
```

The answers you want are v_c and i_{l1} , in **Results**.

{2,1}

These are the initial conditions for the second interval. The second is for $t \geq 0$. We can analyze the second interval using TR. We only care about v_c and i_{l1} .

```
r1,1,0,5
l,1,2,1/2,1
c,2,0,1/8,2
```

$$\{4e^{-2t} - 2e^{-8t}, 2e^{-8t} - e^{-2t}\}$$

The plot tool

Imagine, for example, that in Bo2's Example 6.1 we are asked to plot $i(t)$ and $v(t)$ for times between 0 s and 1.5 s. The first step is to run the `plot` tool:

Open the **Plot** card, leave **Plot type** on *Plot a function of time (TR)*, and give it `vc` as the variable, 0 as the minimum and 1.5 as the maximum. The answers from the solve above are already in scope, so the name is all it needs. The graph of the capacitor's voltage drop between 0 and 1.5 seconds appears beneath.

Compare it to the graph given by the textbook. Repeat this procedure giving `il` as the function. Compare the resulting graph to that given by the textbook.

Bo2's Drill Exercise 6.1

For the circuit shown, find $v(t)$ and $i(t)$ for $t \geq 0$.

Nothing new here. This is a two interval problem, as others seen before. The first interval, for $t < 0$, is analyzed in DC to find the initial conditions.

```
j,0,1,1
r3,1,0,3
r6,1,0,6
l,1,2,1/4
c,2,0,1/3
```

The answers you want are `vc` and `il`, in **Results**.

{2,0}

These conditions are used in the TR analysis of the second interval, for $t \geq 0$.

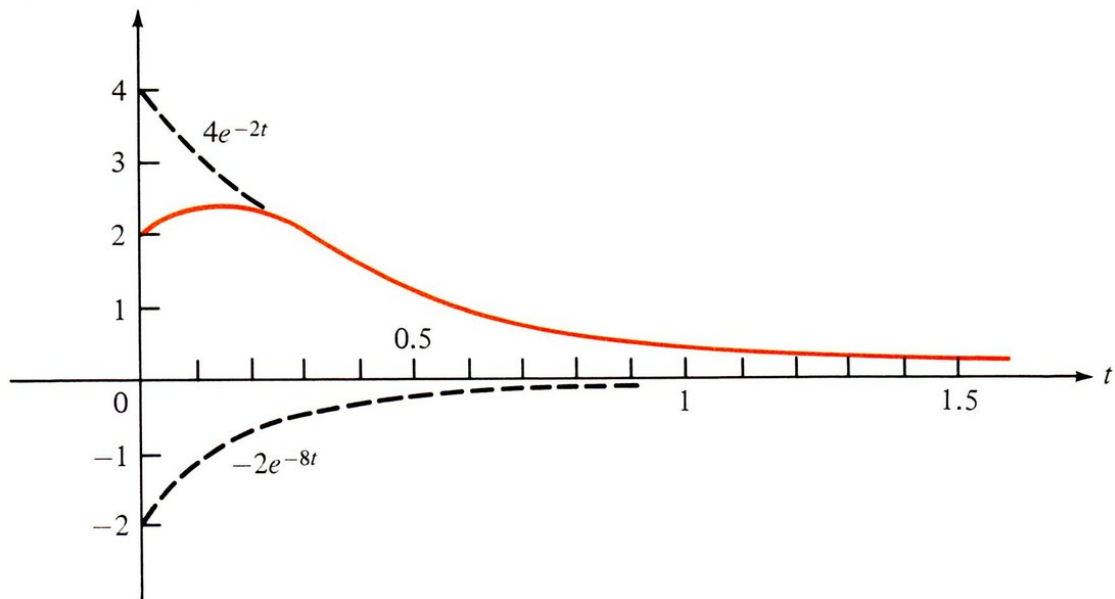
The answers are $vc = 3e^{-2t} - e^{-6t}$ and $il = -2e^{-2t} + 2e^{-6t}$, both in **Results**.

```
r3,1,0,3
r6,1,0,6
l,1,2,1/4,0
c,2,0,1/3,2
```

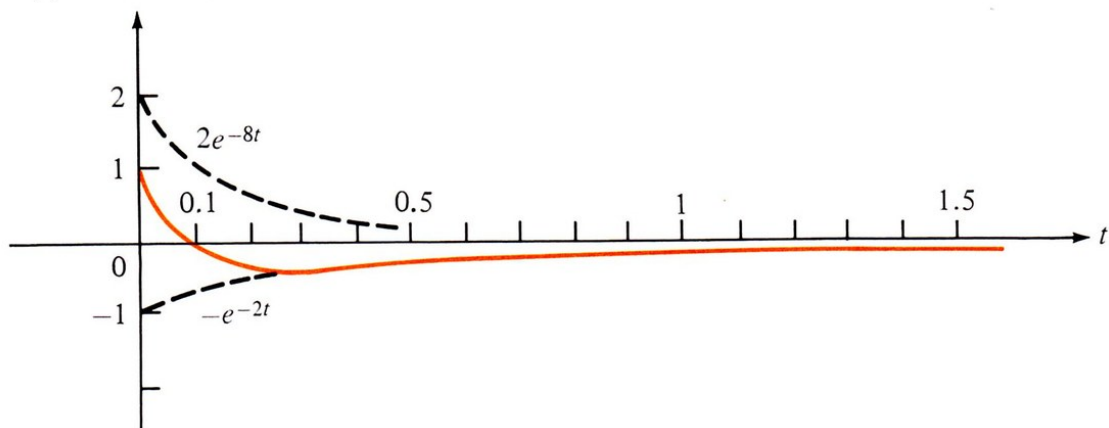
$$3e^{-2t} - e^{-6t}$$

$$2e^{-6t} - 2e^{-2t}$$

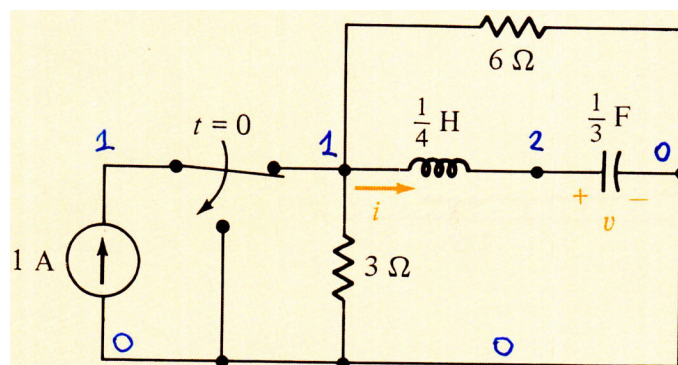
$$v(t) = 4e^{-2t} - 2e^{-8t}$$



$$i(t) = -e^{-2t} + 2e^{-8t}$$

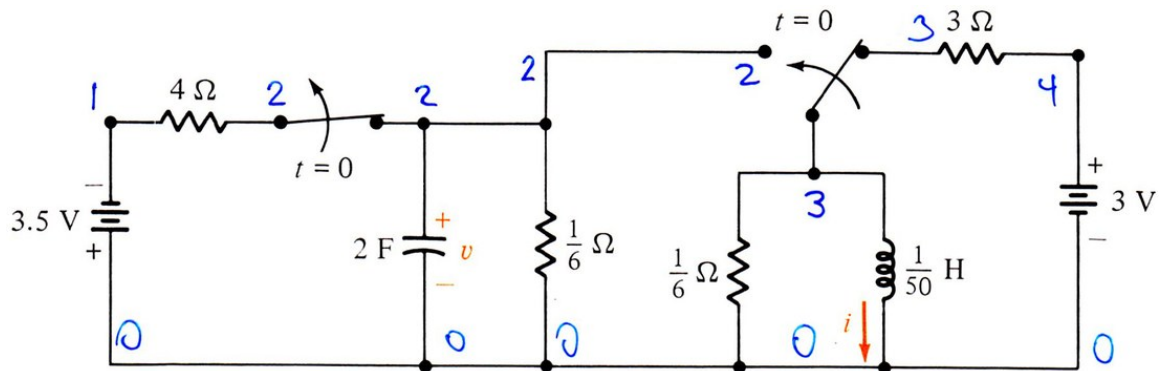


Practice Problems for Lesson 6



Bo2's Example 6.2

Find $i(t)$ and $v(t)$ for the circuit shown.



I decided to simulate this using only fractions, not decimals.

So write the value as the fraction it is: $7/2$ rather than 3.5 . Version 9 needs no conversion step — a fraction stays exact, and it is the decimal point that would make the arithmetic approximate.

$7/2$

Because of the switches, this is a two interval problem. The first interval is to be analyzed using DC. Notice that for $t < 0$ the circuit is such that we basically have two separate circuits. We simulate each one of them separately:

```
e,0,1,7/2
r1,1,2,4
c,2,0,2
r2,2,0,1/6
```

The answer you want is v_c , in **Results**.

$-7/50$

```
r3,3,0,1/6
l,3,0,1/50
r4,3,4,3
e2,4,0,3
```

The answer you want is i_l , in **Results**.

1

The second interval, for $t \geq 0$, is analyzed using TR. I could have described the circuit again from scratch, but out of laziness I copied the descriptions from the DC simulations. To avoid renaming nodes, I simulated the right switch as a short between nodes 2 and 3.

v_c comes back $\frac{(-\sin(4t) - 7\cos(4t))e^{-3t}}{50}$ and $i_l (-\sin(4t) + \cos(4t))e^{-3t}$.

```
c,1,0,2,-7/50
r,1,0,[1/6,1/6]
l,1,0,1/50,1
```

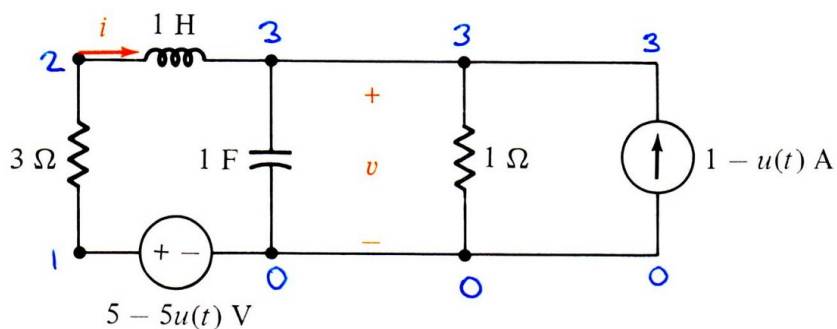

$$(-7/50)e^{-3t} \cos(4t) - (1/50)e^{-3t} \sin(4t)$$

$$e^{-3t} \cos(4t) - e^{-3t} \sin(4t)$$

These are the right answers.

Bo2's Example 6.3

Find $i(t)$ and $v(t)$ for the circuit shown.



Since the sources change value, this is a two interval problem. For $t < 0$, in DC:

```
e,1,0,5
r3,1,2,3
l,2,3,1
c,3,0,1
r1,3,0,1
j,0,3,1
```

The answers you want are v_c and i_l , in **Results**.

{2,1}

Using these as initial conditions, now we use TR for $t \geq 0$:

```
r3,0,2,3
l,2,3,1,1
c,3,0,1,2
r1,3,0,1
```

$$\{(3t + 2)e^{-2t}, (1 - 3t)e^{-2t}\}$$

Is the textbook wrong?

Anybody who has written a textbook, or read one for that matter, can attest that errors in the problems or the answers come with the territory. (I am sure this book has many.) We are all only human. But errors in an engineering textbook can be *very* frustrating to a student: confusion and hair-pulling ensue. If you think you have found one in your circuit analysis textbook, Symbolator can help you confirm whether your guess is right.

Below is an example of using Symbulator to confirm an error in a textbook.

Bo2's Drill Exercise 6.4

DRILL EXERCISE 6.4

For the *RLC* circuit shown in Fig. DE6.4, suppose that $v_s(t) = 2 - 2u(t)$ V. Find $i(t)$ and $v(t)$ for $t \geq 0$.

Answer: $8te^{-4t} + 2e^{-4t}$ A; $-16te^{-4t}$ V

Fig. DE6.4

My copy of Bo2 is second-hand, and this problem had scribbles on it. The previous owner evidently struggled with it and concluded that the inductor had to be $\frac{1}{2}$ H, not the 1 H the schematic says. Symbulator can show he is right and the book wrong. First, solve the circuit as printed, with a 1 H inductor.

```
e,2,0,2
r,1,2,1
l,1,0,1
c,1,0,1/8
```

The answers you want are i_l and v_c , in **Results**.

{2,0}

```
r,1,0,1
l,1,0,1,2
c,1,0,1/8,0
```

The answer we get doesn't match the answer provided by the book. Now let's repeat the TR simulation using the $\frac{1}{2}$ H value for the inductor:

```
r,1,0,1
l,1,0,1/2,2
c,1,0,1/8,0
```

Now the answer matches the book's. An expert user can reach for the `fd` gate and the `t2s` tool, which we have not discussed yet, to find the inductor's value from the book's answer. This is how I found the $\frac{1}{2}$ H myself, working backwards:

```
t2s(-16*t*e^(-4*t))
```

which gives $-\frac{16}{(s+4)^2}$.

$-16/(s+4)^2$

$$2*(s+8)/(s+4)^2$$

```
r,1,0,1
l,1,0,1,2
c,1,0,1/8,0
```

Set **Analysis** to *FD – complex frequency domain*.

The answer you want is v_c , in **Results**.

$$-16*1/(1*s^2+8*1*s+8)$$

That is what the **Solve** card is for – a system that is not a circuit:

$$-16/(s+4)^2 = -16*1/(1*s^2+8*1*s+8)$$

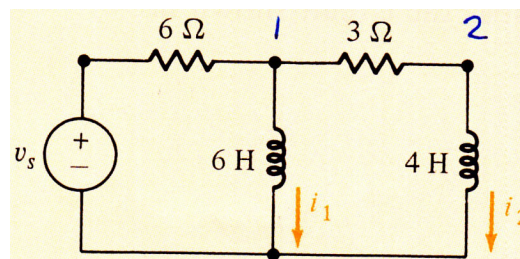
with 1 as the unknown. It answers $l = 1/2$ H.

$$l=1/2$$

The `fd` gate and `t2s` tool are used for frequency domain analysis, and will be discussed separately in a subsequent part of Symulator's documentation.

Bo2's Drill Exercise 6.5

In the circuit, suppose that $v_s(t) = 12 - 12 u(t)$ V. Find $i_2(t)$ for $t \geq 0$.



```
e,3,0,12
r6,3,1,6
l1,1,0,6
r3,1,2,3
l2,2,0,4
```

The answers you want are i_{l1} and i_{l2} , in **Results**.

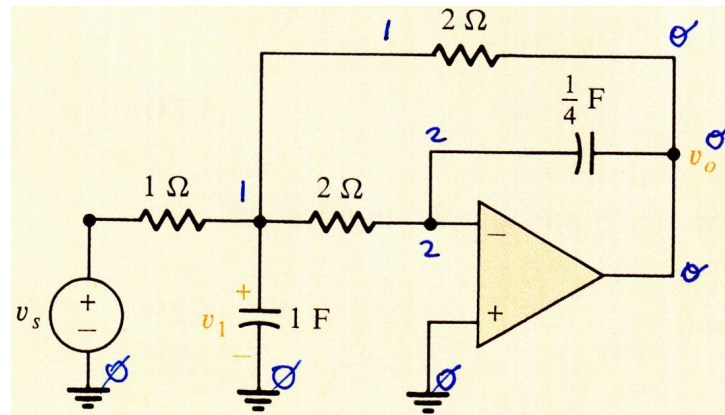
$$\{2,0\}$$

```
r6,0,1,6
l1,1,0,6,2
r3,1,2,3
l2,2,0,4,0
```

$$(12/11)(e^{-3t} - e^{-t/4})$$

Bo2's Drill Exercise 6.6 (Op Amp)

In the circuit, suppose that $v_s(t) = 2 - 2u(t)$ V. Find $v_o(t)$ for $t \geq 0$.



Simulate the first interval in DC to find the initial conditions of the capacitors.

```
e,3,0,2
r1,3,1,1
r2,1,2,2
r3,1,o,2
ca,1,0,1
cb,2,o,1/4
o,0,2,o
```

The answers you want are v_{ca} and v_{cb} , in **Results**.

{0,4}

Then simulate the second interval in TR, to find the voltage in node o .

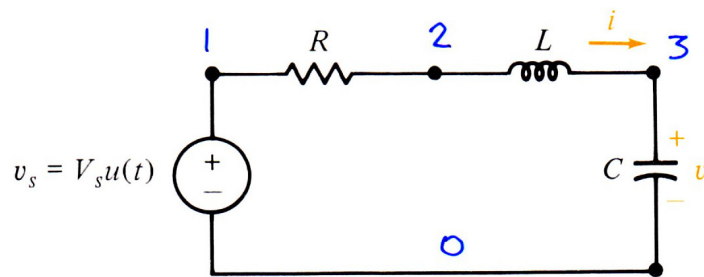
$$(-4t - 4)e^{-t}$$

```
r1,0,1,1
r2,1,2,2
r3,1,o,2
ca,1,0,1,0
cb,2,o,1/4,4
o,0,2,o
```

Bo2's Example 6.5 (Plot)

In the circuit, use $v_s = 2/5$ V, $R = 12\Omega$, $L = 2$ H and $C = 1/50$ F. Find $v(t)$ and $i(t)$, and plot them for time $0 < t < 1.5$ seconds.

```
e,1,0,2/5*u(t)
r,1,2,12
l,2,3,2,0
c,3,0,1/50,0
```



$$-(2/5)e^{-3t}\cos(4t)-(3/10)e^{-3t}\sin(4t)+2/5$$

$$(1/20)e^{-3t}\sin(4t)$$

We look in the results and see that v_c is the first of those and i_l the second.

These are the right answers. To plot them, run the `plot` tool, thus: `s\plot()`. Once the plot window opens, enter v_c as function, 0 as minimal time and 1.5 as maximal time. Compare to the graph from the book. Repeat the plot for i_l , and compare that one to the book's graph too.

Bo2's Drill Exercise 6.7

For the circuit in Bo2's Example 6.5, use $v_s=3V$, $R=5\Omega$, $L=1/2H$ and $C=1/8F$. Find $v(t)$ and $i(t)$.

```
e,1,0,3*u(t)
r,1,2,5
l,2,3,1/2,0
c,3,0,1/8,0
```

$$\{-4e^{-2t} + e^{-8t} + 3, e^{-2t} - e^{-8t}\}$$

Bo2's Drill Exercise 6.8 (Ramp)

For the circuit in Bo2's Drill Exercise 6.7, find the voltage drop in the capacitor $v(t)$ if the source has a value $v_s(t) = 3t$.

```
e,1,0,3t
r,1,2,5
l,2,3,1/2,0
c,3,0,1/8,0
```

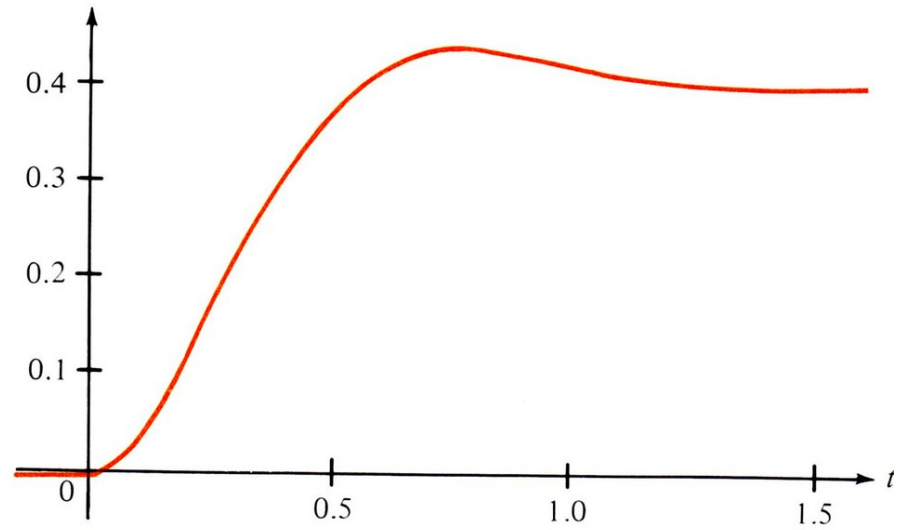
$$2e^{-2t} - (e^{-8t})/8 + 3t - 15/8$$

Bo2's Example 6.6 (Plot)

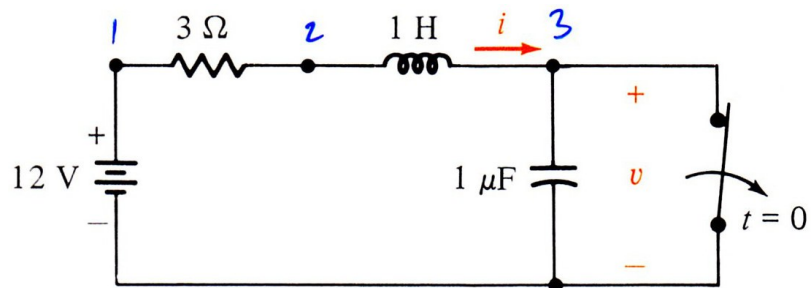
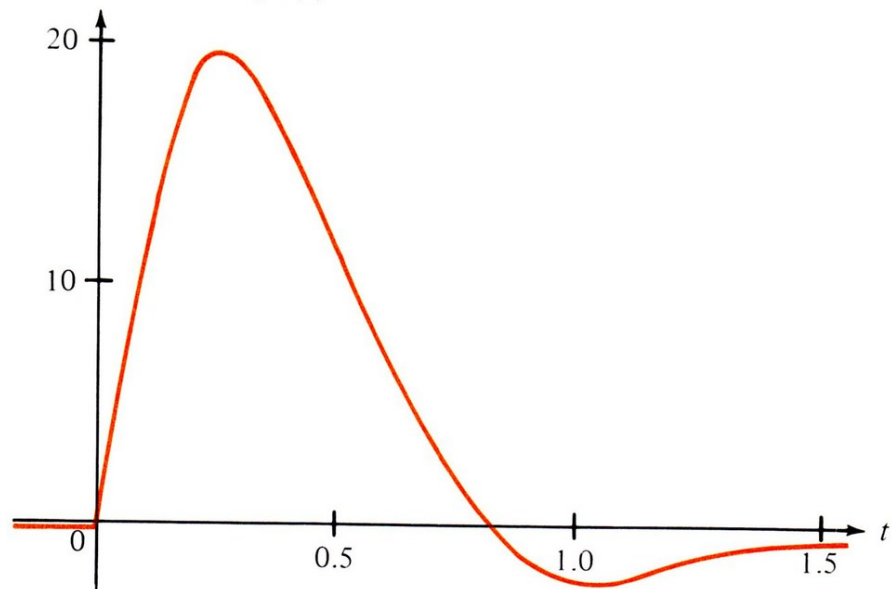
For the circuit, plot the voltage drop in the inductor for $0 < t < 5ms$.

First, find the initial conditions for $t < 0$ by running a DC simulation.

$$v(t) = [0.4 - 0.5 e^{-3t} \cos(4t - 0.64)] u(t)$$



$$i(t) = [0.05 e^{-3t} \sin 4t] u(t)$$



e,1,0,12
r,1,2,3
l,2,3,1
c,3,0,1'μ

```
s1,3,0
```

The answers you want are i_L and v_C , in **Results**.

{4,0}

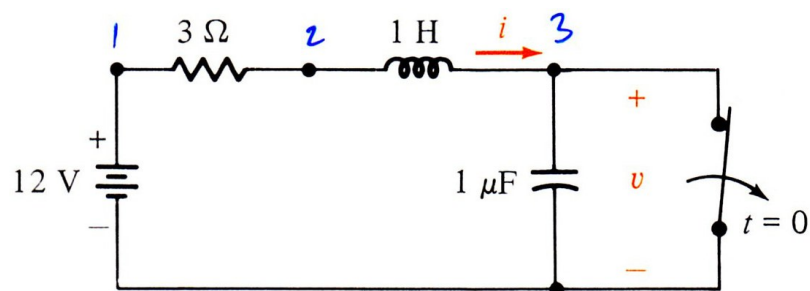
Then, find the v_L for $t \geq 0$ by running a TR simulation.

```
e,1,0,12*u(t)
r,1,2,3
l,2,3,1,4
c,3,0,1'μ,0
```

Finally, run `s\plot()` and enter v_L as function, 0 as minimal time and 0.005 as maximal time. In the resulting plot you will see something outstanding: around $t = 1.57$ ms, the voltage drop across the inductor is 3991 volts!

Bo2's Drill Exercise 6.9

For the circuit, $R=1\Omega$, $L=2H$, $C=1/2F$ and $i_S(t)=u(t)$. Find $i(t)$ and $v(t)$.



```
j,0,1,u(t)
r,1,0,1
l,1,0,2,0
c,1,0,1/2,0
```

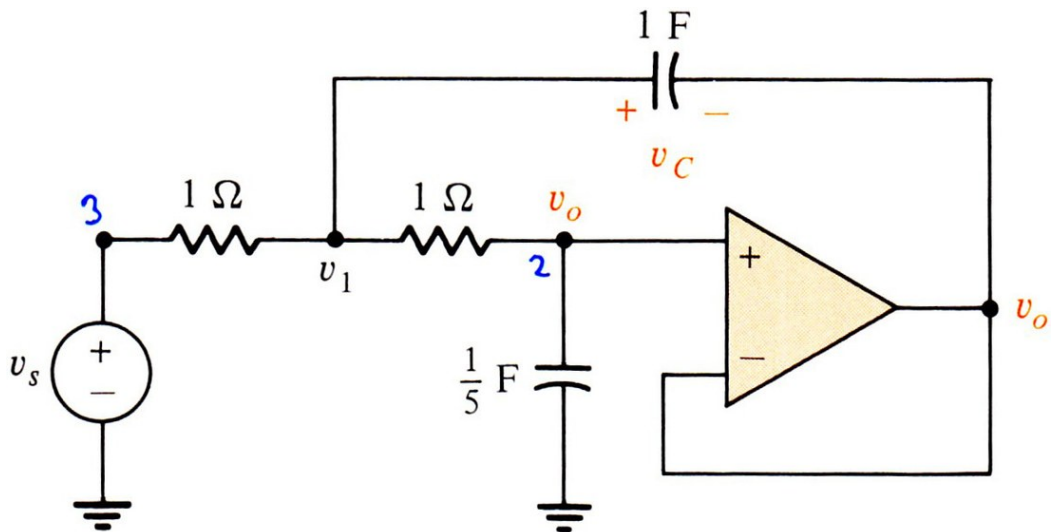
$$\{(-t-1)e^{-t} + 1, 2te^{-t}\}$$

Bo2's Figure 6.23 (Op Amp)

For the circuit, find $v_o(t)$ if $v_S(t) = u(t)$.

```
e,3,0,u(t)
r1,3,1,1
r2,1,2,1
ca,2,0,1/5,0
cb,1,0,1,0
o,2,0,0
```

$$1 - e^{-t} \cos(2t) - (1/2)e^{-t} \sin(2t)$$



Bo2's Drill Exercise 6.10 (Op Amp)

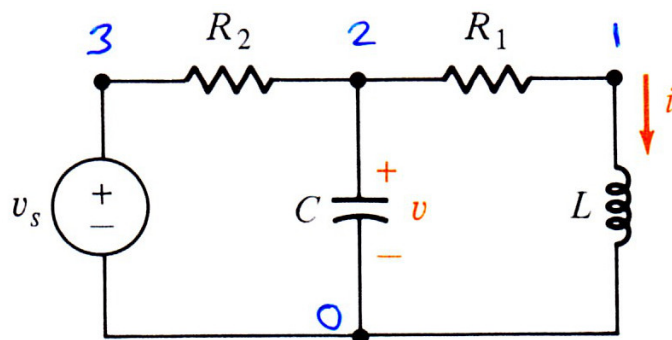
In the previous circuit, change the capacitor's value from $1/5$ F to $25/16$ F. Find $v_o(t)$ if $v_s(t) = 3 u(t)$.

```
e,3,0,3*u(t)
r1,3,1,1
r2,1,2,1
ca,2,0,25/16,0
cb,1,0,1,0
o,2,0,0
```

$$-4e^{-2t/5} + e^{-8t/5} + 3$$

Bo2's Drill Exercise 6.11 (p307)

For the circuit, suppose that $R_1 = R_2 = 1\Omega$, $L = 1\text{H}$, $C = 1\text{F}$ and $v_s(t) = 2e^{-2t} u(t)$. Find $i(t)$ if all initial conditions are zero.



```
e,3,0,2e^(-2t)
r2,3,2,1
r1,2,1,1
c,2,0,1,0
```

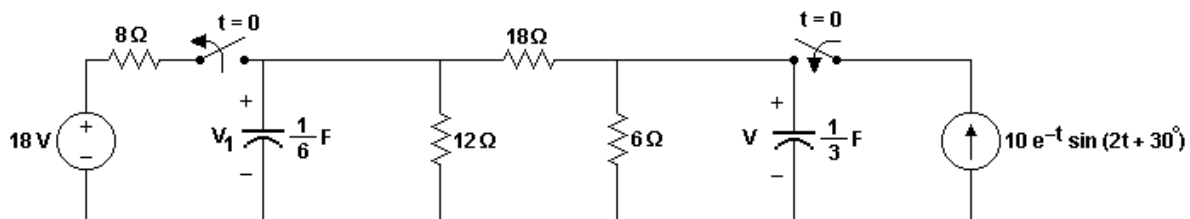


```
1,1,0,1,0
```

$$-(e^t \cos(t) - e^t \sin(t) - 1) e^{-2t}$$

A more complex problem

The circuit comes from the circuits analysis class of Professor Eliane Boulet de Cabrera, from Universidad Tecnológica de Panama. Find v_1 and v .



Because of the switches, this is a two interval problem. The first is run in DC.

```
e,3,0,18
r8,3,1,8
ca,1,0,1/6
r12,1,0,12
r18,1,2,18
r6,2,0,6
cb,2,0,1/3
```

The answers you want are v_{ca} and v_{cb} , in **Results**.

{9,9/4}

These serve as initial conditions for the second interval, which is run in TR. Since we only want v_1 and v_2 , it pays off to let Symbolator know this:

```
ca,1,0,1/6,9
r12,1,0,12
r18,1,2,18
r6,2,0,6
cb,2,0,1/3,9/4
j,0,2,10e^{(-t)}*sin(2t+30*pi/180)
```

Set **Analysis** to *TR – transient / time domain*.

$$\begin{aligned} & (53/17 - 20/17)e^{-t} \cos(2t) \\ & + (-203/17 - 5/17)e^{-t} \sin(2t) \\ & + (803/17 + 193/34)e^{-t/2} + (9/2 - 53)e^{-t} \end{aligned}$$

$$\begin{aligned} & (-2453/34 - 20/17)e^{-t} \cos(2t) \\ & + (245/34 - 203/17)e^{-t} \sin(2t) \\ & + (803/17 + 193/34)e^{-t/2} + (53/2 - 9/4)e^{-t} \end{aligned}$$

Version 9 answers this one in about a second. v_1 and v_2 are in **Results**; they are long enough that they are easier read on screen than in print.

These are the correct answers, which explains why Professor Boulet – who made the problem up – is a living legend among circuit students at UTP.

Advanced use of Expert in TR

The **expert** tool can be very useful in transient analysis. Using it, however, requires some knowledge. Here's two things you need to know in order to use expert mode like a boss:

First, when Symbolator solves a problem using TR, it follows these general steps:

- Move every independent source into the frequency domain, and generate a set of equations and unknowns for the system there.
- Solve these frequency domain equations.
- Convert the answers back to the time domain.

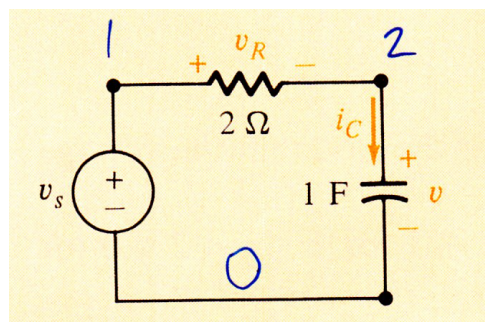
Second, when Symbolator solves a problem using the expert tool, it freezes this process halfway between steps 1 and 2, so that you can tinker with the equations and unknowns before they are solved.

That is where version 9 differs, and it is the more comfortable of the two. The equations are still solved in the frequency domain – that has not changed, and it is why initial conditions and impulses behave as they do – but what you *type* is read in the **time domain**, and converted for you on the way in. Write the answer you know the way you would write it on paper.

The rule covers everything you add: equations, conditions and the expressions inside them. A relation between plain parameters, such as $x = 3$, is left alone – it fixes a symbol in the circuit rather than describing a signal, and there is nothing to transform.

Bo2's Drill Exercise 4.5 (Expert)

For this circuit, we know that the capacitor's initial condition is zero, that v_s is an unknown step source (i.e. of the form $A u(t)$, where A is a constant value in volts) and that the voltage drop in the capacitor for $t > 0$ is found to be $1 - e^{-(t/2)}$. Find the voltage drop in the resistor, the current through the capacitor and the value of the source.



This problem is a match made in heaven for the **expert** tool, because we have one unknown value in the circuit (i.e. the value of the step source) and we have one known

answer (i.e. the voltage drop in the capacitor.) So, the game plan here is to run this circuit through Symbulator's expert mode, add one new equation and one new unknown, and then solve. First, let's generate the new equation.

The non-expert user is right here: the new equation is simply $1 - e^{(-t/2)} = v_c$. Everything you type into expert mode in version 9 — equations, conditions, expressions — is read in the **time domain**, the same domain as the answers on screen. You do not convert, and you do not have to remember which side of the transform you are standing on.

NOTE

The calculator does not offer this at all

Expert mode on the calculator asks *Analysis? 1:DC 2:AC 3:FD* — TR is not among the choices, which is how it avoided the question. Version 9 keeps expert mode for TR and answers the question instead: everything is in time.

If you would rather convert by hand, **t2s** is still there and an equation already written in **s** is left alone rather than transformed twice:

```
t2s(1-e^(-t/2))
```

which gives $\frac{1}{s(2s+1)}$ — the same statement, one domain over.

We have our new equation. Copy this equation into the clipboard, since we will want to paste it in the Expert window. Now let's run the Expert simulation of the circuit. Let's define the value of the source as $a \cdot u(t)$, since we know it's a step source; the variable a will serve as the unknown value, for which we will solve in the Expert mode.

```
e,1,0,a*u(t)
r,1,2,2
c,2,0,1,0
```

Choose TR, then open **Expert Mode**. **Add equations** takes one per line, so there is no **and** to prefix — the word exists on the calculator only because the equation is being appended to a list:

```
1-e^(-t/2) = vc
```

Add unknowns likewise takes the name on its own, with no leading comma:

```
a
```

Let Symbulator solve the equations now. Once it is done, we want to do a quick sanity check. If the system solved correctly, we should see that Symbulator has as voltage drop in the capacitor the expression we already know from the problem statement:

v_c in **Results** reads $1 - e^{-\frac{t}{2}}$, which it does.

This is correct, and the answers we want are on screen with it: i_c is $\frac{e^{-\frac{t}{2}}}{2}$, and the resistor's **voltage drop** line gives $e^{-\frac{t}{2}}$.

These are the correct answers. The value of the step source is 1 volt.

Bo2's Drill Exercise 4.13 (Expert)

For the circuit shown in Fig. DE4.13, the voltage across the $1\text{-}\Omega$ resistor is $v_R(t) = e^{-t}$ V for $t \geq 0$. Given that $v_C(0) = 0$ V, find the following functions for $t \geq 0$: (a) $i_R(t)$; (b) $v_C(t)$; and (c) $v_s(t)$.

Answer: (a) e^{-t} A; (b) $1 - e^{-t}$ V; (c) 1 V

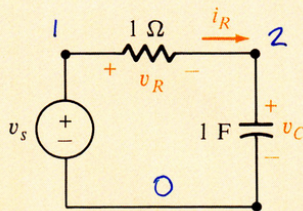


Fig. DE4.13

This problem is solved just like the one we already saw. First, find the new equation:

No transform this time either — write what you know:

$$v_R = e^{-t}$$

Then run the Expert simulation. I decided to use v_s as the step value of the source:

```
e,1,0,vs*u(t)
r,1,2,1
c,2,0,1,0
```

Choose TR, and put v_s in **Add unknowns** beside the equation above.

Run it. The sanity check is already on screen: v_R reads e^{-t} , which is what we told it.

And the answers with it: i_R is e^{-t} and the source's value v_s is **1** V.

Ain't that a beauty? Yes, ma'am! Atta boy, Symulator! Atta boy!

Bo2's 164 (Expert)

This is the same thing as the previous two, so there is no need for commentary.

```
e,1,0,vs*u(t)
r,1,2,2
l,2,0,1,0
```

Choose TR, put v_s in **Add unknowns**, and give **Add equations** the answer you know — in time, as always here:

$$1 - e^{-2t} = i_L$$

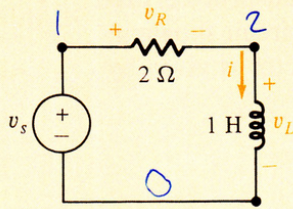
Run it, then put the sanity check into **Evaluate**:

For the circuit shown in Fig. DE4.2, the current through the inductor is

$$i(t) = \begin{cases} 0 & \text{for } -\infty < t < 0 \\ 1 - e^{-2t} & \text{for } 0 \leq t < \infty \end{cases}$$

Find, for $t \geq 0$, (a) $v_L(t)$, (b) $v_R(t)$, (c) $v_s(t)$, and (d) the power absorbed by the inductor.

Answer: (a) $2e^{-2t}$ V; (b) $2(1 - e^{-2t})$ V; (c) 2 V; (d) $2(e^{-2t} - e^{-4t})$ W



It does add up — i_L reads $1 - e^{-2t}$ — and the answers are on screen with it: v_L is $2e^{-2t}$, v_R is $2 - 2e^{-2t}$, and the source's value v_s is 2 V.

This is the end of the TR section.

Alternating current analysis

Learn how to run an *alternating current* analysis (via phasor analysis) using **ac**. Learn how to describe capacitors, inductors, *impedances and admittances*, and sources for AC analysis.

In this lesson you will learn how to use the **ac** analysis to solve AC circuits in Symbulator. You will also learn how to describe the elements you already know so they suit AC analysis, and how to use **er** and **th** in their AC mode.

7.1 AC analysis in Symbulator

The main thing to know is that in AC mode Symbulator understands phasors — complex numbers. It uses them in the input, in the analysis and in the output.

⚠ CAUTION

Symbulator does not understand sinusoids

You should not feed Symbulator any sinusoids, or expect any sinusoid answers from it. Any conversion to and from sinusoids has to be done by you.

The AC analysis

Version 9 keeps AC on the same menu as the other analyses: set **Analysis** to *AC — alternating current*. Unlike DC and TR it needs one more input — the frequency of the circuit in radians per second, typed into the ω — **angular frequency** box that appears beside the menu.

AC mode in the other tools

The equivalence tools work in AC the same way: choose *Find equivalent* as usual, set **Analysis** to *AC — alternating current*, and give the ω box a frequency where the circuit needs one.

Describing elements for AC

When describing a circuit for AC analysis, element values have to be written the way this analysis expects. Valueless elements, like the short circuit and the op amp, are unchanged. Here's how to describe the ones with values:

- Current sources **j** and voltage sources **e** accept complex values. You can declare their

value in rectangular form, e.g. $10-3i$, or in angular form, e.g. $(100\angle120^\circ)$.

- The element **r** no longer represents just a resistor: it now represents an **impedance**. If you are wondering why we didn't use the letter **z**, wait until we get to two-ports. In AC analysis, **r** accepts complex numbers as its value, such as $10-3i$. We use **r** to describe resistors (real values) and conductances (real values, inverted), but also impedances (complex values) and admittances (the inverse of their complex value).
- The elements **c** and **l** are used to describe capacitors and inductors *only* when their values are given in farads and henries, respectively. For AC analysis their description no longer requires the fifth field for the initial condition, since alternating current analysis focuses on the steady state after transient effects are long gone. So we only use the name, the two nodes, and the value.

WARNING

If it's in Ω , it's an **r**

When the problem gives us the values of capacitors and inductors in ohms, we **must** describe them as impedances. If a capacitor has its value in farads, describe it with **c**. If an inductor has its value in henries, describe it with **l**. But anything that has its value in ohms — resistor, impedance, capacitor or inductor — must be described using **r**.

The values of all of these elements can use SI prefixes.

What answers do you get

You get the same answers you did in DC, except that now they are phasors. The one difference is that now, besides the real power consumed, you also get the complex power consumed. We will discuss this in Lesson 8 (page 193).

Rectangular or polar

A phasor can be written two ways, and circuit problems use both: in *rectangular* form as $3 + 4j$, or in *polar* form as an amplitude and an angle, $5\angle53.13^\circ$. They are the same number. Which one you want depends on the question — rectangular adds and subtracts easily, polar multiplies and divides easily, and textbook answers are usually quoted in polar.

Symulator answers in rectangular form unless you ask otherwise. In **Settings**, under *Display*, tick **Show AC answers as polar phasors** and every answer is shown as an amplitude and an angle instead. The circuit is re-solved as soon as you tick it, so you can switch back and forth on a solved circuit and watch the same answers change form.

Three things it deliberately leaves alone, and they are worth knowing so the setting does not look broken:

- **Average power stays a plain number.** It is a real quantity, not a phasor, so an angle would be meaningless — 1234 W is 1234 W. Complex power *is* converted, because its polar form is the apparent power and the power-factor angle, which is usually what you want. See Lesson 8 (page 193).
- **Symbolic answers are left as they are**, exactly as SI prefixes leave them. There is

no angle to take of $v_{in}r_b/(r_a + r_b)$.

- **It applies to AC only.** In DC every answer is real, and in the other two analyses the answers are functions of s or t , so there is nothing to take the angle of. The checkbox greys out there and tells you why.

An answer that happens to be real still gets its angle, of 0° or 180° . That is deliberate: it is what the **aa** tool does, and it means a purely resistive circuit does not look as though the setting failed.

 TIP

It combines with SI prefixes

Ticking both gives you the amplitude with a prefix and the angle in plain degrees — $632.5 \text{ m}\angle 108.4^\circ$. The angle is never prefixed, which would be nonsense.

One consequence to expect: a polar answer is always a decimal, even under *exact*. An angle in degrees is a measurement rather than a closed form, so there is nothing exact to preserve. This is the same trade SI prefixes make.

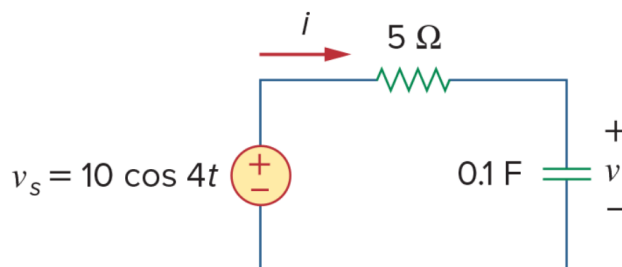
The **aa** tool has not gone anywhere, and it is still the better choice when you want one value converted — or when you want the amplitude and angle of something that is not an answer by itself, such as a difference between two node voltages. Lesson 9 (page 200) uses it that way throughout.

7.2 Solved numerical examples

With values in F and H

AS7's Example 9.9

Find $v(t)$ and $i(t)$.



AS7's Example 9.9

SOLUTION

Since this is the first AC problem we will solve, I will explain every step, including the manual ones, in full detail.

Step 1: sinusoids to phasors. We have to express the value of the source in terms Symbulator can understand, so we convert it from a sinusoid to a phasor: $10\angle 0^\circ$. We can pass this as $(10\angle 0^\circ)$ or, since the angle is zero, simply as 10.

Step 2: note the frequency. The circuit has a capacitor with a value given in farads. Because of this, Symbolator will need the frequency, given by the problem as 4 rad/s, to convert that value into an impedance in ohms.

Step 3: describe the circuit.

```
e,1,0,10  
r1,1,2,5  
c,2,0,.1
```

Choose *AC* — *alternating current*, and put **4** in the ω — **angular frequency** box that appears beside it.

Notice that, in our circuit description, the order we give to the nodes of the resistor and the capacitor is chosen to be convenient for the answers we will ask for.

Step 4: read the answers. Once Symbolator is done, you should take a look at what it found:

- the usual voltages in the nodes, voltage drops in the elements, and currents through the elements, in variables that should be familiar by now
- the *average power consumed* in the source and the resistor, in `ape` and `apr1`. None is given for the capacitor, since capacitors and inductors do not consume real power.
- the *complex power consumed* in all elements, in `sc`, `se` and `sr1`
- the *equivalent impedance* of the rest of the circuit as seen by the source, in `ze`

To get the answers we need for this problem in particular, we ask for `ir1` and `vc`. It is likely that the computer will give you the answers in rectangular form, with a real part and an imaginary part.

If we want to see them as an amplitude and angle, open the **Mini-Tools** card under the results, leave the tool set to *aa* — *amplitude and angle*, and give it the answer's name:

```
ir1
```

It reads **1.789** $\angle$ **26.57°** A.

That is the one-value way of doing it. If you would rather see *every* answer in polar form, there is a setting for it — see *Rectangular or polar*, above.

This is correct. You manually convert it to a sinusoid, by putting it back in the same terms the input was given in:

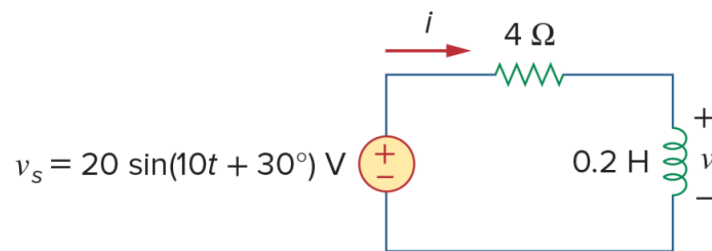
$$i(t) = 1.789 \cos(4t + 26.57^\circ) \text{ A}$$

Asking for the capacitor voltage the same way gives $4.472 \angle -63.43^\circ$, that is, $v(t) = 4.472 \cos(4t - 63.43^\circ) \text{ V}$, which is correct.

Future solved problems will not include this level of detail in the solution, only the circuit description and the commands we give.

AS7's Practice Problem 9.9

Determine $v(t)$ and $i(t)$.



AS7's Practice Problem 9.9

SOLUTION

```
e,1,0,(20∠30°)
r1,1,2,4
l,2,0,.2
```

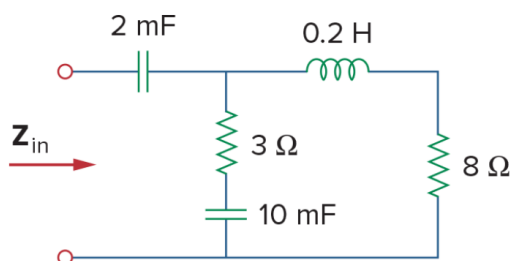
The source is given in polar form, and version 9 reads it as written — magnitude, the angle sign, then the angle in degrees.

Then AC, with **10** for ω . **Mini-Tools** with *aa* reads *ir1* as **4.472∠3.43°** A and *v1* as **8.944∠93.43°** V. Both are correct.

We ask for the current in the resistor and get 4.472/3.43°, and for the voltage in the inductor and get 8.944/93.43°. Both are correct.

AS7's Example 9.10

Find the input impedance of the circuit. Assume that the circuit operates at $\omega = 50$ rad/s.



AS7's Example 9.10

SOLUTION

Since this is a passive circuit, to reduce it we will use the **er** tool.

```
ca,1,2,2'm
r1,2,3,3
cb,3,0,10'm
l1,2,4,.2
r2,4,0,8
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Resistance / impedance*, with nodes **1** and **0**, in AC at ω **50**. The answer is called z_{eq} .

3.22-**i**11.07

That is $3.22 - j11.07 \Omega$, which is correct.



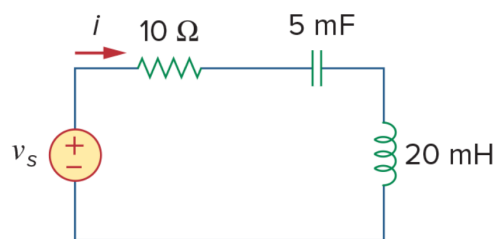
TIP

The pr tool only reduces impedances

In case you were tempted to reduce this circuit by hand instead of simulating it, remember that parallel reduction works on resistors and impedances only. You would need to convert the capacitors and the inductor to impedances in ohms before you do that.

AS7's Problem 9.35

Find the steady-state current i in the circuit when $v_s(t) = 50 \cos 200t$ V.



AS7's Problem 9.35

SOLUTION

The capacitor is in farads and the inductor in henries, so Symbulator needs the frequency, and the source is entered as its phasor: $50 \cos 200t$ is 50 at an angular frequency of 200.

```
e,1,0,50
r,1,2,10
c,2,3,5'm
l,3,0,20'm
```

AC, with ω — **angular frequency** set to **200**.

Read it with **Mini-Tools** set to **aa**: aa(ir) gives **4.789** $\angle$ **-16.70**° A.

That is 4.789 A at an angle of -16.7° , which is correct.



TIP

Why the resistor is r on one calculator and r1 on the other

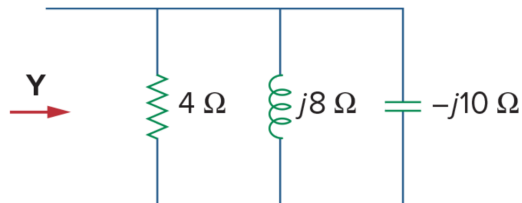
The version 7 description above names its resistor r ; the version 8 one cannot, because the Nspire reserves r (and rx), as Lesson 1 (page 9) notes — $r1$ is the usual fix there. Version 9 reserves neither, and its panel keeps the plain r .

With values in Ω only

When the values of all the capacitors and inductors in the circuit are given in imaginary ohms, they must be entered in the circuit description as impedances, **r**. In these cases, Symbulator will not need a frequency.

AS7's Problem 9.37

Determine the admittance Y for the circuit.



AS7's Problem 9.37

SOLUTION

Since the values are in ohms, these are all impedances. This is a passive circuit, so we could use the `er` tool. However, the structure of this circuit is so simple that we can reduce it by hand. The problem asks for the equivalent admittance, so our answer will be the inverse of the equivalent impedance:

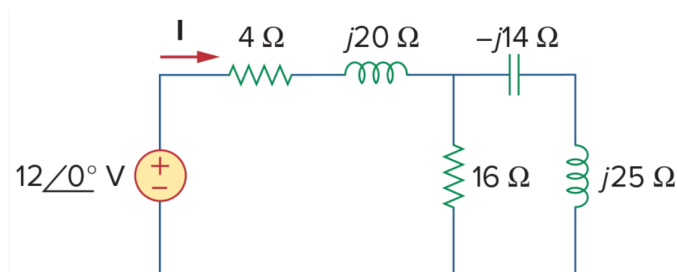
```
1/pr(4,8j,-10j)
```

```
0.25-0.025i
```

That is $0.25 - j0.025$ S, which is correct.

AS7's Problem 9.39

For the circuit shown, find the equivalent impedance, and use that to find the current I . Let $\omega = 10$ rad/s.



AS7's Problem 9.39

SOLUTION

Notice that the question gives you a frequency and that it is entirely superfluous: every value is already in ohms, so there is nothing for a frequency to convert. You will not need it.

Notice also that nothing here asks for a full simulation. The equivalent impedance falls out of one expression, using the shorthand for parallel combination:

```
4+20j+pr(16,-14j+25j)
```

```
9.135+i27.47
```

That is $9.135 + j27.47 \Omega$. The current is the source voltage divided by it:

There is no stored z_{eq} to divide by, so put the whole thing in **Mini-Tools** with **aa**: `aa(12/(4+20j+pr(16,-14j+25j)))` gives **0.4145** $\angle$ **-71.60°** A.

That is 414.5 mA at an angle of -71.6° , which is correct.



TIP

The same answer from a simulation

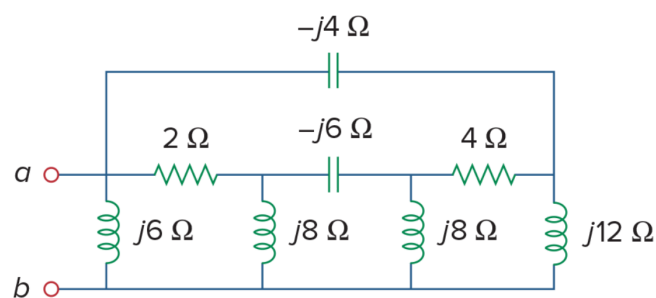
You can also let Symbolator do the reduction, using the square-bracket shorthand for parallel elements inside a value:

```
e,1,0,12
r1,1,0,4+20j+[16,-14j+25j]
```

No frequency is needed to solve this, but AC asks for one anyway — put in anything you like, because every value is already an impedance and none of them depends on it. Then `aa(ir1)` gives the same **0.4145** $\angle$ **-71.60°** A.

AS7's Problem 9.73

Determine the equivalent impedance for the circuit.



AS7's Problem 9.73

SOLUTION

Eight impedances and no source, so this is a job for the equivalent-impedance tool rather than a solve. The names below carry the two nodes each element bridges, which is only a convenience — any unique names would do.

```
r10,1,0,6j
r20,2,0,8j
r30,3,0,8j
r40,4,0,12j
```

```
r12,1,2,2
r23,2,3,-6j
r34,3,4,4
r14,1,4,-4j
```

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Resistance / impedance*, with nodes **1** and **0**, in AC. The answer is called z_{eq} .

```
0.3794+i1.46
```

That is $0.3794 + j1.46 \Omega$, which is correct.

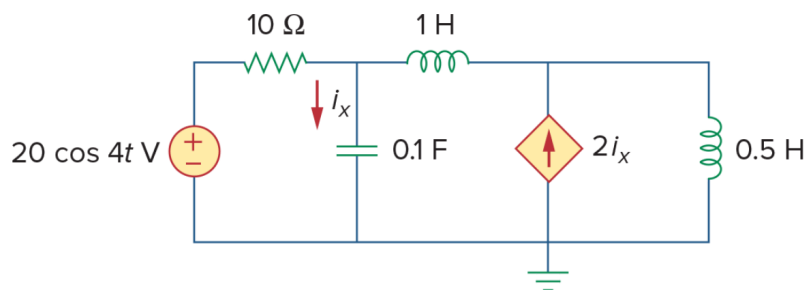
With dependent sources

A dependent source is described exactly like an independent one — the value is just an expression naming another element's answer instead of a number.

The spelling of that name is the one thing that changed between the versions. Version 7 runs the quantity and the element together, as in $2icx$; version 9 uses the same names it reports its answers under, so the current through cx is icx and the voltage across rx is vrx .

AS7's Example 10.1

Find i_x in the circuit.



AS7's Example 10.1

SOLUTION

The current source is controlled by the current through the capacitor cx , which is what the last field of $j1$ says.

```
e,1,0,20
r1,1,2,10
cx,2,0,.1
l1,2,3,1
j1,0,3,2*icx
l2,3,0,.5
```

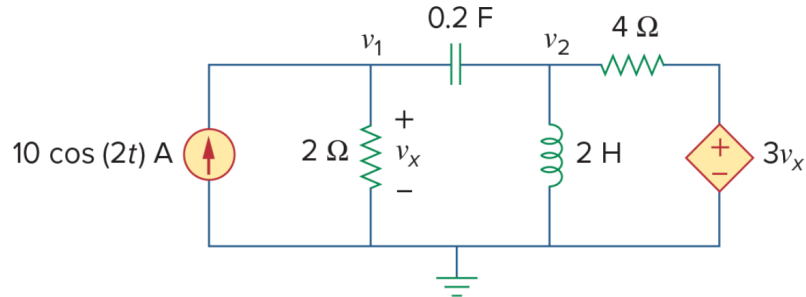
AC, with ω — **angular frequency** set to **4**.

`aa(icx)` gives **7.589** $\angle$ **108.4**° A.

That is 7.59 A at an angle of 108.4° , which is correct.

AS7's Practice Problem 10.1

Find v_1 and v_2 in the circuit.



AS7's Practice Problem 10.1

SOLUTION

This one is controlled by a *voltage* rather than a current — three times the voltage across v_x .

```
j,0,1,10
rx,1,0,2
c,1,2,.2
l,2,0,2
r,2,3,4
e,3,0,3*vx
```

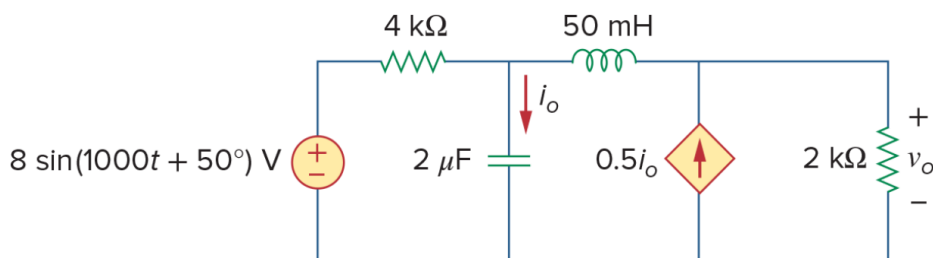
AC, with ω — **angular frequency** set to 2.

aa(v1) gives **11.33**∠**60.02**° V and aa(v2) gives **33.02**∠**57.13**° V.

Both are correct.

AS7's Example 10.13

Obtain v_o and i_o in the circuit.



AS7's Example 10.13

SOLUTION

The book wants its answers in terms of cosine rather than sine, so the source is taken as $8 \cos(1000t - 40^\circ)$ — as a phasor, $(8 \angle -40^\circ)$.

Note the decimal points on the values. They make the arithmetic approximate, which here is what you want: an exact solve of this circuit carries surds through every step for no benefit.

```
e,1,0,(8∠-40°)
r1,1,2,4'k
co,2,0,2'μ
l1,2,3,50'm
j1,0,3,.5*ico
ro,3,0,2'k
```

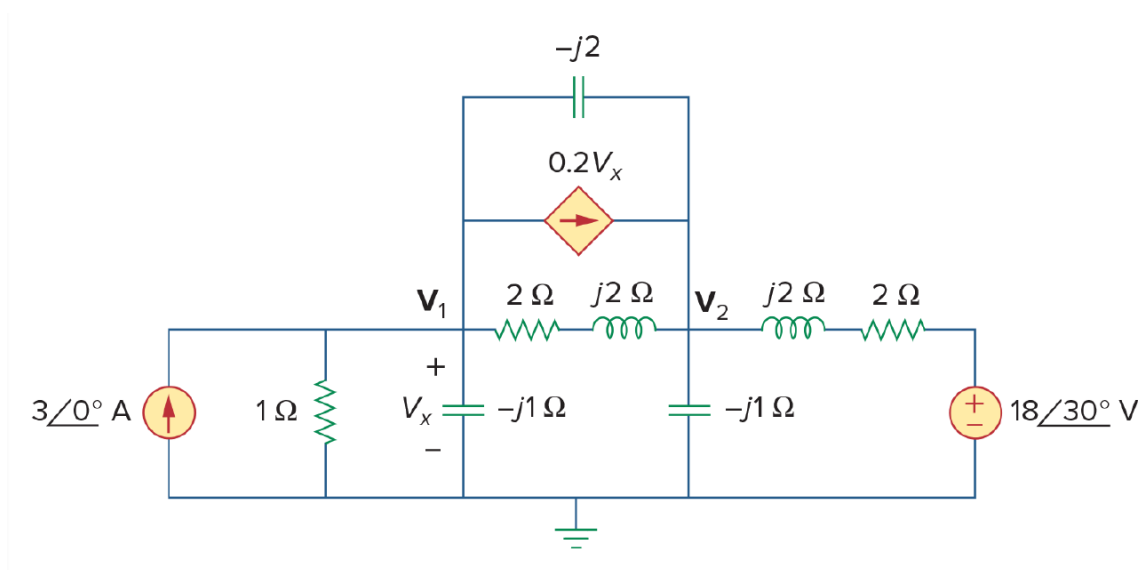
AC, with ω — **angular frequency** set to **1000**.

aa(vro) gives **1.550∠-95.18°** V and aa(ico) gives **0.003264∠-3.743°** A.

That is 1.55 V at -95.18° and 3.26 mA at -3.74°, both correct.

AS7's Example 10.14

Find V_1 and V_2 in the circuit.



AS7's Example 10.14

SOLUTION

Every value here is already in ohms, so no frequency matters. Again the decimal points keep the arithmetic approximate.

```
j1,0,1,3
r1,1,0,1
rx,1,0,-1j
r3,1,2,-2j
j2,1,2,.2*vrX
r4,1,2,2+2j
r5,2,0,-1j
r6,2,3,2+2j
e,3,0,(18∠30°)
```


AC. The frequency is asked for but never used, so anything will do.

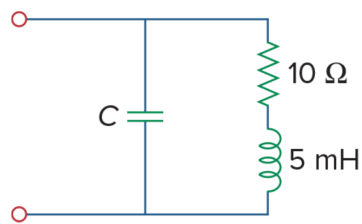
aa(v1) gives **2.708**∠**-56.73**° V and aa(v2) gives **6.914**∠**-80.70**° V.

That is 2.708 V at -56.73° and 6.914 V at -80.70°, both correct.

7.3 Solved numerical-from-symbolic examples

AS7's Problem 9.89

Calculate the value of C so that the net impedance is purely resistive at 2 kHz.



AS7's Problem 9.89

SOLUTION

The answer given by the book is 25 μF . But I think this is one of those unfortunately common instances where there is a mistake in the book. I believe they meant to ask “at 2k rad/s”, because that is the frequency at which the answer they give is right. Let me tell you how I solved this problem and I will let you be the judge.

```
c,1,0,c
r1,1,2,10
l,2,0,5'm
```

Find equivalent, Resistance / impedance, nodes **1** and **0**, AC. The problem gives 2000 Hz and the ω box wants radians per second, so type $2\pi \times 2\text{e}3$ straight into it — it takes an expression, not only a number.



TIP

No cSolve, no ceiling

The exact-versus-approximate problem that plagues the calculator versions comes from a bug in the TI CAS's cSolve. Symbulator 9 solves through SymPy instead, so exact inputs are safe, and you can leave the frequency as $2\pi \times 2\text{e}3$ rather than pre-computing it.

Once the simulation finishes, solve for the value of c that makes the imaginary part of the equivalent impedance zero:

Now open the **Solve** card and ask for the capacitance that leaves no imaginary part:

```
im(zeq) = 0
```

c

Tick **real solutions only** before running it. Without it the answer comes back carrying an $\text{im}(c)$ term, because nothing has told Symbolator that a capacitance is a real number.

The answer is $c = 0.000001235$, or $1.235 \mu\text{F}$. This is not the answer the book gives.

I first used cSolve here, which also works, but my friend Qifan Wang — who verified my answers — pointed out, correctly, that it is not necessary: focusing on the imaginary part removes every reference to the complex operator, so we are solving an equation in real terms only.

Let's now repeat the process, using what I suspect is the frequency they meant: 2000 rad/s. I also want to show you something cool. Now that we know the answer is in the range of μF , we can declare the value of the capacitor as $c' \mu$, so that the value of c we get will be in that scale:

```
c,1,0,c'\mu
r1,1,2,10
l,2,0,5'm
```

Find equivalent, Resistance / impedance, nodes **1** and **0**, in AC at ω **2000**. That gives $z_{eq} = \frac{500i}{-c + 25 + 25i}$, and the condition goes to the **Solve** card:

```
im(zeq) = 0
```

with c as the unknown and **real solutions only** ticked — the same equation as before, now against the new z_{eq} — which answers $c = \mathbf{25}$. The calculator spells that function `imag`; version 9 uses `im`.

The answer is $c = 25$, that is to say $25 \mu\text{F}$. This is the right answer in the book. So they asked the wrong question — they asked it using the wrong frequency unit.

7.4 Solved symbolic examples

AS7's Problem 10.69

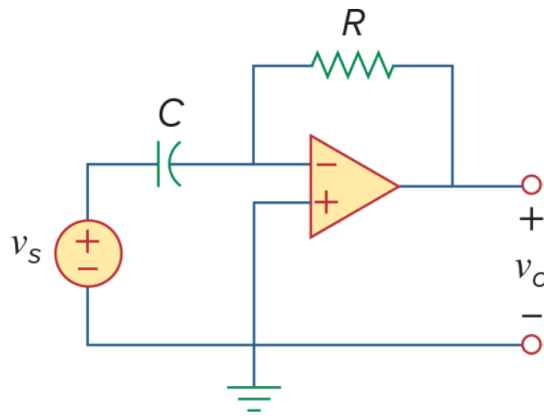
Find V_o/V_s .

SOLUTION

```
e,1,0,vs
c,1,2,c
r1,2,0,r
o,0,2,o
```

The ω box takes a name as readily as a number: type `omega` into it and the answers come back as functions of it.

We ask for v_o/v_s and get $-c \cdot \omega \cdot r \cdot i$, which is correct.



AS7's Problem 10.69

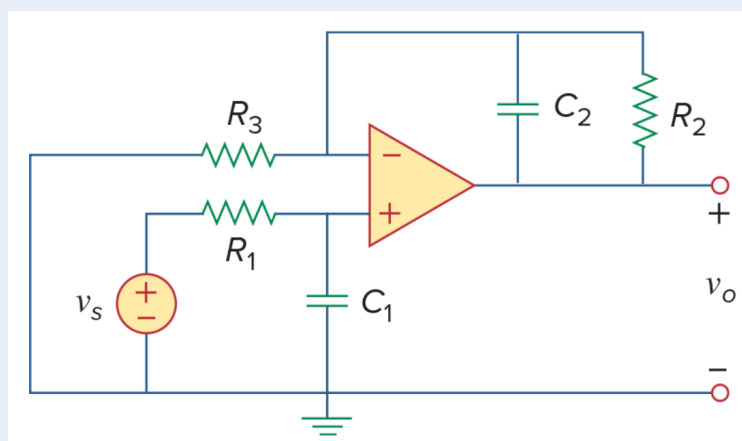
NOTE

Know Symulator's limits

Symulator 9 is not bound by the memory of a handheld, so it goes considerably further than versions 7 and 8 before it slows down. It is still a symbolic solver, though, and symbolic effort grows quickly with the number of nodes. A circuit that would fill a page with algebra is one you should be solving numerically anyway. What matters is that you know the limits, so that they don't take you by surprise in an exam. Think of Symulator as a wonderful Swiss knife of circuit simulators: no matter how good it is, it is still a pocket knife. You would not try to chop a tree down with it. SPICE and its children — LTspice, PSpice, MultiSim — are the chainsaws of circuit simulation. They won't give you symbolic answers, but when you need to chop a tree down, they are the way to go.

AS7's Problem 10.77

Compute the closed-loop gain V_o/V_s for the op-amp circuit.



AS7's Problem 10.77

SOLUTION

Here is an example that takes Symulator to the limit of what it can solve symbolically — everything in it is a symbol, including the frequency.

```
e,1,0,vs
r1,1,2,r1
ca,2,0,ca
r3,0,3,r3
o,2,3,o
cb,3,0,cb
r2,3,0,r2
```

The capacitors are called `ca` and `cb` because the calculator versions had to avoid the reserved `c1` and `c2`; version 9 has no such restriction, but the names are kept so the description matches.

Then ask for the ratio:

```
vo/vs
```

$$\frac{V_o}{V_s} = \frac{j c_b r_2 r_3 \omega + r_2 + r_3}{r_3 (-c_a c_b r_1 r_2 \omega^2 + j c_a r_1 \omega + j c_b r_2 \omega + 1)}$$

It looks different from the answer in the book, but evaluating both shows they are the same expression arranged differently.

The same circuit takes version 9 about ten seconds, with every value still symbolic. That is the difference a desktop makes — the algebra is the same size, there is simply more room to do it in.

LESSON 8

Power in AC circuits

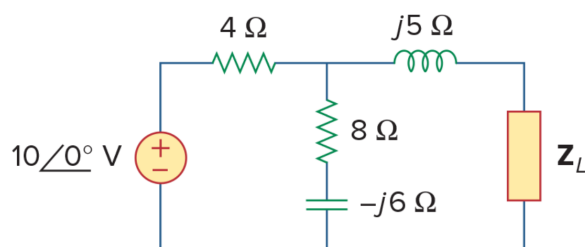
Learn to solve *average power* and *complex power* problems in AC. Learn how you can use RMS values in Symbulator with the **rms** flag. Solve power factor problems using the **pf** tool.

In this lesson you will learn how to run more advanced *alternating current* simulations. You will learn how to solve *average power* problems, when and how to use the **rms** flag, and how to solve *complex power* problems.

8.1 Average power

AS7's Example 11.5

Determine the load impedance Z_L that maximises the average power drawn from the circuit. What is the maximum average power?



AS7's Example 11.5

SOLUTION

Symbulator's **th** tool helps us find all the answers we need. Notice this circuit description does not include the load, because it is not needed as input to **th**.

```
e,1,0,10
r1,1,2,4
r2,2,0,8-6j
r3,2,3,5j
```

Find equivalent, Thévenin / Norton, nodes **3** and **0**, in AC. In AC the equivalent impedance is called z_{eq} rather than r_{eq} .

Thévenin / Norton saves the equivalent impedance in z_{eq} , which evaluates approximately to $2.933 + j4.467$. The load that will deliver the maximum power is its conjugate, $2.933 - j4.467$.

The average maximum power delivered by the circuit is in $p_{\max}$, which evaluates to 2.3674 W. There is no separate $a_{\max}$ in Symbolator 9: $p_{\max}$ looks at the domain you asked for, and in the AC domain it is already the *average* maximum power, computed from the real part of the equivalent impedance.

Now, here's the cool part. Symbolator 9 carries no load expressions on the result — the tool reports v_{th} , i_{no} , z_{eq} and $p_{\max}$, and nothing else — but you do not have to build anything: the maximum average power is one of the four answers it already gives you, in $p_{\max}$.

Results shows $p_{\max} = 2.367$ W, alongside v_{th} , i_{no} and z_{eq} . That is the power delivered when the load is the conjugate of the equivalent impedance, which is the whole point of the exercise — Symbolator has done the conjugating and the halving for you.

8.2 RMS and PF

In the previous lesson we mentioned that after an AC simulation you also get the complex power consumed. How to read these values depends on a flag called the RMS flag.

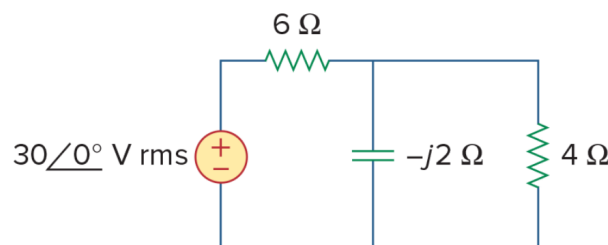
The RMS setting

You will learn in your circuits course that phasor analysis can be done in RMS values. There are advantages to it, and you will have to solve AC circuits both in normal values (also called magnitude values) and in RMS. Working in RMS simply means every current and voltage in the analysis is taken to be RMS. That's it.

Where the calculator versions used a flag variable that persisted between runs, Symbolator 9 puts it in **Settings**, under **AC power convention**: tick **RMS phasors**. Off means peak amplitude, the convention with the divide-by-two, and the setting affects AC power only. Being a setting rather than a stored variable, it is always in view — it cannot be left set from a problem you finished an hour ago.

AS7's Example 11.10

Determine the power factor of the entire circuit as seen by the source. Calculate the average power delivered by the source.



AS7's Example 11.10

SOLUTION

The source value in this problem is given in RMS values, so we must say so.

```
e,1,0,30
r1,1,2,6
r2,2,0,-2j
r3,2,0,4
```

AC, with ω typed into the ω box, and **RMS phasors** ticked in **Settings**.

Or, if you want to simplify the impedances, the resistors can be collapsed into a single parallel combination.

The average power *consumed* by the source is in p_e . To get the average power *delivered*, we ask for the negative of it, which gives us 125.4 W.

The pf tool

Symbulator 9 has the same tool, in the **Mini-Tools** card. Choose *pf* – *power factor*; it asks for the voltage and the current separately rather than for a single complex power or an element name:

```
ve
```

```
-ie
```

It answers **0.97342** leading.

It returns the value and the verbal description together, in one string, just as the calculator prints them.

WARNING

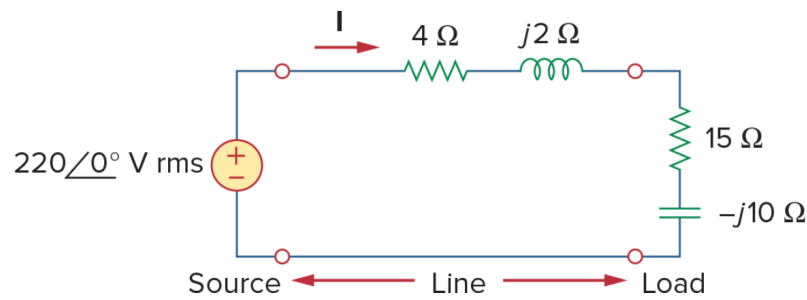
Mind the sign for a source

The minus sign in front of i_e is not a typo, and leaving it out will quietly give you the wrong word. Symbulator reports the power, voltage and current *consumed by* each element, source or not, so the current stored in i_e runs into the source rather than out of it. The calculator's pf tool knew, from the element name you handed it, that e was a source, and flipped the sign for you. The pf tool here is given two bare phasors and cannot know where they came from, so it cannot. Call it with $-i_e$ – the current the source *delivers* – and you get **0.97342 leading**; call it with i_e and you get 0.97342 lagging, which is the same magnitude and the wrong answer. The rule of thumb: negate the current for a source, leave it alone for a load.

8.3 Complex power

AS7's Example 11.13

The figure shows a load being fed by a voltage source through a transmission line. The impedance of the line is represented by the $(4+j2) \Omega$ impedance and a return path. Find the real power and reactive power delivered by the source, and absorbed by the line and load.



AS7's Example 11.13

SOLUTION

The value of this circuit's source is given in RMS. Since the problem asks for answers regarding the line and the load, we should have separate impedances, one for each.

```
e,1,0,220
r1,1,2,4+2j
r2,2,0,15-10j
```

Again AC with `omega` and **RMS phasors** ticked.

The complex power absorbed in the source, line and load are in `-se`, `sr1` and `sr2`:

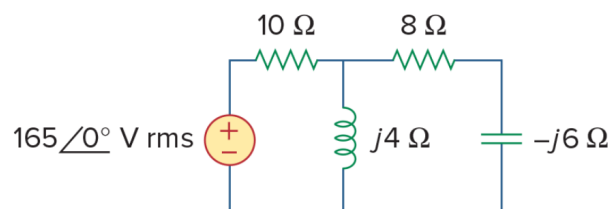
We look in the results and see:

- `-se` = **2163.8** – **911.1i** VA
- `sr1` = **455.5** + **227.8i** VA
- `sr2` = **1708.2** – **1138.8i** VA

These are correct.

AS7's Practice Problem 11.10

Calculate the power factor of the entire circuit as seen by the source. What is the average power supplied by the source?



AS7's Practice Problem 11.10

SOLUTION

The values are RMS, so tick **RMS phasors** in **Settings**.


```
e,1,0,165
r1,1,2,10
r2,2,0,4j
r3,2,3,8
r4,3,0,-6j
```

Written out as four elements rather than one, which is clearer and costs nothing here. The average power supplied is the opposite of the power the source consumes — $-p_e$ in **Evaluate** gives **2007.1 W**.

For the power factor, use **Mini-Tools** with pf , giving it the source's voltage and the current it delivers — which is the opposite of the current through the source element:

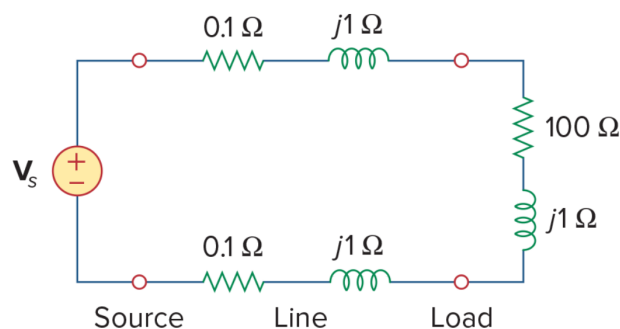
```
ve
```

```
-ie
```

It reads **0.93595 lagging**.

AS7's Problem 11.97

A power transmission system is modelled as shown. If $V_s = 240$ V rms, find the average power absorbed by the load.



AS7's Problem 11.97

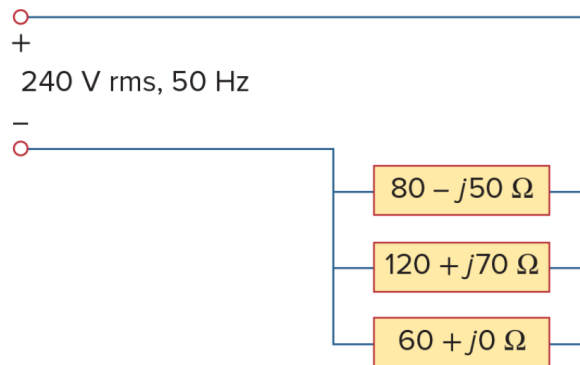
SOLUTION

Three impedances in series: the outgoing line, the load, and the return path.

```
evs,1,0,240
r11,1,2,.1+1j
r1,2,3,100+1j
r12,3,0,.1+1j
```

AC with **RMS phasors** ticked. The $r1$ block's **power consumed** line reads **573.2 W**.

Correct.



AS7's Problem 11.75

AS7's Problem 11.75

Consider the power system shown. Calculate the total complex power, the power factor, and the parallel capacitance needed for a unity power factor.

SOLUTION

Three loads in parallel across a 240 V rms source. The first two parts need no frequency at all, so leave omega as anything.

```
e,1,0,240
r1,1,0,80-50j
r2,1,0,120+70j
r3,1,0,60
```

(a) The complex power delivered is the opposite of the power the source consumes: -se in **Evaluate** gives **1835.9 - 114.7j** VA.

(b) The power factor, from *pf* in **Mini-Tools** with *ve* and -ie, is **0.99805** leading.

(c) This one needs a frequency, because it needs a capacitor. Add one in parallel with a symbolic value (let's say x), and run at the stated 50 Hz — $(2\pi)(50)$ in rad/s.

```
e,1,0,240
r1,1,0,80-50j
r2,1,0,120+70j
r3,1,0,60
c,1,0,x
```

Put $2\pi \cdot 50$ in the ω — **angular frequency** box — it takes an expression. Then ask the **Solve** card for the value of x that leaves no reactive power:

```
im(se) = 0
```

```
x
```

Tick **real solutions only**.

It answers **-6.34** μF .

A negative capacitance, and no positive value satisfies the equation — which is probably why the textbook gives no number for this part. It is not a failure of the method: the load is *already* leading, as part (b) said, so no capacitor can bring it to unity. What it needs is the opposite.

Try an inductor instead:

```
e,1,0,240
r1,1,0,80-50j
r2,1,0,120+70j
r3,1,0,60
l,1,0,x
```

Solving the same way gives **1.5987** H, which is positive and therefore the real answer.

Check it by putting that number back in as the inductor's value: the power factor comes out **1**. An inductor of about 1.6 H in parallel is what brings this system to unity.

Three-phase circuits

Learn to solve simple *three-phase* circuits in Y-Y, Y- Δ , Δ - Δ and Δ -Y configurations, both balanced and unbalanced, to find line and phase currents, voltages and complex power.

In this lesson you will learn to use Symbulator to solve simple **three-phase** systems, in their four basic configurations of wye-wye, wye-delta, delta-delta and delta-wye. We will see both balanced and unbalanced examples, where you are asked to find currents, voltages and complex power in the source, line and load.

9.1 About solving three-phase circuits

Three-phase circuits can be tricky to solve in Symbulator, for several reasons. It is not always clear which node to use as ground. There are three times as many nodes as in an equivalent single-phase circuit. And sources in a delta array bring mathematical trouble: more unknowns than equations.

Symbulator does a great job on the simple three-phase circuits of a basic Circuits I / II textbook, and here we will see examples and one or two tricks. It would not be my tool of choice for anything larger: as circuits grow, the number of nodes quickly passes what the computer can solve.

9.2 Wye-Wye

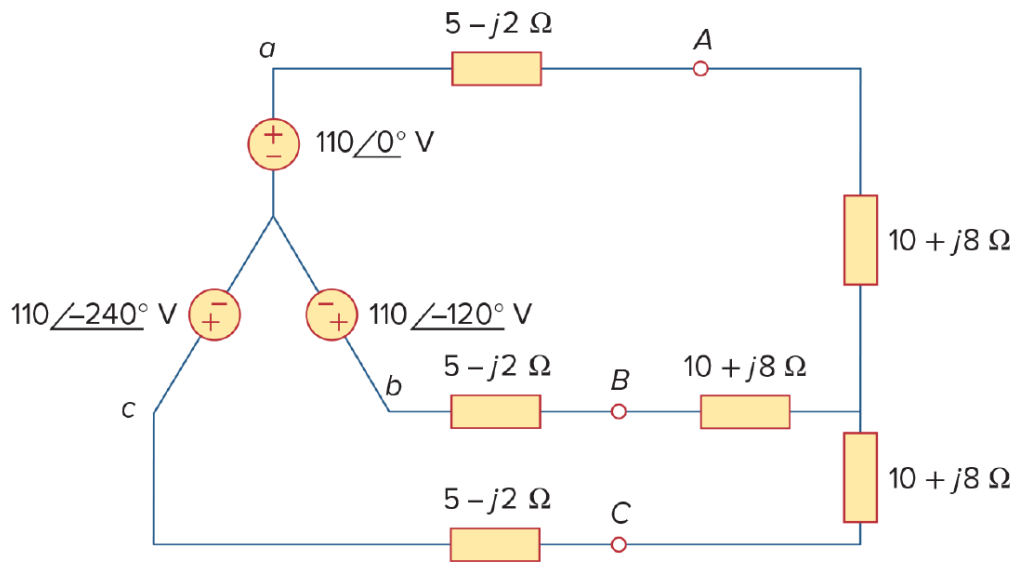
Balanced wye-wye system

AS7's Examples 12.2 & 12.6

Calculate the line currents in the three-wire Y-Y system. Determine the total average power, reactive power, and complex power absorbed at the source and at the load. Assume that the values given for the source are RMS.

SOLUTION

Although not specified in the textbook, to get the answers they give we need to assume the values are RMS. One must describe the circuit carefully in the case of three-phase circuits, because it is very easy to make a mistake. Below is how I describe this one:



AS7's Examples 12.2 and 12.6

```
ea0,ag,0,(110∠0°)
eb0,bg,0,(110∠-120°)
ec0,cg,0,(110∠120°)
rat,ag,ad,5-2j
rbt,bg,bd,5-2j
rct,cg,cd,5-2j
ra0,ad,0,10+8j
rb0,bd,0,10+8j
rc0,cd,0,10+8j
```

AC, with **RMS phasors** ticked. Every value is already an impedance, so the frequency is asked for but never used.

Some observations about my description:

- Notice that I have specified the node at the centre of both Y's as node 0. This is something you can do only in the case of balanced Y-Y systems, not for other configurations and not for unbalanced systems.
- Since Symbolator does not differentiate between lower and upper case variables, nodes called a and A would be considered the same node. Instead, we use the names ag and ad, where the g reminds us a node is on the generation side and the d reminds us it is on the demand side.

Once the simulation is completed, we look in the results for the currents on the transmission lines:

Tick **Show AC answers as polar phasors** in **Settings** and each line impedance's *current through* reads straight off:

- irat = **6.809∠-21.8°** A
- irbt = **6.809∠-141.8°** A
- irect = **6.809∠98.2°** A

The complex power at the source is $s_{ea0} + s_{eb0} + s_{ec0}$, which gives $-2086.2 - 834.5j$ VA. The real part is the average power absorbed by the source; since it is negative, the source is delivering an average power of 2086 W. The imaginary part is the reactive power: 834 var.

The complex power at the load is $s_{ra0} + s_{rb0} + s_{rc0}$, which gives $1390.8 + 1112.6j$ VA. The load is consuming an average power of 1391 W and a reactive power of 1113 var.

Before we move on, try simulating this circuit with a different node as the centre of the load's Y array — call it d0, for the demand side. The voltage in that node evaluates to 0 V, because the system is balanced. That is why we could use 0 for both. In an unbalanced system we cannot.

Further wye-wye problems

AS7's Practice Problem 12.2 & 12.6

Calculate the line voltages and the line currents. Also calculate the complex power at the source and at the load. Assume the source voltage is given as RMS.

A Y-connected balanced three-phase generator with an impedance of $0.4 + j0.3 \Omega$ per phase is connected to a Y-connected balanced load with an impedance of $24 + j19 \Omega$ per phase. The line joining the generator and the load has an impedance of $0.6 + j0.7 \Omega$ per phase. Assuming a positive sequence for the source voltages and that $\mathbf{V}_{an} = 120 \angle 30^\circ$ V, find: (a) the line voltages, (b) the line currents.

AS7's Practice Problem 12.2 and 12.6

SOLUTION

The textbook does not say so, but you only get its answers if you take the values as RMS.

There are **three** impedances in each line here: one inside the generator, one for the transmission line, and one for the load. The node names below say which is which — ag in the generator, as on the supply side, ad on the demand side.

```
ea0,ag,0,(120/30°)
eb0,bg,0,(120/-90°)
ec0,cg,0,(120/150°)
rag,ag,as,.4+.3j
rbg,bg,bs,.4+.3j
rcg,cg,cs,.4+.3j
rat,as,ad,.6+.7j
rbt,bs,bd,.6+.7j
rct,cs,cd,.6+.7j
ra0,ad,0,24+19j
rb0,bd,0,24+19j
rc0,cd,0,24+19j
```

AC, with **RMS phasors** ticked. Every value is already an impedance, so the frequency is asked for but never used.

The question asks for the line voltages — the differences between the lines. For the answers the textbook wants, ask for these:

In **Mini-Tools** with *aa*:

- $aa(v_{ag}-v_{bg}) = 207.8 \angle 60.00^\circ \text{ V}$
- $aa(v_{bg}-v_{cg}) = 207.8 \angle -60.00^\circ \text{ V}$
- $aa(v_{cg}-v_{ag}) = 207.8 \angle 180.0^\circ \text{ V}$



TIP

Which line voltage did you want?

If you ask me, that is not really the voltage between the lines, because the transmission line starts *after* the generator. At the start of the transmission line the answer is $204.6 \angle 60.03^\circ \text{ V}$, and at the end of it $198.7 \angle 59.71^\circ \text{ V}$ — ask for $aa(v_{as}-v_{bs})$ and $aa(v_{ad}-v_{bd})$ respectively. All three are different, and which one you want depends on where you are standing.

The line currents are the currents through the transmission-line impedances:

With polar phasors on, each transmission-line impedance's *current through* reads:

- $i_{rat} = 3.748 \angle -8.660^\circ \text{ A}$
- $i_{rbt} = 3.748 \angle -128.66^\circ \text{ A}$
- $i_{rct} = 3.748 \angle 111.34^\circ \text{ A}$

Finally the complex power. At the source:

Evaluate $s_{ea0}+s_{eb0}+s_{ec0}$, which gives $-1054 - 842.9i \text{ VA}$.

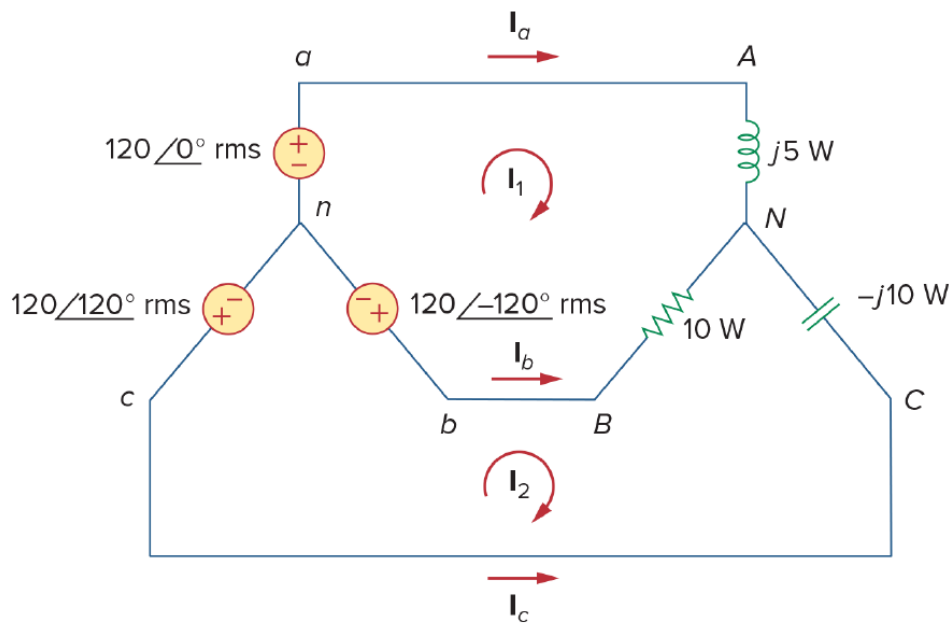
Note this does *not* include the power lost in the source's own internal impedances, which you could add if you wanted them. At the load:

Evaluate $s_{ra0}+s_{rb0}+s_{rc0}$, which gives $1012 + 800.8i \text{ VA}$.

All correct.

Unbalanced wye-wye system

The balanced case let you use node 0 for the centre of *both* wyes, because the voltage at the centre of the load is zero. Unbalanced, it is not — so the load needs a centre node of its own.



AS7's Example 12.10

AS7's Example 12.10

For the unbalanced circuit, find (a) the line currents, (b) the total complex power absorbed by the load, and (c) the total complex power absorbed by the source.

SOLUTION

The centre of the generator's wye is ground, and the centre of the load's wye is a node of its own, called n .

```
ea,a,0,(120∠0°)
eb,b,0,(120∠-120°)
ec,c,0,(120∠120°)
ra,a,n,5j
rb,b,n,10
rc,c,n,-10j
```

AC, with **RMS phasors** ticked.

The three line currents read **56.78∠0°**, **25.46∠135.0°** and **42.76∠-155.1°** A. The complex powers in the load are **16120i**, **6480** and **-18280i** VA, and in the three sources **-6814**, **790.6 - 2951i** and **-456.5 + 5111i** VA.

All correct. Two things are worth checking for yourself. Complex power is conserved, so $\text{sea} + \text{seb} + \text{sec} + \text{sra} + \text{srb} + \text{src}$ evaluates to zero. And the voltage at the centre of the load is *not* zero, as it was in the balanced case — v_n is **120.0 - 283.9i** V.



TIP

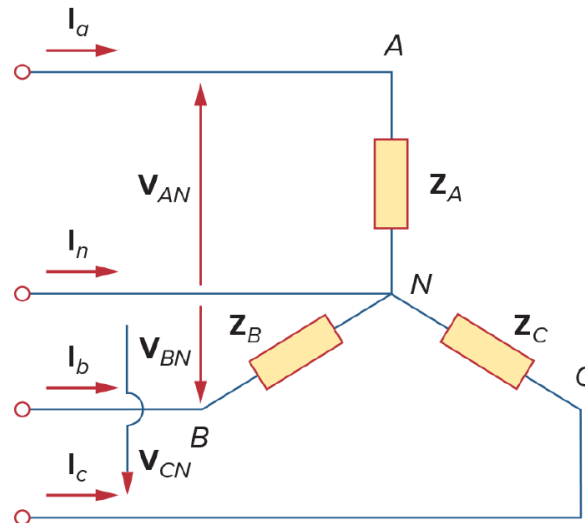
Why ec and rc here, and ecc and rcc on the calculator

The doubled letters exist only to dodge the calculator's reserved names. Version 9

has no reserved names to dodge, so the third phase is named like the other two.

AS7's Example 12.9

The unbalanced Y-load has balanced voltages of 100 V in the *acb* sequence. Calculate the line currents and the neutral current. Take $Z_A = 15 \Omega$, $Z_B = 10 + j5 \Omega$ and $Z_C = 6 - j8 \Omega$.



AS7's Example 12.9

SOLUTION

Two things differ from the last one. The source values are *not* RMS this time. And there is a neutral line joining the centre of the load to the centre of the source, which you describe as a short circuit – an **s** element, which takes just a name and two nodes.

Note also the angles: this is the *acb* sequence, so they run the other way round.

```
ea,a,0,(100∠0°)
eb,b,0,(100∠120°)
ec,c,0,(100∠-120°)
sn,0,n
ra,a,n,15
rb,b,n,10+5j
rc,c,n,6-8j
```

AC, with **RMS phasors** left unticked this time.

The three line currents read **6.667∠0°**, **8.944∠93.43°** and **10.00∠-66.87°** A, and the neutral current **10.06∠178.5°** A.

All four are correct. The neutral carries a current precisely because the load is unbalanced; in the balanced case it would be zero, which is why a balanced three-wire system needs no neutral at all.

9.3 Wye-Delta

A wye source feeding a delta load. The source still has a centre to use as ground; the load does not, which is the only new thing here.

Balanced

AS7's Example 12.3

For the balanced Y- Δ circuit, find the phase currents and the line currents.

A balanced *abc*-sequence Y-connected source with $V_{an} = 100 \angle 10^\circ$ V is connected to a Δ -connected balanced load $(8 + j4)\Omega$ per phase. Calculate the phase and line currents.

AS7's Example 12.3

SOLUTION

Nothing says these values are RMS, so leave the convention alone — the question asks for no powers, so it makes no difference either way. The three sources get the angles of the *abc* sequence.

```
ea0,a,0,(100∠10°)
eb0,b,0,(100∠-110°)
ec0,c,0,(100∠130°)
rab,a,b,8+4j
rca,c,a,8+4j
rbc,b,c,8+4j
```

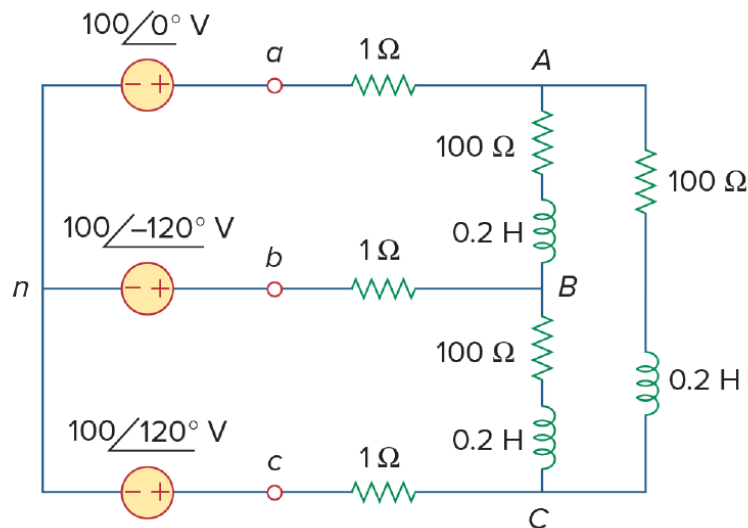
The phase currents are the currents in the three load impedances:

`aa(irab)`, `aa(irbc)` and `aa(irca)` read **19.36∠13.43°**, **19.36∠-106.57°** and **19.36∠133.43°** A.

The line currents are the currents the sources deliver, which is the opposite of the current through each source element:

`aa(-iea0)`, `aa(-ieb0)` and `aa(-iec0)` read **33.54∠-16.57°**, **33.54∠-136.57°** and **33.54∠103.43°** A.

Both sets are correct, and they show the relationship you would expect of a delta load: the line current is $\sqrt{3}$ times the phase current, and lags it by 30° .



AS7's Example 12.11

AS7's Example 12.11

For the balanced Y-Δ circuit, find the line current I_{aA} , the phase voltage V_{AB} , and the phase current I_{AC} . The source frequency is 60 Hz.

SOLUTION

This one has line impedances, so each source reaches the load through a resistor, and the load's three nodes are separate from the source's three.

```
ea1,na1,0,100
eb1,nb1,0,(100∠-120°)
ec1,nc1,0,(100∠120°)
raa,na1,na2,1
rbb,nb1,nb2,1
rcc,nc1,nc2,1
rac,na2,nc2,100+24*pi*j
rcb,nc2,nb2,100+24*pi*j
rba,nb2,na2,100+24*pi*j
```

The line current is the current in one of the line resistors, and the phase voltage is the difference between two load nodes:

$aa(iraa)$ reads **2.35**∠**-36.2°** A and $aa(vna2-vnb2)$ reads **169.94**∠**30.81°** V. Both are correct.

For the phase current, $aa(irac)$ reads **1.357**∠**-66.2°** A.

Unbalanced

If you find a simple unbalanced wye-delta problem, let me know.

9.4 Delta-Delta

Until now, choosing a ground node was easy: the centre of the wye of sources. A delta has no centre, which is the first problem. It is solved by picking one of the delta's own nodes on the generator side and calling it 0.

The second problem is subtler, and it is not Symulator's alone — SPICE-like simulators dislike it too. **A triangle of three voltage sources cannot be solved.** The third source adds no information, because the first two already fix the voltages at all three nodes, but it does add an unknown: the current through it. Drop the redundant equation and the system has one unknown too many.

WARNING

Describe a delta source with two sources, not three

The trick is to leave one source out — the one opposite the node you chose as ground. Two sources fix all three node voltages, so everything outside the delta behaves exactly as it should.

The cost is that you learn nothing about the currents *inside* the sources: the currents you get for the two you kept are not the currents they would carry in a real three-source arrangement. Everything beyond the delta is right; the inside of the generator is not.

Balanced

AS7's Example 12.4

A balanced Δ -connected load of $20 - j15 \Omega$ is fed by a Δ -connected, positive-sequence generator with $V_{ab} = 330 \angle 0^\circ$ V. Find the phase currents of the load and the line currents.

A balanced Δ -connected load having an impedance $20 - j15 \Omega$ is connected to a Δ -connected, positive-sequence generator having $V_{ab} = 330 \angle 0^\circ$ V. Calculate the phase currents of the load and the line currents.

AS7's Example 12.4

SOLUTION

Node **c** becomes ground, so the source left out is the one opposite it. To read the line currents we add three shorts to act as the lines.

```
e0a,0,ag,(330/120°)
eb0,bg,0,(330/-120°)
sat,ag,ad
sbt,bg,bd
sct,0,cd
rab,ad,bd,20-15j
rbc,bd,cd,20-15j
rca,cd,ad,20-15j
```

The line currents are the currents through the shorts:

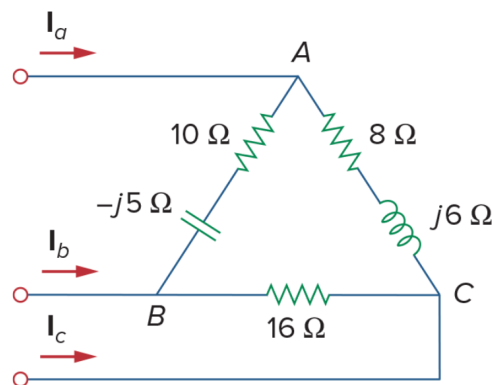
aa(isat), aa(isbt) and aa(isct) read **$22.86\angle 6.87^\circ$** , **$22.86\angle -113.13^\circ$** and **$22.86\angle 126.87^\circ$** A.
Correct.

Unbalanced

Nothing changes in the method. The load impedances simply differ, and the answers stop being three copies of one another.

AS7's Practice Problem 12.9

The unbalanced Δ -load is supplied by balanced line-to-line voltages of 440 V in positive sequence. Find the line currents, taking V_{ab} as the reference.



AS7's Practice Problem 12.9

SOLUTION

```
e0a,0,ag,(440∠120°)
eb0,bg,0,(440∠-120°)
sla,ag,ad
slb,bg,bd
slc,0,cd
rab,ad,bd,10-5j
rbc,bd,cd,16
rca,cd,ad,8+6j
```

aa(isla), aa(islb) and aa(islc) read **$39.71\angle -41.07^\circ$** , **$64.12\angle -139.77^\circ$** and **$70.13\angle 74.27^\circ$** A.

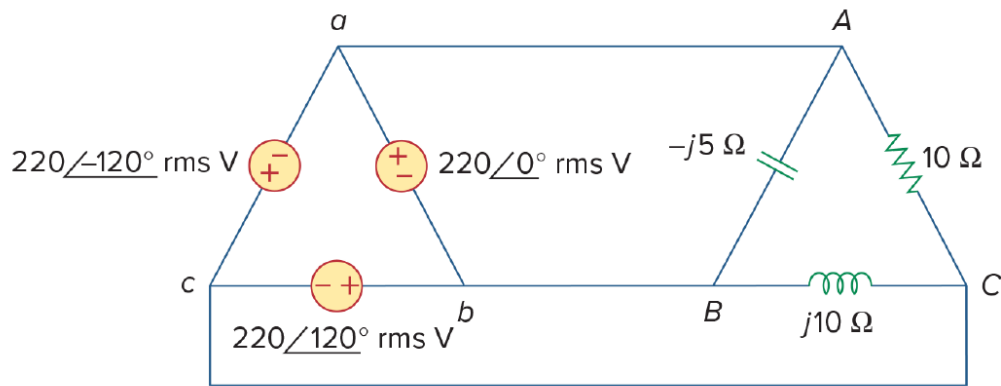
Correct — and unlike the balanced case, all three differ in magnitude as well as angle.

AS7's Practice Problem 12.10

Find the line currents in the unbalanced three-phase circuit, and the real power absorbed by the load.

SOLUTION

These values are RMS, so tick **RMS phasors** in **Settings** — this one does ask for power.



AS7's Practice Problem 12.10

```
e0a,0,ag,(220∠-120°)
eb0,bg,0,(220∠120°)
sla,ag,ad
slb,bg,bd
slc,0,cd
rab,ad,bd,-5j
rbc,bd,cd,10j
rca,cd,ad,10
```

The three line currents read **64.00∠80.1°**, **38.11∠-60°** and **42.50∠-135°** A.

For the power, add the three loads' consumption in **Evaluate**:

```
prca+prab+prbc
```

It gives **4840** W.

Correct.

AS7's Example 12.12

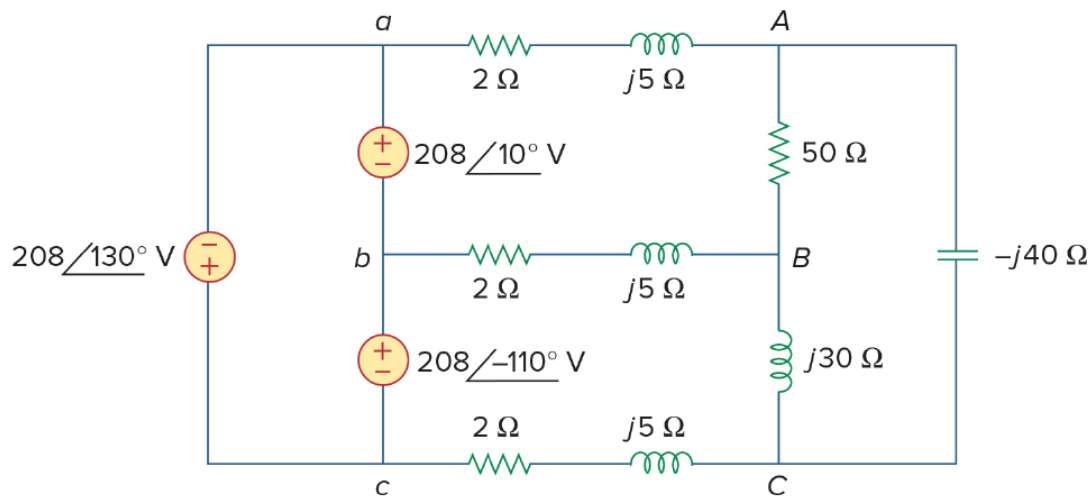
For the unbalanced Δ - Δ circuit, find the generator current I_{ab} , the line current I_{bB} and the phase current I_{BC} .

SOLUTION

Two of the three are ordinary element currents. The first — the current *inside* the generator — is the awkward one, and it is dealt with after them.

```
e0a,0,ag,(208∠130°)
eb0,bg,0,(208∠-110°)
rla,ag,ad,2+5j
rlb,bg,bd,2+5j
rlc,0,cd,2+5j
rab,ad,bd,50
rbc,bd,cd,30j
rca,cd,ad,-40j
```

Find equivalent is not needed here — a plain AC solve gives both. `aa(ir1b)` reads



AS7's Example 12.12

9.106 $\angle 168.48^\circ$ A and $aa(irbc)$ reads **5.500** $\angle 172.47^\circ$ A, matching the printed answers.

The remaining answer, the generator current, is trickier. They ask for the current in the source we did not simulate — and even for one of the two we did have, we could not trust it, since the real circuit uses three.

Now, I tried something, and I think I got lucky, because I got the answer the book gives. This is what I tried:

$-(ie0a+ie0b)/3$ in **Mini-Tools** with aa , which takes an expression as readily as a name and answers with the magnitude and the angle.

What I thought was: the current coming out of the two sources in my simulation would, in reality, come out of three. So adding those two currents and dividing by three may approximate the current out of one source. I am confident that holds in a balanced circuit. This one is not balanced, but it was the best I had. So I tried it.

5.959 $\angle -177.18^\circ$ A

And it worked. That is the answer in the book.

WARNING

Treat that as a guess that was checked, not as a method

The argument holds in a balanced circuit. This one is not balanced; it was used because there was nothing better to hand, and it happened to land on the book's figure. Check it against something before trusting it elsewhere.

9.5 Delta-Wye

The Δ -Y is the platypus of three-phase systems. The sensible route is usually to convert the Δ source into an equivalent Y and solve it as a Y-Y — and if you are doing this by hand, do that.

Inside Symulator there is another way: make the **centre of the wye load** the ground

node, and describe the delta source with two sources as before.

Balanced

AS7's Example 12.5

For the balanced Δ -Y circuit, find the line currents.

A balanced Y-connected load with a phase impedance of $40 + j25 \Omega$ is supplied by a balanced, positive sequence Δ -connected source with a line voltage of 210 V. Calculate the phase currents. Use V_{ab} as a reference.

AS7's Example 12.5

SOLUTION

```
eca,c,a,(210/120°)
ebc,b,c,(210/-120°)
ra,a,0,40+25j
rb,b,0,40+25j
rc,c,0,40+25j
```

Node 0 here is the centre of the load's wye, not a node of the source at all.

aa(ira), aa(irb) and aa(irc) read **$2.570 \angle -62.01^\circ$** , **$2.570 \angle 177.99^\circ$** and **$2.570 \angle 57.99^\circ$** A.

Correct — balanced, so three equal magnitudes 120° apart.

Unbalanced

If you find a simple unbalanced delta-wye problem, let me know.

Mutual inductance and ideal transformers

Learn how to describe a *mutual inductance* between two inductors or two impedances using the **m** element. Learn how to describe an ideal *transformer* using the **t** element.

Two coils that share a magnetic field are not two separate elements, and a circuit description has to say so. This lesson covers the two ways of saying it: **m** for a mutual inductance between a pair of elements, and **t** for an ideal transformer.

10.1 How to describe mutual inductance

Textbooks give a coupled pair one of two ways, and Symbolator takes both.

- Two inductors in **henries**, coupled by a mutual inductance also in henries.
- Two impedances in $j\Omega$, coupled by a mutual impedance also in $j\Omega$. Written this way the coils are **r** elements with imaginary values.

Either way the coupling itself is an **m** element with four fields: a name starting with **m**, the names of the two coupled elements, and the value of the coupling. For example: `m1, 11, 12, 1.5`

That is a mutual inductance of 1.5 H between inductors 11 and 12. In $j\Omega$ it looks the same, naming two **r** elements instead: `m1, r1, r2, 2j`

WARNING

Don't mix the two

`m1, 11, r1, 2` couples an inductor given in henries to one given in $j\Omega$. It will not be refused and it will not be right. Keep all three values in the same units: all henries, or all $j\Omega$.

The dots are the node order

Every coupled-inductor problem in a textbook has a dot on each coil, marking the ends that share polarity. Symbolator has no way to draw a dot, so **the first node you name in each coupled element is its dotted end**.

That is the whole convention, and it is the thing most likely to give you a correct-looking wrong answer. If you list the nodes of one coil the other way round, the sign of the cou-

pling flips and every current comes back wrong.

⚠ WARNING

Keep the dots consistent

Read the schematic, decide which end of each coil is dotted, and declare that node first — in both coupled elements. Look at the solved examples below and notice that the dotted ends are always the first node given.

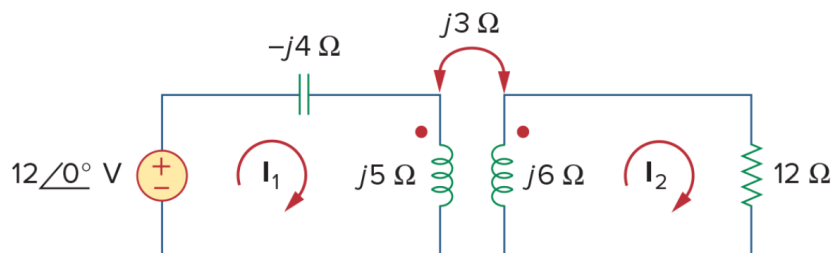
The value of a mutual inductance can use SI prefixes.

What answers do you get? None. An `m` element is a statement about two other elements, not a component with its own voltage and current — its effect shows up in the currents of the elements it couples.

10.2 Instructive m problems

AS7's Example 13.1

Calculate the phasor currents I_1 and I_2 in the circuit.



AS7's Example 13.1

SOLUTION

Three things to notice in the description. The bottom node of both halves is ground. The coils are given in $j\Omega$, so they are `r` elements with imaginary values, and the mutual is imaginary too, to match. And the dotted node of each coupled element is named first.

```
e,1,0,12
r1,1,2,-4j
r2,2,0,5j
m,r2,r3,3j
r3,3,0,6j
r4,3,0,12
```

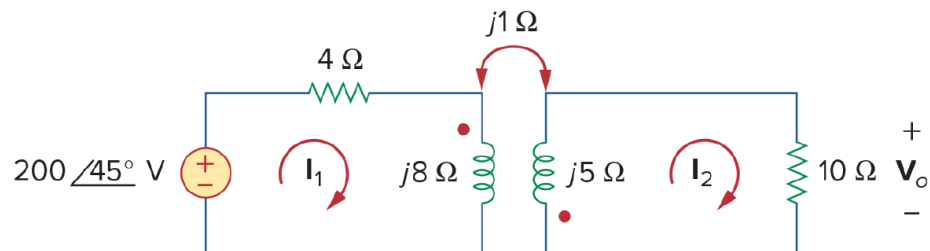
AC, and leave **RMS phasors** unticked. Then read the two currents with **Mini-Tools** set to `aa`:

`aa(ir2)` reads **13.02∠-49.4°** A and `aa(ir4)` reads **2.910∠14.04°** A.

Both are correct.

AS7's Practice Problem 13.1

Determine the voltage V_o in the circuit.



AS7's Practice Problem 13.1

SOLUTION

Again the dotted node of the coupled inductor comes first.

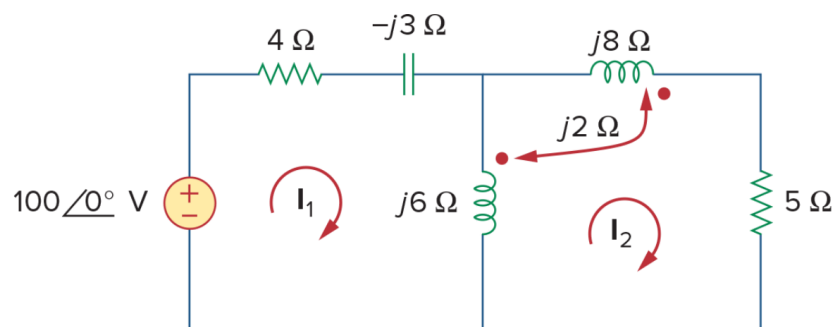
```
e,1,0,(200∠45°)
r1,1,2,4
r2,2,0,8j
m,r2,r3,1j
r3,0,0,5j
r4,0,0,10
```

aa(vo) reads **20.00 ∠ -134.43° V**.

Correct.

AS7's Example 13.2

Calculate the mesh currents in the circuit.



AS7's Example 13.2

SOLUTION

Notice the node order again, matching the dots.

```
e,1,0,100
r1,1,2,4-3j
r2,2,0,6j
```

```
m,r2,r3,2j  
r3,3,2,8j  
r4,3,0,5
```

aa(ir1) reads **20.30**∠**3.5°** A and aa(ir4) reads **8.693**∠**19.03°** A.

Correct.

10.3 How to describe ideal transformers

An ideal transformer has four terminals — two a side. Symulator simulates the case where **one terminal on each side is grounded**, which covers the textbook problems; you name only the two live nodes.

A **t** element takes five fields: a name starting with **t**, the live node on the first side, the live node on the second side, and the turns on each side. For example: **t1,1,2,80,800**

That is 80 turns on the side connected to node 1 and 800 on the side connected to node 2, wound the same way. Only the ratio matters, so **1,10** would describe the same transformer.

Polarity is a minus sign. If the dots are on opposite ends, make one of the two turn counts negative: **t,a,b,1,-5**

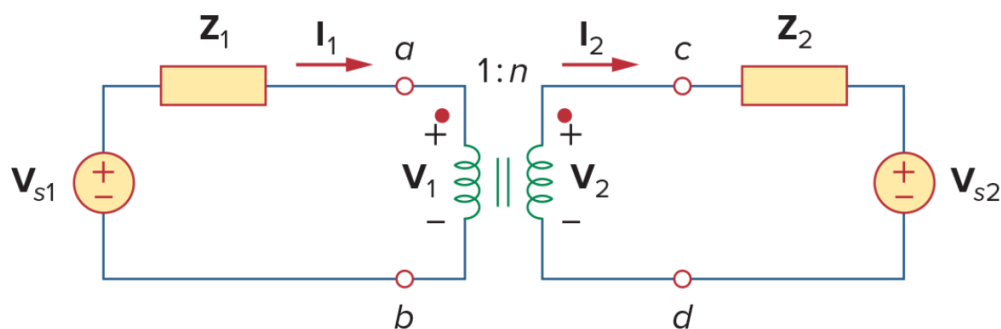
It does not matter which of the two you make negative, as long as one of them is.

What answers do you get? The voltages at the two live nodes, as for any node in the circuit, and the current entering the transformer, named with the transformer and the node — for a transformer **t** on node 2, that is **it2**.

10.4 Instructive t problems

AS7's Figure 13.33

Obtain V_{Th} and Z_{Eq} for the part of the circuit to the right of nodes a and b.



AS7's Figure 13.33

SOLUTION

Everything left of a and b is ignored. Note that this one is entirely symbolic — the turns ratio, the impedance and the source are all names.

```
t, 2, 3, 1, n
r2, 3, 4, z2
e2, 4, 0, vs2
```

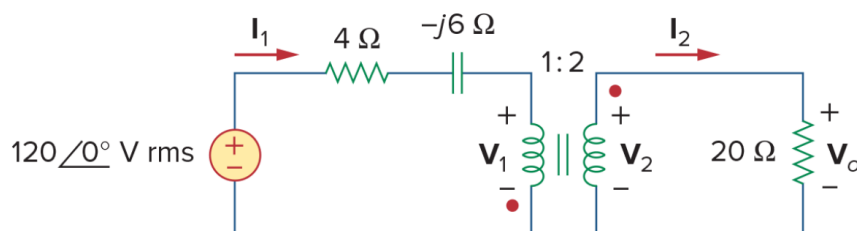
Find equivalent, Thévenin / Norton, nodes **2** and **0**, in DC.

v_{th} is v_{s2}/n and r_{eq} is z_2/n^2 .

Correct — the transformer refers the source by the turns ratio and the impedance by its square, which is exactly what an ideal transformer does.

AS7's Example 13.8

For the ideal transformer circuit, find the source current I_1 , the output voltage V_o , and the complex power supplied by the source.



AS7's Example 13.8

SOLUTION

One of the turn counts is negative here, because the dots are of opposite polarity.

```
e, 1, 0, 120
r1, 1, 2, 4-6j
t, 2, 3, 1, -2
ro, 3, 0, 20
```

AC, with **RMS phasors** ticked this time — the question asks for complex power, and that is the convention it wants.

Three readings from **Mini-Tools** with *aa*: i_{r1} is **11.09∠33.69°** A, v_{ro} is **110.9∠-146.31°** V, and *-se* — the complex power *supplied*, so the opposite of the power consumed by the source — is **1331∠-33.69°** VA.

Three conversions in a row is the point at which the setting is less work than the tool: tick **Show AC answers as polar phasors** in **Settings** and the first two are already in that form when the circuit solves. The third still wants *aa*, because *-se* is an expression rather than an answer.

All three are correct.

LESSON 11

Bode plots and resonance

Learn to do gain and phase *Bode plots* for transfer functions of ω and s with the **bode** tool. Find the *resonant frequency* of an answer by solving for the frequency at which the reactance vanishes.

Two things that only make sense across a range of frequencies rather than at one: the Bode plot, which draws a transfer function's gain and phase as frequency sweeps, and resonance, which is the one frequency where a circuit's reactance vanishes.

11.1 Do Bode plots

A Bode plot shows the gain and the phase of a transfer function against frequency, on a logarithmic axis.

Symulator 9 draws it for you, in the **Plot** card below the results. Solve the circuit first — the plot is of an answer, so there has to be one — then set **Plot type** to *Bode plot of a variable (FD)*. The two time boxes relabel themselves as frequencies.

```
e,1,0,1  
r1,1,2,1000  
c1,2,0,1e-6
```

- **Variable to plot:** v_2
- **Start frequency (Hz)** and **End frequency (Hz):** 10 and 100000
- **Points:** 300

Both curves appear together, magnitude in decibels above and phase in degrees below. For this low-pass RC that is the rolloff you would expect: flat at low frequency, falling 20 dB per decade after the corner, with the phase heading for -90° .

WARNING

The variable, not an expression

The box takes an answer's *name* — v_2 , i_{r1} — not a formula of your own, and it has to be one this circuit actually has. **Results** lists every name above it. The one place a formula does belong is the transfer-function plot type, next.

It is also tempting to solve in AC and then plot against frequency. There is nothing to sweep: an AC result is a phasor computed at one fixed ω , a single point. The Plot card re-solves the circuit at every frequency instead, which is why it works from the circuit rather than from a result.

When you have $H(s)$ itself

The textbook often hands you the transfer function and no circuit — which is exactly what the calculator's tool took. Version 9 takes it too: set **Plot type** to *Bode plot of a transfer function $H(s)$ (FD)*, and the variable box becomes **Transfer function $H(s)$** . Type the function there; the Circuit Description is ignored for this plot type, so it works on an empty page.

- Write the function **in terms of s** . A textbook function of $j\omega$ becomes one of s by writing s wherever $j\omega$ appears — a Bode plot sweeps the imaginary axis, where s is $j\omega$.
- The expression takes the same shorthand a circuit value does — $^$ for powers, implied multiplication, $1/k$ — and it must be numeric apart from s . A leftover symbol is refused by name.
- The frequency boxes are in **hertz**. The calculator examples in this lesson give their ranges in rad/s; divide by 2π for the same sweep in Hz (0.1 to 300 rad/s is about 0.016 to 48 Hz).

AS7's Example 14.3

Construct the Bode plots for the given transfer function.

Version 9 takes the expression directly, like the calculator: set **Plot type** to *Bode plot of a transfer function $H(s)$ (FD)* and type the problem's function into the box, writing s wherever the textbook writes $j\omega$ — no imaginary operator involved. The frequency boxes are in hertz, so the calculator's sweep of 0.1 to 300 rad/s is about 0.016 to 48 Hz. Reading the answers off the curve: the maximum the calculator found at 4.47 rad/s sits near 0.71 Hz, and the zero crossing it found at 199.8 rad/s near 31.8 Hz.

AS7's Practice Problem 14.3

Draw the Bode plots for the given transfer function.

The transfer-function plot type again: write the function in terms of s and sweep. The calculator's 0.1 to 100 rad/s is about 0.016 to 16 Hz.

AS7's Example 14.4

Construct the Bode plots for the given transfer function.

Solved the same way: the *Bode plot of a transfer function $H(s)$ (FD)* type, with the function written in terms of s .

AS7's Practice Problem 14.4

Construct the Bode plots for the given transfer function.

The calculator had to be asked for gain and phase together; version 9 always draws both, magnitude above and phase below, so there is nothing extra to ask for.

AS7's Example 14.5

Draw the Bode plots for the given transfer function.

The same transfer-function type. There is no typing the expression “outside the tool first” to check it — the box shows it in full as you type, and if a symbol other than s slips in, the error names it rather than plotting the wrong thing.

AS7's Practice Problem 14.5

Draw the Bode plots for the given transfer function.

Nothing new here either: the transfer-function type, the function written in terms of s , both curves drawn together.

11.2 Resonance

The resonant frequency is the one at which a circuit's reactance vanishes — where the impedance the source sees becomes purely real.

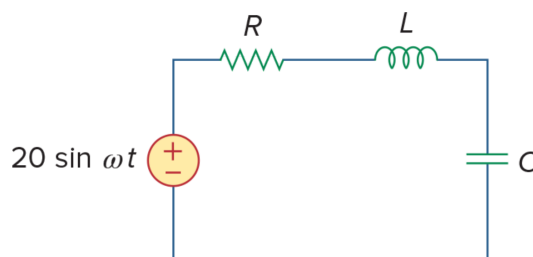
There is no dedicated tool, and none is needed: solve the circuit in AC with w in the ω — **angular frequency** box, then put the condition to the **Solve** card.

Two things make it work, and both are easy to leave out:

- **Tick real solutions only.** Solving $\text{im}(ze) = 0$ over the complex field returns answers written in terms of $\text{re}(w)$ and $\text{im}(w)$ rather than w — useless. Declaring the unknown real is what makes it solvable.
 - **Add $w > 0$ as a condition.** The algebra gives $\pm$ the frequency, and only one of the two is a frequency.
-

AS7's Example 14.7

Find the resonance frequency ω_0 and the bandwidth frequencies ω_1 and ω_2 .



AS7's Example 14.7

SOLUTION


```
e,1,0,20
r,1,2,2
l,2,3,1'm
c,3,0,.4'u
```

Put w in the ω box and solve in AC. Then, in the **Solve** card:

```
im(ze) = 0
```

```
w
```

```
w>0
```

with **real solutions only** ticked. It gives $\omega_0 = 50000$ rad/s.

The bandwidth frequencies are the half-power points, where the impedance magnitude is $\sqrt{2}$ times its resistance. Same circuit, different equation:

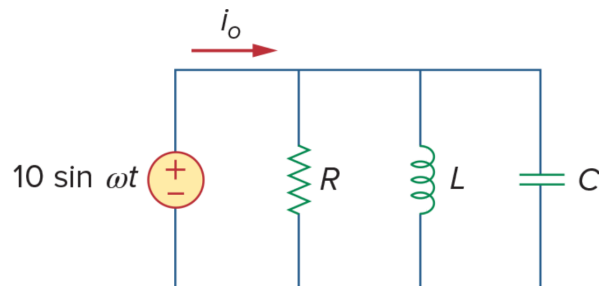
```
abs(ze) = sqrt(2)*2
```

With the condition $w < 50000$ that gives $\omega_1 = 49010$ rad/s, and with $w > 50000$ it gives $\omega_2 = 51010$ rad/s.

All three are correct.

AS7's Example 14.8

Find the resonance frequency and the bandwidth frequencies.



AS7's Example 14.8

SOLUTION

```
e,1,0,10
r,1,0,8'k
l,1,0,.2'm
c,1,0,8'u
```

Trying the same equation as before does not go well here. The peak of resonance is so sharp — a needle — that a numerical search struggles to find its way to the solution. A different condition converges immediately: ask instead for the frequency at which the source's current has no imaginary part.

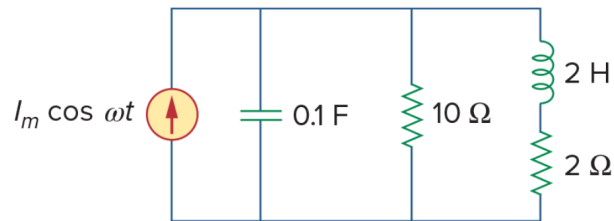
$$\text{im}(ie) = 0$$

with $\omega > 0$ and **real solutions only**, giving $\omega_0 = 25000$ rad/s.

Version 9 solves this symbolically rather than by searching, so the sharpness of the peak costs it nothing — but the alternative equation is worth knowing anyway, because it says something true about the circuit: at resonance the source sees a purely real load, so its current is in phase with its voltage.

AS7's Example 14.9

Determine the resonant frequency ω_0 of the circuit.



AS7's Example 14.9

SOLUTION

```
j,0,1,1
c,1,0,.1
r1,1,0,10
l,1,2,2
r2,2,0,2
```

This time, look for the frequency at which the imaginary part of the voltage at node 1 vanishes.

$$\text{im}(v1) = 0$$

with $\omega > 0$ and **real solutions only**.

ω_0 is 2 rad/s. Correct.

LESSON 12

The complex frequency domain

Learn to solve *complex-frequency domain* problems using **fd**. Move between the time domain and the s-domain with the **t2s** and **s2t** shortcuts. Learn to solve transfer function problems.

Every analysis so far has answered in the same domain you asked in. This one does not, and that is the point of it: the complex frequency domain, or s-domain, is where a differential equation becomes an algebraic one. Symulator will do the algebra; your job is to be clear about which domain each number is in.

12.1 s-domain analysis

Symulator has an analysis type called *FD – complex frequency domain* that solves a circuit in the domain of complex frequency, where $s = j\omega$.

The rule that matters is this one, and it catches everybody once:

WARNING

FD is in s. TR is in time.

The value of a source given to **FD** is read as an s-domain expression. The same value given to **TR** is read as a function of time.

A source value of 1 in FD is not the same thing as a source value of 1 in TR. In FD it is an impulse; in TR it is a constant. Neither is wrong — they are answers to different questions — but they are not interchangeable, and nothing will warn you.

The same applies to what comes back: **FD answers in s**. Voltages and currents are functions of s, and if you want them in time you have to transform them back.

Resistances are read in the s-domain too, which is more useful than it sounds: it means you can describe a capacitor or an inductor *as an impedance*, writing its value in terms of s, as long as its initial condition is zero.

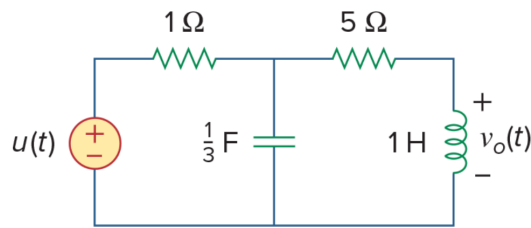
Other than the domain, FD and TR are twins. Capacitors and inductors given in farads and henries are described identically in both, and so is everything else — shorts, op-amps, transformers, all of it.

AS7's Example 16.1

Find $v_o(t)$ in the circuit, assuming zero initial conditions.

SOLUTION

Before we start: this problem does not need the s-domain at all. The source is given as



AS7's Example 16.1

a function of time and the answer is wanted as a function of time, so TR will do it in one step. Notice that the source value is written in the time domain, because that is what TR reads:

```
e,1,0,u(t)
r1,1,2,1
r2,2,0,5
c,2,0,1/3,0
l,0,0,1,0
```

Run it in TR. The answer is on the o node's line:

$$v_o(t) = \frac{3\sqrt{2}}{2} e^{-4t} \sin(\sqrt{2}t)$$

Now let us do it properly, in the s-domain, which is what you would want if any of the intermediate answers interested you.

The extreme version describes everything as an impedance in s — the source as $1/s$, the capacitor as $3/s$, the inductor as s :

```
e,1,0,1/s
r1,1,2,1
r2,2,0,5
r3,2,0,3/s
r4,0,0,s
```

That was to prove it can be done. In practice the schematic gives you farads and henries, so give Symulator farads and henries and let it do the converting:

```
e,1,0,1/s
r1,1,2,1
r2,2,0,5
c,2,0,1/3,0
l,0,0,1,0
```

Choose *FD — complex frequency domain*.

Both give the same answer, in the s-domain:

$$v_o(s) = \frac{3}{s^2 + 8s + 18}$$

Note the source: $1/s$, not 1. The unit step is $1/s$ in the s-domain, and writing 1 there would have described an impulse instead.

12.2 The t2s and s2t shortcuts

Having to transform every source by hand before typing it, and every answer by hand afterwards, would make the s-domain more trouble than it saves. Symbulator gives you both directions.

`t2s(expression)` converts an expression that is a function of time into its s-domain equivalent, and `s2t(expression)` converts back. Both work wherever you can write an expression: in a source value, in **Evaluate**, and in the **Solve** card.

So the whole problem can be done in the s-domain while you write the source in the domain the schematic gave it to you in:

```
e,1,0,t2s(u(t))
r1,1,2,1
r2,2,0,5
c,2,0,1/3,0
l,0,0,1,0
```

Run that in FD, then put the answer back into the time domain from **Evaluate**:

```
s2t(vo)
```

The curly-bracket shorthand

Version 9 has the same shortcut: wrap the value in curly brackets — $\{u(t)\}$ — and FD reads it as `t2s(u(t))`. As on the calculator, it works only inside a circuit description, so the answer still needs `s2t(...)` when you ask for it in **Evaluate**.

12.3 Instructive FD problems

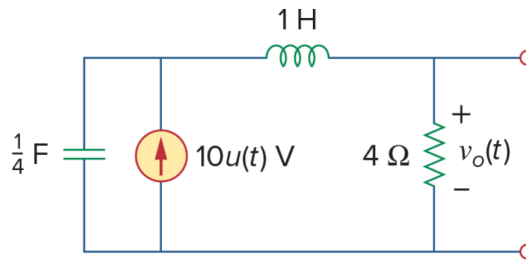
AS7's Practice Problem 16.1

Determine $v_o(t)$ in the circuit.

SOLUTION

The same shape as the example above, so the same approach: describe it in FD with the source converted, then transform the answer back.

```
e,1,0,t2s(u(t))
r1,1,2,6
c,1,0,1/4,0
l,1,0,2,0
```



AS7's Practice Problem 16.1

r2,o,0,3

FD, then $s2t(v_o)$ in **Evaluate**.

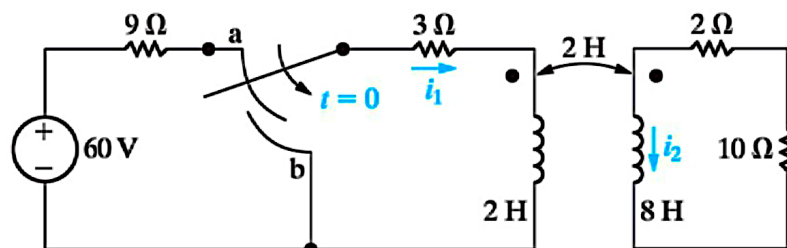
12.4 Transfer function problems

A transfer function is a ratio of two answers, so there is nothing new to learn: solve the circuit with a symbolic source, then divide.

Because both answers are functions of s , so is their ratio — which is exactly what a transfer function is meant to be.

NR11's Example 13.7

Find the transfer function $H(s) = V_o/V_s$.



NR11's Example 13.7

SOLUTION

Give the source a symbolic value and solve in FD:

e,1,0,vs
r1,1,2,1'k
c,2,0,1'μ
r2,2,o,1'k

Solve in FD, then ask **Evaluate** for the ratio:

vo/vs

The answer is a function of s , as a transfer function should be.

LESSON 13

Two-ports

Find the *two-port equivalent* of a network using the **port** script. Learn how to include *two-ports* in your circuit using the **z**, **y**, **h**, **g**, **a** and **b** elements. Calculate gains with the **gain** tool.

A two-port is a network you have stopped caring about the inside of. What is left is four numbers relating the voltage and current at one pair of terminals to the voltage and current at the other. Symbolator will find those four numbers for you, and will also take them as an element in a larger circuit.

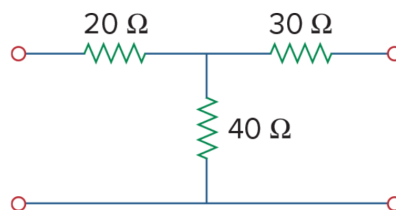
13.1 Find two-port equivalents

Set **Type of analysis** to *Find equivalent* and **Type of equivalent** to *Two-port parameters*. It takes the network and the two **top** nodes of the pair of terminals you want to reduce. The two bottom nodes are always assumed to be ground.

There are six kinds of parameters — *z*, *y*, *h*, *g*, *a* and *b* — and you choose which you want.

AS7's Example 19.1

Determine the *z* parameters for the circuit.



AS7's Example 19.1

SOLUTION

First describe the network, exactly as you would describe any circuit:

```
r1,1,2,20
r2,2,0,40
r3,2,3,30
```

Find equivalent, then *Two-port parameters*, with **Parameters** set to *z* — *impedance* and **1** and **3** in the two node boxes. In DC.

There is no name to invent: version 9 does not store its answers in calculator variables,

so the four come back as z_{11} , z_{12} , z_{21} and z_{22} .

Results gives $z_{11} = 60 \Omega$, $z_{12} = 40 \Omega$, $z_{21} = 40 \Omega$ and $z_{22} = 70 \Omega$.

Correct.

13.2 Use two-ports as elements

A two-port can be an element in a bigger circuit. Its description is three fields: a name whose first letter says which kind it is, then the **top left** node and the **top right** node. Both bottom nodes are ground, always — which is why neither top node may be 0, and the two may not be the same node; Symbolator stops with a message if they are.

```
z,1,2
```

That is a z-type two-port between nodes 1 and 2. An optional fourth term carries its four parameters — see below.

Giving it its four parameters

Three possibilities.

Give them in the description, as an optional fourth term — a list of four, in the order 11, 12, 21, 22:

```
z,1,2,[40,20j,30j,50]
```

This is the way to prefer: the parameters travel with the circuit, so they survive being pasted into a link, saved to a file, or handed to any other tool — and they work in *every* analysis, the equivalent tools included. The entries can be numbers, SI-prefixed values, or expressions, symbols among them.

Store them in Define, calculator-style, and keep the description bare:

```
z11 = 40
z12 = 20j
z21 = 30j
z22 = 50
```

Define any subset and the rest stay symbolic. This is the closest cousin of how versions 7 and 8 stored the values in variables before simulating.

Leave them undefined, in which case they stay symbolic and appear in the answers as z_{11} , z_{12} , z_{21} and z_{22} . That is often what you want — the answer as a formula in the parameters. (Pinning them afterwards in Expert Mode's **Add equations** still works too, in a plain solve, and an equation or condition there overrides the description's own values.)

NOTE

How the variables are named

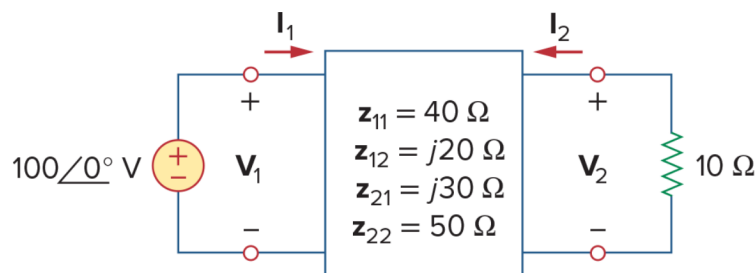
The four variables are always the element's name plus 11, 12, 21, 22: a two-port

called z owns $z_{11}...z_{22}$, and one called z_1 owns $z_{111}...z_{122}$ — the name simply gains the digits. And note that while element and node names ignore case ($2*VR1$ is $2*v_{r1}$), a variable that names nothing in the circuit is case-sensitive: z_a and zA are two different symbols.

What answers do you get? The current entering each port, named with the two-port and the **node** it sits on — for a two-port z on nodes 1 and 2, that is $iz1$ and $iz2$.

AS7's Example 19.2

Find I_1 and I_2 in the circuit.



AS7's Example 19.2

SOLUTION

```
e,1,0,100
z,1,2,[40,20j,30j,50]
r,2,0,10
```

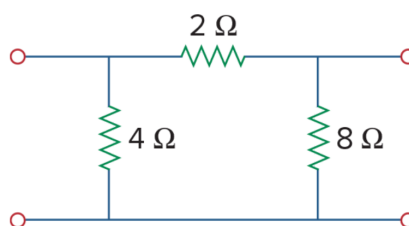
The four parameters ride in the description's fourth term. In AC.

`aa(iz1)` reads $2\angle 0^\circ$ A and `aa(iz2)` reads $1\angle -90^\circ$ A.

13.3 Instructive two-port problems

AS7's Example 19.3

Obtain the y parameters for the network.



AS7's Example 19.3

SOLUTION

```
r1,1,0,4  
r2,1,2,2  
r3,2,0,8
```

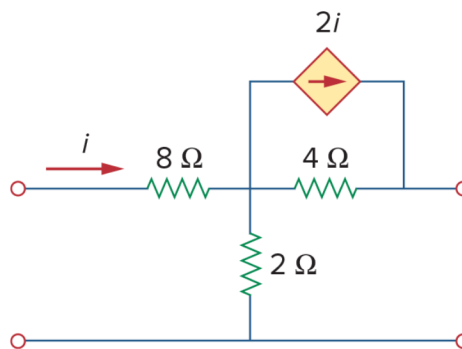
Find equivalent, Two-port parameters, y – admittance, nodes **1** and **2**, DC.

3/4, -1/2, -1/2, 5/8, all in S.

Correct.

AS7's Example 19.4

Determine the y parameters for the two-port shown. Note the dependent source.



AS7's Example 19.4

SOLUTION

```
r1,1,2,8  
r2,2,0,2  
r3,2,3,4  
j,2,3,2*ir1
```

y , nodes **1** and **3**, DC. A two-port equivalent works perfectly well around a controlled source.

3/20, -1/20, -1/4, 1/4, all in S.

Correct.

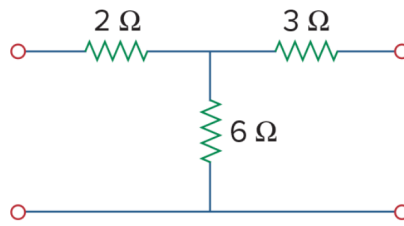
AS7's Example 19.5

Find the hybrid parameters for the two-port network.

SOLUTION

```
r1,1,2,2  
r2,2,0,6  
r3,2,3,3
```

h – hybrid, nodes **1** and **3**, DC.



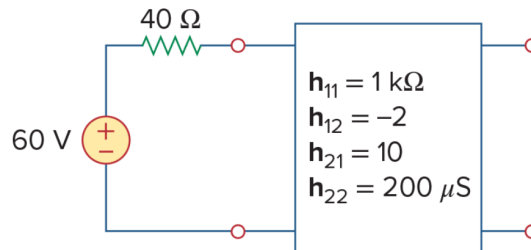
AS7's Example 19.5

4, 2/3, -2/3, 1/9.

Correct.

AS7's Example 19.6

Determine the Thévenin equivalent at the output port of the circuit, whose h parameters are $1000\ \Omega$, -2 , 10 and $2 \times 10^{-4}\ \text{S}$.



AS7's Example 19.6

SOLUTION

```
e,1,0,60
r,1,2,40
h,2,3,[1000,-2,10,2e-4]
```

Find equivalent, Thévenin / Norton, nodes **3** and **0**, DC. Because the parameters ride in the description, they reach the equivalent tools like any other value, and the answers come straight back as numbers.

Leave the fourth term off — $h, 2, 3$ alone — and the same run answers with formulas instead:

$$v_{th} = \frac{-60 h_{21}}{h_{11} h_{22} - h_{12} h_{21} + 40 h_{22}}$$

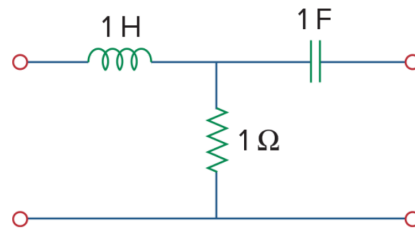
which is useful in its own right: put v_{th} in **Evaluate** with the four parameters in its **Conditions** box, and the same number falls out.

v_{th} comes back **-29.69 V** and r_{eq} **51.46 Ω**.

Correct.

AS7's Example 19.7

Find the g parameters as functions of s for the circuit.



AS7's Example 19.7

SOLUTION

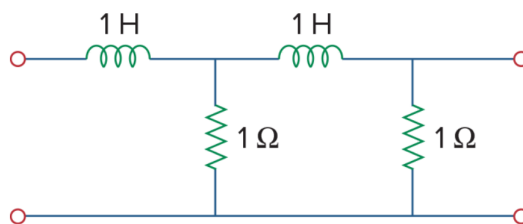
```
l,1,2,1  
r,2,0,1  
c,2,3,1
```

g —*inverse hybrid*, nodes **1** and **3**, and this time in **FD**, since the question asks for functions of s .

$g_{11} = 1/(s + 1)$, $g_{12} = -1/(s + 1)$, $g_{21} = 1/(s + 1)$ and $g_{22} = (s^2 + s + 1)/(s(s + 1))$.

AS7's Practice Problem 19.7

For the ladder network, determine the g parameters in the s domain.



AS7's Practice Problem 19.7

SOLUTION

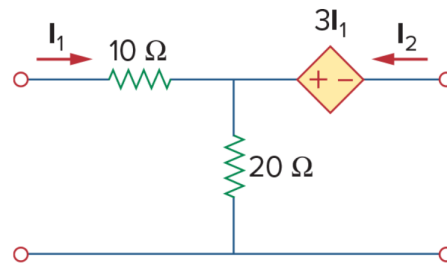
```
l1,1,2,1  
r1,2,0,1  
l2,2,3,1  
r2,3,0,1
```

g , nodes **1** and **3**, FD. The four come back as $(s+2)/(s^2+3s+1)$, $-1/(s^2+3s+1)$, $1/(s^2+3s+1)$ and $s(s+2)/(s^2+3s+1)$.

Correct.

AS7's Example 19.8

Find the transmission parameters for the two-port network.



AS7's Example 19.8

SOLUTION

```
r1,1,2,10  
r2,2,0,20  
e,2,3,3*ir1
```

a – *transmission*, nodes **1** and **3**, DC.

1.765, 15.29, 0.0588, 1.176.

Correct.

13.4 The gain tool

Network problems ask for gains: voltage gain, current gain, power gain, and the impedance seen at the input. Symbolator has a tool for that, in the **Mini-Tools** card: choose *gain*.

It wants four values – the voltage and current at the input, then the voltage and current at the output – and answers with all four figures at once.

Gain Example 1

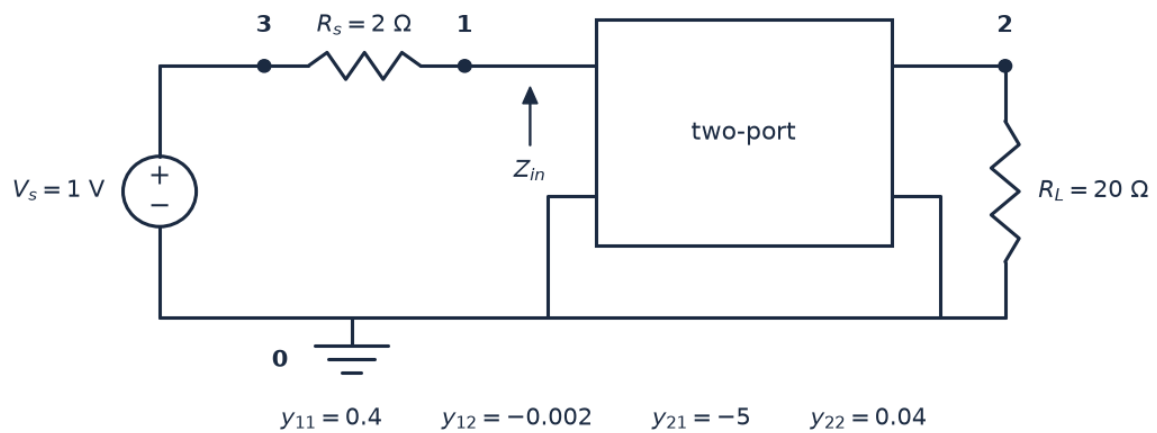
For the circuit, find G_v , G_i , G_p and Z_{in} . The two-port has y parameters $y_{11} = 0.4 \text{ S}$, $y_{12} = -0.002 \text{ S}$, $y_{21} = -5 \text{ S}$ and $y_{22} = 0.04 \text{ S}$.

SOLUTION

The source is 1 V because its value does not matter to a gain – it cancels. A symbolic *vs* would do just as well.

```
es,3,0,1  
rs,3,1,2  
r1,2,0,20  
y,1,2,[0.4,-0.002,-5,0.04]
```

Solve in DC, then open **Mini-Tools**, choose *gain*, and give it the four:



Gain Example 1

v1

iy1

v2

iy2

The voltage gain A_v is **55.56**, the current gain A_i **-9.615**, the power gain A_p **534.2**, and the input impedance Z_i **3.462 Ω** .

These are correct.

Gain Example 2

Find G_v , G_i , G_p and Z_{in} for a two-port with z parameters $z_{11} = 4 \Omega$, $z_{12} = 1.5 \Omega$, $z_{21} = 10 \Omega$ and $z_{22} = 3 \Omega$, driven by a source V_s with 5Ω in series and loaded with 2Ω .

There is no picture of this one — the description below is the whole circuit.

SOLUTION

The same shape as before, with z parameters instead of y .

```

es,3,0,1
rs,3,1,5
r1,2,0,2
z,1,2,[4,1.5,10,3]

```

Solve in DC, then open **Mini-Tools**, choose *gain*, and give it the four:

v1

iz1

v2

iz2

The voltage gain A_v is **4**, the current gain A_i **-2**, the power gain A_p **8**, and the input impedance Z_i **1** Ω .

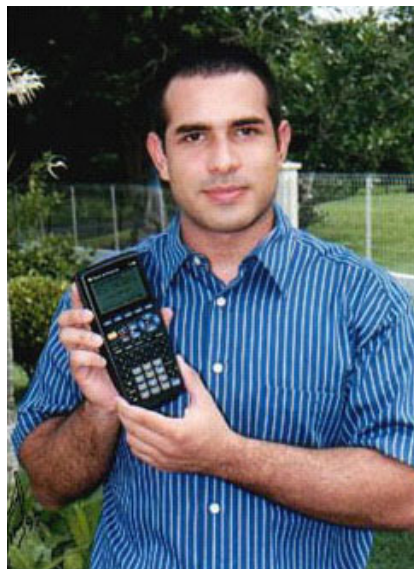
These are correct.

Roll the credits

If you are interested in the people behind Symbulator, here's who made it and who helped, plus some acknowledgements. And some glowing reviews of Symbulator.

About the author

Both Symbulator and this documentation were made by me, **Roberto Perez-Franco**. I started working on it in April 1999, as an engineering student at the Technological University of Panama (Universidad Tecnológica de Panamá) in Azuero. An early version won first place at the IEEE Student Paper Contest for Latin America in 2000, and version 5 (also known as Symbulator Q) served as my graduation thesis in January 2001.



Roberto in 2000, with a TI-89

The first five versions were written between April 1999 and January 2001. The sixth came a dozen years later, in 2013, while I was a researcher at MIT. The seventh and eighth a decade later, in 2023, and the ninth in 2026.

What Symbulator did for me

Thanks in great part to Symbulator, I landed my first job, received a Distinguished Alum award from UTP in 2008, and won a Fulbright Scholarship that took me to a Master of Engineering and later a PhD in Engineering Systems at the Massachusetts Institute of Technology. MIT then hired me, and I lived in Boston for almost twelve years. I now live in Melbourne, Australia. You can say that writing Symbulator changed my life. In a way, I think, when I am no more, Symbulator will preserve a part of my mind as it solves

circuits for new generations, the same way a composer lives in the tunes or a writer in the words they conceived.

Dedication

Two people made Symbulator possible, with their love and support.

The first is my wife **Monica**, who had the patience to live with a cyborg for all the years my mind was focused on coding and documenting. *Mi reina*, my attention was elsewhere, but my heart was always with you.

The second is my late father **Tito**, who invested time and money in my education and bought me every gadget and widget I needed or wanted for my studies, with a blind faith that one day something good would come of it. I'm happy to know that I made you proud, Dad, and I miss you every day.

How to reach me

If you liked Symbulator, let me know. My email is my given name, then my last name with a hyphen, as a dot-com address, like so: rxxxxxx@pxxxx-fxxxx.cxx

Acknowledgements

Human collaborators

Over the past decades, many people have made Symbulator better through their suggestions and corrections, on both the software and the documentation. Many were complete strangers at first and became dear friends through our exchanges.

Among them are: José Vega (Panama), Tim Hutcheson (USA), Lars Frederiksen (Denmark), Joe Riel (USA), Arne Harstad (Norway), Erwin Baert (Belgium), Charles 'Chuck' Ware (USA), Doug Burkett (USA), Reinhard Willinski (Germany), Kamil Malinski (Austria), Jake Adams (USA), Daniele Martini (Italy), Rozgonyi Szabolcs (Hungary), Michael Rans (UK), Alex Astashyn (Russia), Al Charpentier (USA), Nevin McChesney (USA), Ivan Oro Yu (Panama), Pepe Iborra (Spain), Dave Conklin (USA), Carlos Perez Ortega (Chile), Qifan Wang (China) and Antony García (Panama).

My thanks to all of them.

Technology

Symbulator 9 runs on Python and is powered by the algebraic capabilities of **SymPy**. I am thankful to the creators of such wonderful open-source software, and to the maintainers who keep their development going. I am proud to honour the open-source community by making Symbulator free and open-source software, from 2026.

AI collaborator

I also want to acknowledge that porting Symbulator to Python/SymPy, creating its interface and websites and updating its documentation, would not have been possible without the coding prowess of Anthropic’s artificial intelligence assistant. Claude allowed me to do in two weeks something that I was unable to find the time to do in two decades: to port Symbulator to Python.

 WARNING

Use AI responsibly

AI is not without risks, and needs careful policies and regulatory oversight on fronts like ethics, systemic biases, its impact on labour, and its voracious use of water and energy. I believe there is a right way to deploy this technology, and — after working on Symbulator 9 with Claude — I am now convinced that AI will revolutionise the way we work. We just need to do it the right way.

Problem credits

To make this book useful to actual students, the problems were chosen from widely used textbooks on electric circuit analysis, and their circuit diagrams are reproduced here as they appear in the source.

Code	Textbook
AS2	<i>Fundamentals of Electric Circuits</i> (2nd ed.), Charles K. Alexander and Matthew N. O. Sadiku, McGraw-Hill, 2004
AS5	<i>Fundamentals of Electric Circuits</i> (5th ed.), Alexander and Sadiku, McGraw-Hill, 2013
AS7	<i>Fundamentals of Electric Circuits</i> (7th international ed.), Alexander and Sadiku, McGraw-Hill, 2020
B11	<i>Introductory Circuit Analysis</i> (11th ed.), Robert L. Boylestad, Pearson – Prentice Hall, 2007
Bo2	<i>Elementary Linear Circuit Analysis</i> (2nd ed.), Leonard S. Bobrow, Oxford, 1987
HK5	<i>Engineering Circuit Analysis</i> (5th ed.), William H. Hayt Jr. and Jack E. Kemmerly, McGraw-Hill, 1993
NR11	<i>Electric Circuits</i> (11th ed.), James W. Nilsson and Susan A. Riedel, Pearson, 2020
RM3	<i>Circuit Analysis: Theory and Practice</i> (3rd ed.), Allan H. Robbins and Wilhelm C. Miller, Thomson – Delmar Learning, 2004
TR5	<i>The Analysis and Design of Linear Circuits</i> (5th ed.), Roland E. Thomas and Albert J. Rosa, Wiley, 2006

For the purpose of teaching students how to use Symbulator, these diagrams are reproduced under the principle of fair use. No copyright infringement is intended.

Praise received

Over the past quarter of a century, the words of encouragement I received from users of Symbulator were my incentive to keep working on it. Here is a little of that precious collection.

The masterpiece of TI-89 programming.

— **Alex Astashyn**, EE+CoE+CoS major, Polytechnic University, Brooklyn, New York, USA

Symbulator is and always will be the best program for TI-89/92/V200.

— **Charles Ware**, EE major, Columbus, Ohio, USA

Roberto, your program Symbulator is by itself a reason to buy the calc. It's a masterpiece of programming.

— **Pepe Iborra**, Telecommunications, Spain

The program is really a masterpiece. It gives you everything you need, very accurate, very fast. It's really what every Electrical Engineer dreams of. It's so easy to use and it does everything! DC, AC, Transient analysis, Thévenin, Bode plots... everything! I'm really amazed.

— **Nikolaos Trichakis**, EE major, Aristotle University of Thessaloniki, Greece

I wish I had it while going through school.

— **Joe Riel**, BSEE, creator of Syrup (a symbolic circuit simulator for Maple V), University of California, Irvine, USA

Congratulations for your program: it's simply fantastic. Symbulator is worth the cost of a TI-89!

— **Pier Giorgio Raponi**, engineering student, Pisa, Italy

You've made an excellent program. Symbulator is really the best circuit simulator ever made for a calculator.

— **Rui Sebastião**, EE major, University of Coimbra, Portugal

*Symbulator is probably the most useful program ever made for a TI calculator. I am an electronics student and even paid for the EE*Pro software, yet I find myself using Symbulator more because it is much better at what it does.*

— **Josh Cunningham**, electronics major

Symbulator, by far, has been the best tool for EE I have come across yet. [...] It is truly an impressive piece of work. You should be proud. It certainly is helping me to focus on my studying of EE instead of trying to remember all the math.

— **Jim Wachtel**, EE, Process Control & Electrical Co., St. Louis, Missouri, USA

Licence

Symbulator has always been free of cost, and always will be. From June 2013 it was offered under a Creative Commons BY-NC-SA licence. Since 2016 it is open source under the MIT licence, which supersedes it. If you want to port it to another platform, you are free to do so, provided the copyright notice and the attribution of authorship travel with it.

Index

aa tool, 181

Bode plot, 218

capacitor, 135

circuit description, 10

conductance, 40

current source, 39

dependent source, 44, 186

er script, 73

expert mode, 33

frequency domain, 223

gain tool, 233

ground node, 11

impedance, 179

inductor, 136

Laplace transform, 225

mutual inductance, 213

operational amplifier, 107

parallel resistors, 41

phasor, 178

polar phasors, 179

power factor, 195

resistor, 10

resonance, 220

RMS, 194

short circuit, 71

SI prefixes, 11

symbolic circuit, 29

th script, 74

three-phase circuits, 200

transformer, 216

two-port, 227

unbalanced system, 203

voltage source, 12