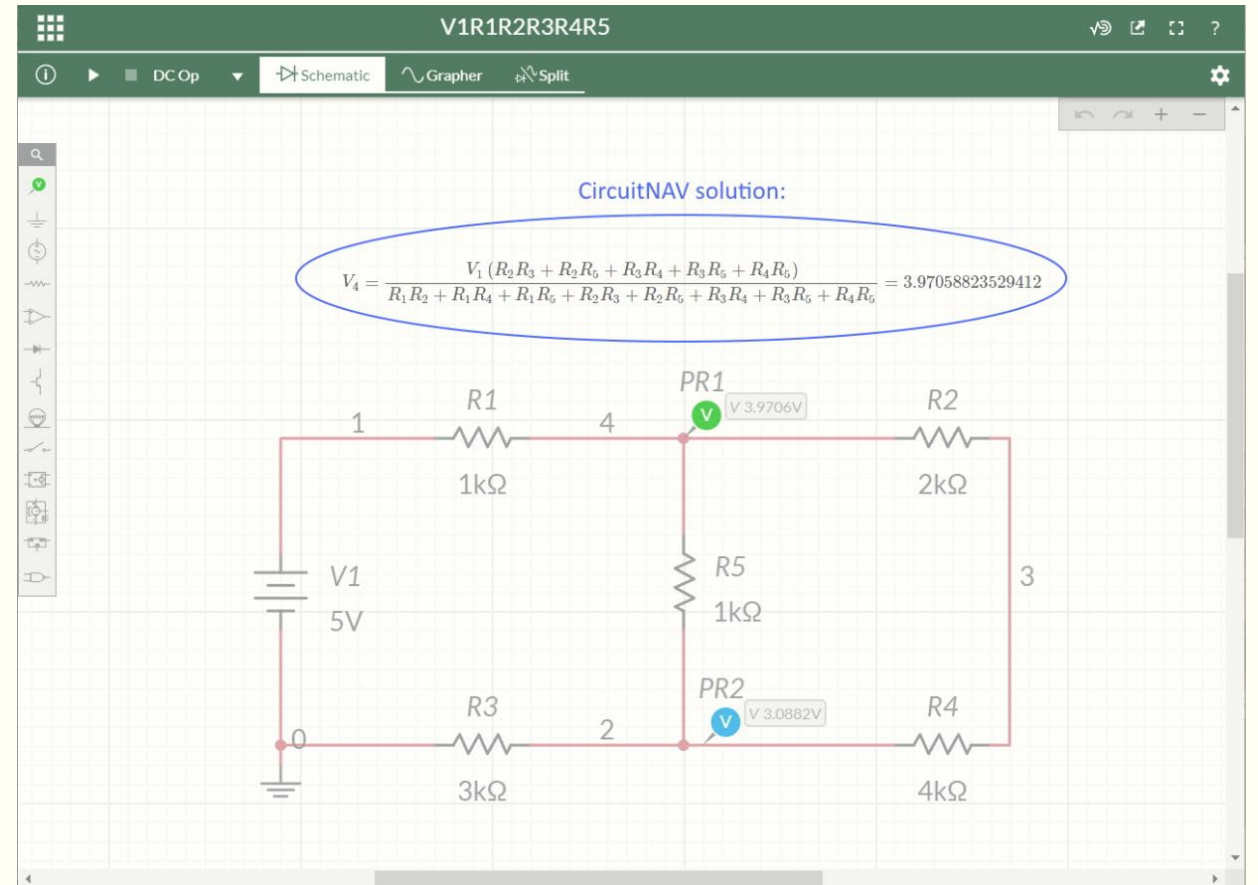


CIRCUITNAV MULTISIM WORKFLOW



Step 1: Draw Schematic in MultisimLive at multisim.com

The screenshot displays the MultisimLive web interface. The browser address bar shows the URL `multisim.com/content/sXWNTrhth4Pa32Nq...`. The main window title is `V1R1R2R3R4R5`. The interface includes a toolbar with icons for simulation (DC Op), schematic, grapher, and split. A left sidebar shows the SPICE Netlist, and a central workspace displays a circuit schematic on a grid.

SPICE Netlist:

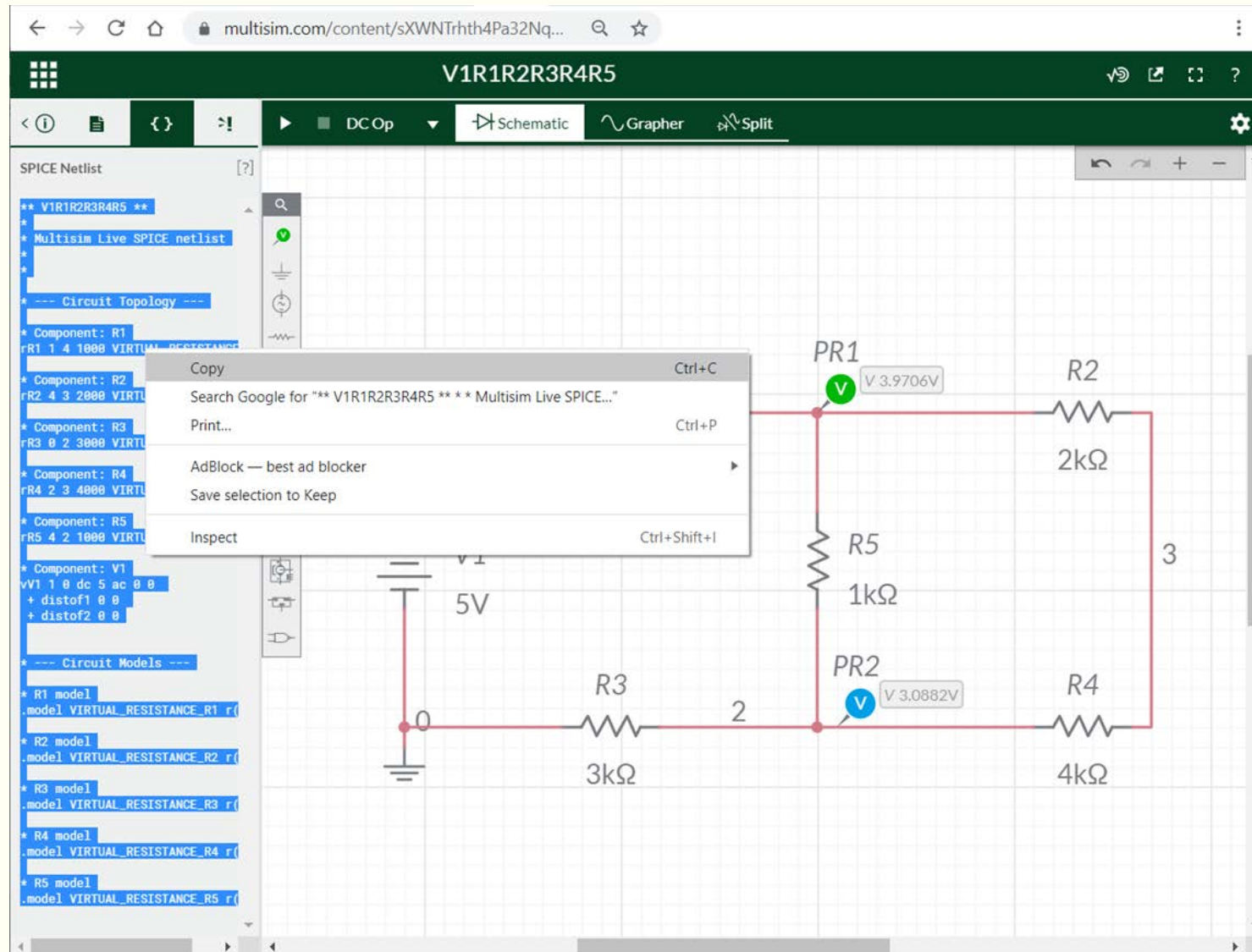
```
[?]  
** V1R1R2R3R4R5 **  
*  
* Multisim Live SPICE netlist  
*  
* --- Circuit Topology ---  
* Component: R1  
rR1 1 4 1000 VIRTUAL_RESISTANCE  
* Component: R2  
rR2 4 3 2000 VIRTUAL_RESISTANCE  
* Component: R3  
rR3 0 2 3000 VIRTUAL_RESISTANCE  
* Component: R4  
rR4 2 3 4000 VIRTUAL_RESISTANCE  
* Component: R5  
rR5 4 2 1000 VIRTUAL_RESISTANCE  
* Component: V1  
vV1 1 0 dc 5 ac 0 0  
+ distof1 0 0  
+ distof2 0 0  
  
* --- Circuit Models ---  
* R1 model  
.model VIRTUAL_RESISTANCE_R1 r(  
* R2 model  
.model VIRTUAL_RESISTANCE_R2 r(  
* R3 model  
.model VIRTUAL_RESISTANCE_R3 r(  
* R4 model  
.model VIRTUAL_RESISTANCE_R4 r(  
* R5 model  
.model VIRTUAL_RESISTANCE_R5 r(  

```

Circuit Schematic:

The circuit consists of a 5V DC voltage source `V1` connected to a network of resistors. The circuit is divided into two loops by a central resistor `R5`. The top loop contains resistors `R1` (1kΩ) and `R2` (2kΩ). The bottom loop contains resistors `R3` (3kΩ) and `R4` (4kΩ). Resistor `R5` (1kΩ) is connected between the nodes between `R1` and `R2` and between `R3` and `R4`. Two probes are connected: `PR1` (green) is connected across `R2` and shows a voltage of 3.9706V; `PR2` (blue) is connected across `R4` and shows a voltage of 3.0882V. Nodes are labeled with numbers 1, 2, 3, 4, and 0 (ground).

Step 2: Select the Netlist on the Left Panel and Copy



The screenshot displays the Multisim software interface. On the left, the SPICE Netlist panel is visible, showing the following code:

```
*** V1R1R2R3R4R5 ***
* Multisim Live SPICE netlist
*
* --- Circuit Topology ---
* Component: R1
rR1 1 4 1000 VIRTUAL_RESISTANCE
* Component: R2
rR2 4 3 2000 VIRTUAL_RESISTANCE
* Component: R3
rR3 0 2 3000 VIRTUAL_RESISTANCE
* Component: R4
rR4 2 3 4000 VIRTUAL_RESISTANCE
* Component: R5
rR5 4 2 1000 VIRTUAL_RESISTANCE
* Component: V1
vV1 1 0 dc 5 ac 0 0
+ distof1 0 0
+ distof2 0 0
* --- Circuit Models ---
* R1 model
.model VIRTUAL_RESISTANCE_R1 r(
* R2 model
.model VIRTUAL_RESISTANCE_R2 r(
* R3 model
.model VIRTUAL_RESISTANCE_R3 r(
* R4 model
.model VIRTUAL_RESISTANCE_R4 r(
* R5 model
.model VIRTUAL_RESISTANCE_R5 r(
```

The main workspace shows a circuit diagram with a 5V DC source (V1) connected to a network of resistors (R1, R2, R3, R4, R5) and two probes (PR1 and PR2). The circuit is configured as follows:

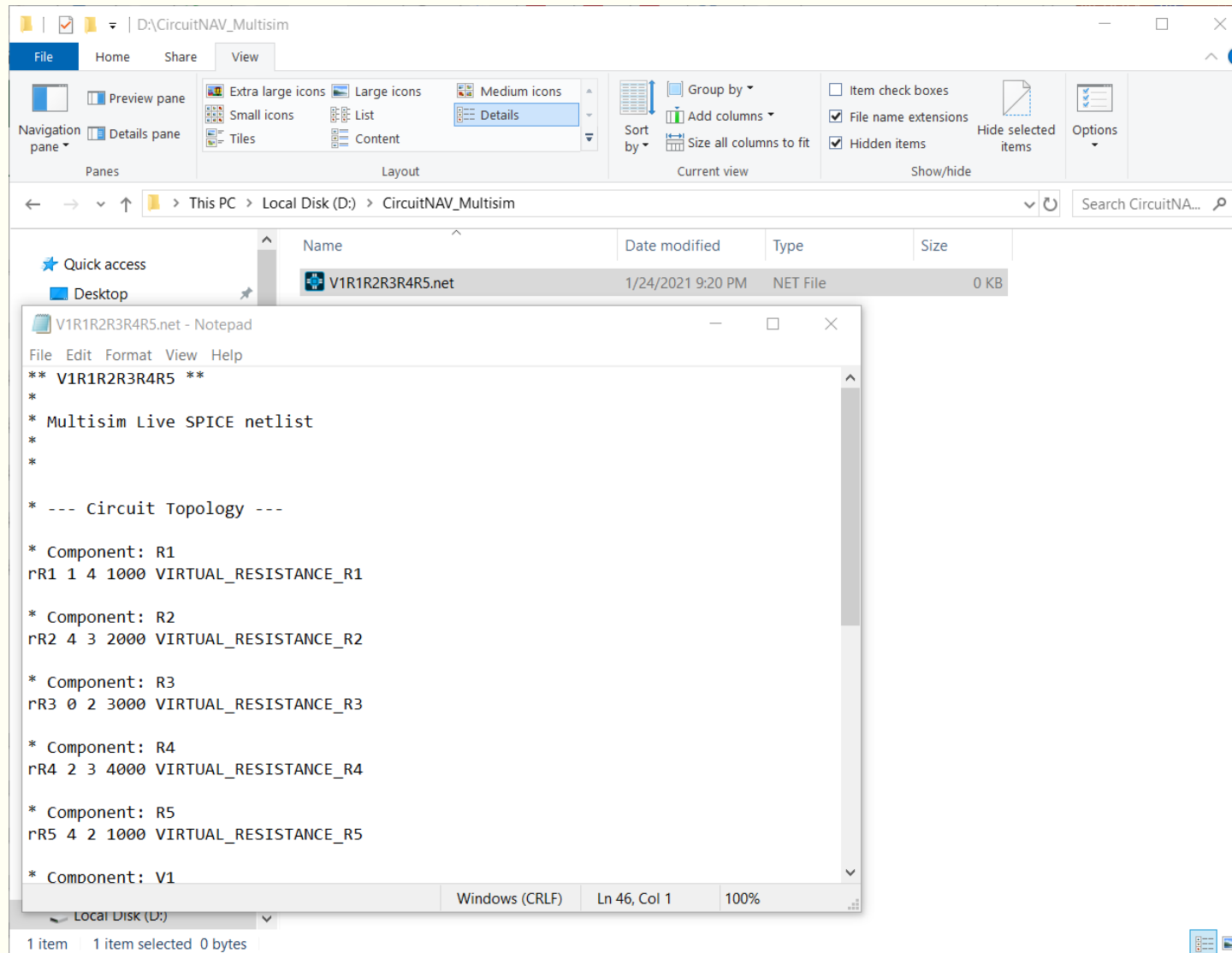
- A 5V DC source (V1) is connected to node 1.
- Node 1 is connected to node 4 via resistor R1 (1000 Ω).
- Node 4 is connected to node 3 via resistor R2 (2000 Ω).
- Node 3 is connected to node 2 via resistor R4 (4000 Ω).
- Node 2 is connected to node 0 via resistor R3 (3000 Ω).
- Node 0 is connected to ground.
- Node 4 is also connected to node 2 via resistor R5 (1000 Ω).

Two probes are connected to the circuit:

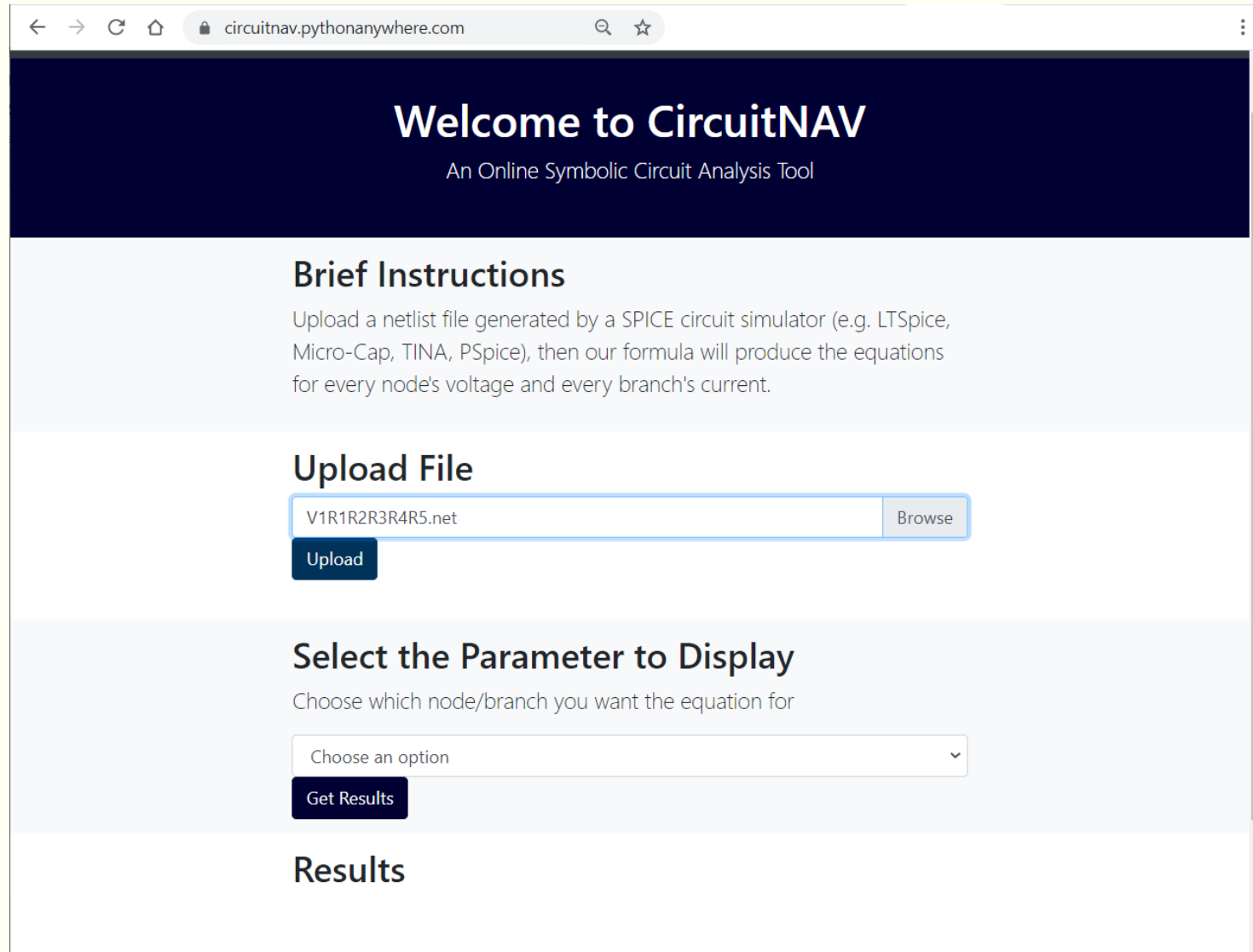
- Probe PR1 (green) is connected across resistor R2, showing a voltage of 3.9706V.
- Probe PR2 (blue) is connected across resistor R4, showing a voltage of 3.0882V.

A context menu is open over the netlist, with the 'Copy' option selected. The menu also includes options for 'Search Google for...', 'Print...', 'AdBlock', 'Save selection to Keep', and 'Inspect'.

Step 3: Paste it into a local *.net File



Step 4: At CircuitNAV, Browse to the *.net File and Upload it



The screenshot shows a web browser window with the URL `circuitnav.pythonanywhere.com`. The page has a dark blue header with the text "Welcome to CircuitNAV" and "An Online Symbolic Circuit Analysis Tool". Below the header, there are three main sections: "Brief Instructions", "Upload File", and "Select the Parameter to Display".

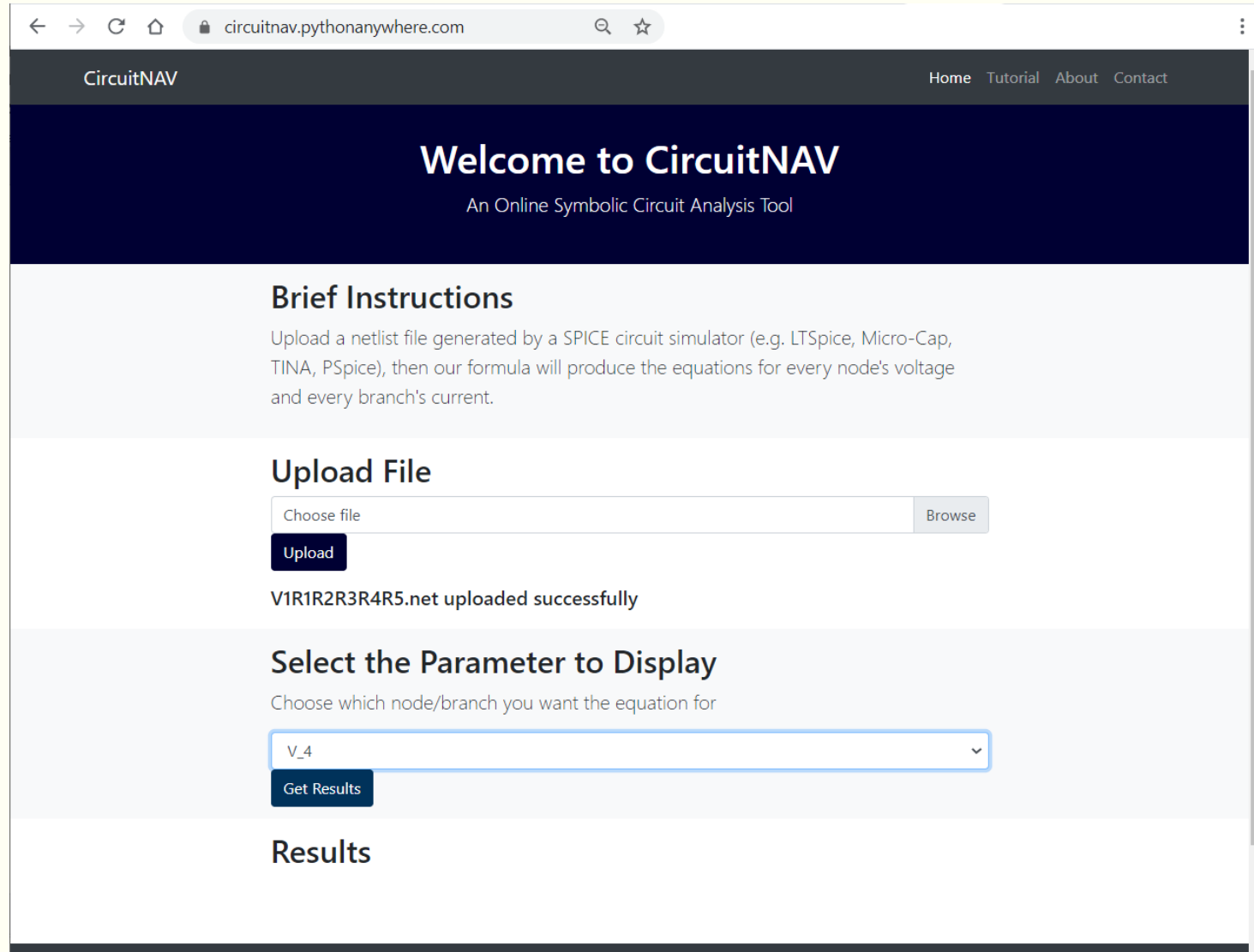
Brief Instructions
Upload a netlist file generated by a SPICE circuit simulator (e.g. LTSpice, Micro-Cap, TINA, PSpice), then our formula will produce the equations for every node's voltage and every branch's current.

Upload File
A text input field contains the filename `V1R1R2R3R4R5.net`. To the right of the input field is a "Browse" button. Below the input field is an "Upload" button.

Select the Parameter to Display
Choose which node/branch you want the equation for
A dropdown menu is shown with the text "Choose an option" and a downward arrow. Below the dropdown menu is a "Get Results" button.

Results

Step 5: Select the Node of Interest and Click on “Get Results”



The screenshot shows the CircuitNAV website interface. At the top, the browser address bar displays 'circuitnav.pythonanywhere.com'. The website header includes the 'CircuitNAV' logo and navigation links for 'Home', 'Tutorial', 'About', and 'Contact'. A dark blue banner contains the text 'Welcome to CircuitNAV' and 'An Online Symbolic Circuit Analysis Tool'. Below this, a 'Brief Instructions' section explains the process of uploading a netlist file. The 'Upload File' section features a file input field with a 'Browse' button and an 'Upload' button. A confirmation message states 'V1R1R2R3R4R5.net uploaded successfully'. The 'Select the Parameter to Display' section includes a dropdown menu with 'V_4' selected and a 'Get Results' button. The 'Results' section is visible at the bottom but is currently empty.

← → ↻ 🏠 🔒 circuitnav.pythonanywhere.com 🔍 ☆

CircuitNAV Home Tutorial About Contact

Welcome to CircuitNAV

An Online Symbolic Circuit Analysis Tool

Brief Instructions

Upload a netlist file generated by a SPICE circuit simulator (e.g. LTSpice, Micro-Cap, TINA, PSpice), then our formula will produce the equations for every node's voltage and every branch's current.

Upload File

Choose file

V1R1R2R3R4R5.net uploaded successfully

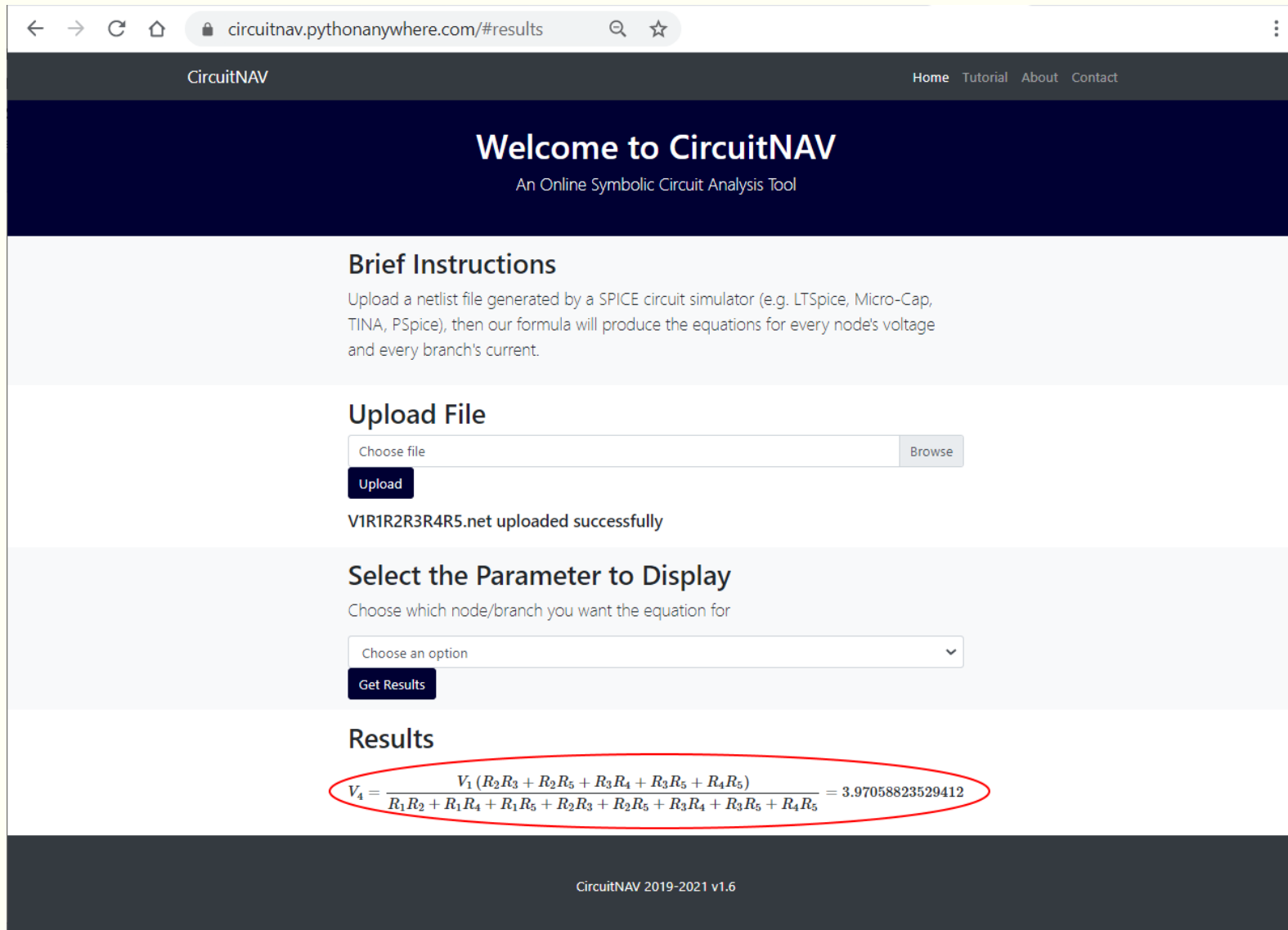
Select the Parameter to Display

Choose which node/branch you want the equation for

V_4

Results

Step 6: Review the Results



The screenshot shows the CircuitNAV website interface. At the top, the browser address bar displays "circuitnav.pythonanywhere.com/#results". The website header includes "CircuitNAV" and navigation links for "Home", "Tutorial", "About", and "Contact". The main heading is "Welcome to CircuitNAV" with the subtitle "An Online Symbolic Circuit Analysis Tool".

The "Brief Instructions" section explains that users should upload a netlist file generated by a SPICE circuit simulator (e.g., LTSpice, Micro-Cap, TINA, PSpice) to produce equations for node voltages and branch currents.

The "Upload File" section shows a file upload interface. A file named "V1R1R2R3R4R5.net" has been successfully uploaded. Below this, the "Select the Parameter to Display" section has a dropdown menu set to "Choose an option" and a "Get Results" button.

The "Results" section displays the calculated voltage V_4 , which is circled in red. The equation is:

$$V_4 = \frac{V_1 (R_2 R_3 + R_2 R_5 + R_3 R_4 + R_3 R_5 + R_4 R_5)}{R_1 R_2 + R_1 R_4 + R_1 R_5 + R_2 R_3 + R_2 R_5 + R_3 R_4 + R_3 R_5 + R_4 R_5} = 3.97058823529412$$

The footer of the page indicates the version: "CircuitNAV 2019-2021 v1.6".