

# winCasino Instructions

Raynald Gauvin et. al., adapted by J. R. Minter

Started: 2020-06-27, Last modified: 2020-06-27

## Contents

<b>Introduction</b>	<b>1</b>
Quick explanation of the <b>generated</b> and <b>emitted</b> electrons . . . . .	1
What is Casino? . . . . .	1
What can CASINO do? . . . . .	3
Why version 2 . . . . .	3
Who wrote CASINO? . . . . .	3
<b>Creating a Simulation</b>	<b>3</b>
Step 1: Creating the Simulation . . . . .	3
Step 2: Creating Layers . . . . .	4
Step 3: Editing Layers . . . . .	5
Step 4: Setting up the Microscope and Simulation . . . . .	5
Step 5: Selecting Distributions . . . . .	6
Step 6: Choosing Physical Models . . . . .	7
Step 7: Options . . . . .	7
Step 8: The End . . . . .	8

[Back to Index](#)

## Introduction

### Quick explanation of the **generated** and **emitted** electrons

Nomenclature is important. This simulation, first suggested by Paul Carpenter (Lehigh 2017) is reproduced in color from a screenshot in the application...

### What is Casino?

Adapted from here to be able to get html and pdf output. Minor reformatting from the original and adding PDF output.

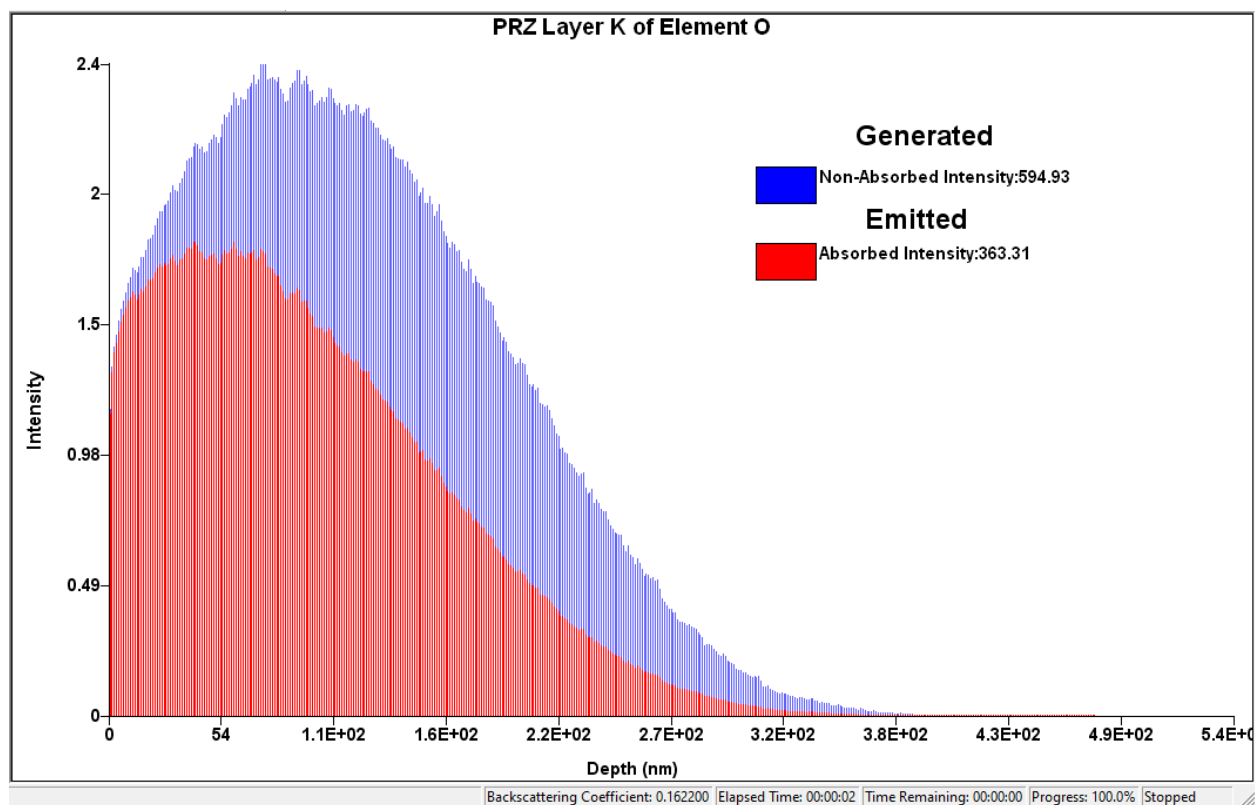
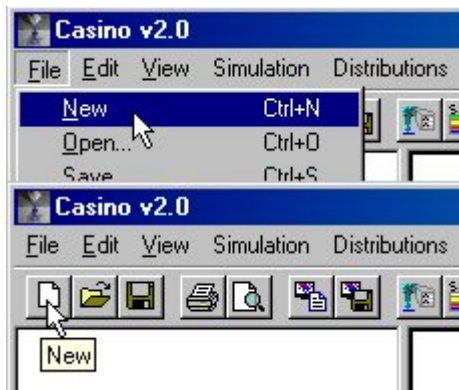


Figure 1: Note the “non-absorbed” electrons are the **generated** electrons and the “absorbed” electrons are **emitted** electrons



The CASINO acronym has been derived from the words “monte CARlo SIMulation of electroN trajectory in sOLids”. This program is a Monte Carlo simulation of electron trajectory in solid specially designed for low beam interaction in a bulk and thin foil. This complex single scattering Monte Carlo program is specifically designed for low energy beam interaction and can be used to generate many of the recorded signals (X-rays and backscattered electrons) in a scanning electron microscope. This program can also be efficiently used for all of the accelerated voltage found on a field emission scanning electron microscope (0.1 to 30 KeV).

## What can CASINO do?

This program is designed to simulate a large amount of electron trajectories in a solid of your choice. The main idea is to simulate enough electron trajectories to represent the condition used to image structures in a scanning electron microscope (SEM). Thus it is possible to predict theoretically the signals observed in the SEM. This Monte Carlo program use different models to simulate the interaction of electrons with a solid. For now, the versions 2 of CASINO produce the following signals: backscattered coefficient and X-ray. It also handles those geometry : muti-layers samples and grain boundary.

## Why version 2

Version 1 of CASINO was starting to show some age with it's DOS based interface. A new Windows™ based interface has made CASINO much easier to use.

## Who wrote CASINO?

Version 1:

The Monte Carlo program CASINO has been programmed by the research team of Raynald Gauvin (Ph.D., Full professor at Université de Sherbrooke, Québec, Canada). The program has been initially programed by Pierre Hovongton, Ph.D. and Dominique Drouin, Ph.D. in 1996 and it has been updated by Paula Horny, M.Sc.A. student and by Hendrix Demers, undergraduate student in 1999. All this work have been performed under the supervision of Professor Raynald Gauvin.

## Creating a Simulation

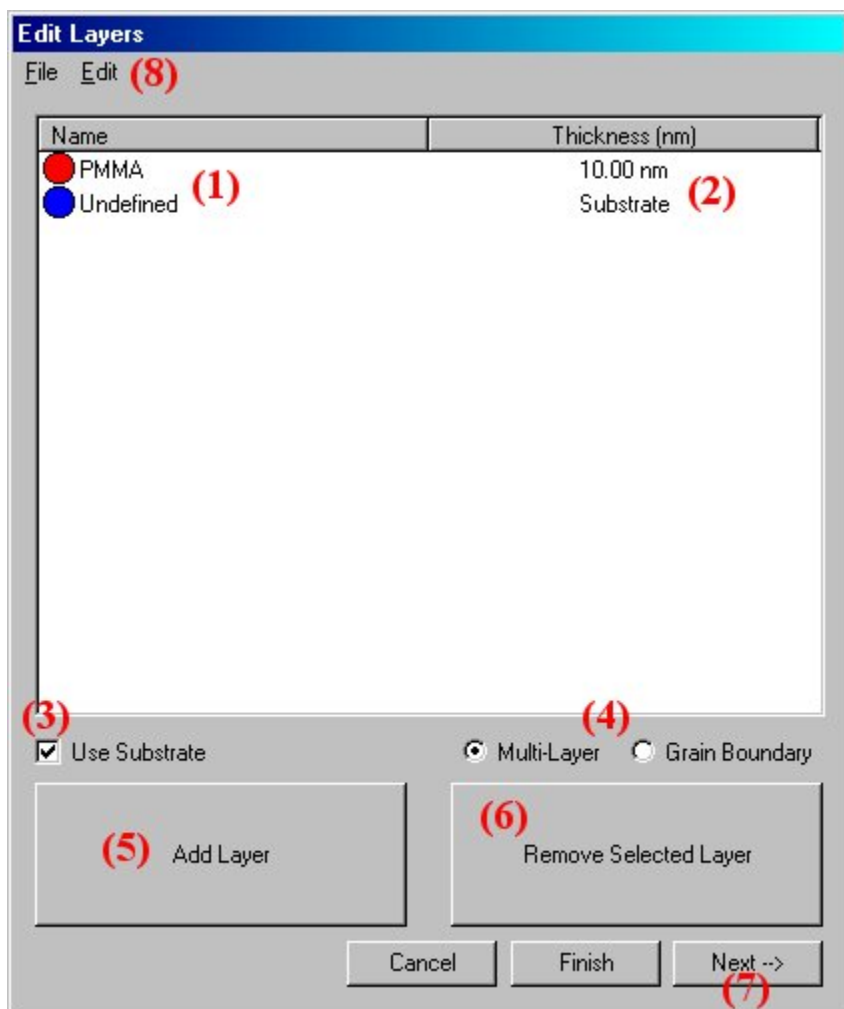
### Step 1: Creating the Simulation

To create a new simulation select File/New from the menu or click of the  icon.

The NEW SIMULATION WIZARD will appear on the screen.

## Step 2: Creating Layers

The first dialog box will be the EDIT LAYERS DIALOG. In this dialog you will create the layers of your sample.



To add a layer, you simply press the ADD LAYER BUTTON (5).

If you add too many layers, you can remove them by selecting them and pressing the REMOVE SELECTED LAYER BUTTON (6) or by pressing DEL on the keyboard.

The NAME and CHEMICAL COMPOSITION of each layer can be defined by double-clicking the names of the layers (1). This will bring up the LAYER CHEMICAL COMPOSITION DIALOG which is described in Step 3.

If you wish to define the last layer as a substrate, make sure the USE SUBSTRATE CHECKBOX (3) is checked.

If your sample is multiple horizontal layers, select MULTI-LAYER (4), else if you sample is composed of vertical layers select GRAIN BOUNDARY (4).

To change the thickness of the layers simply click on the current thickness (2) and type in the new thickness.

You can copy and paste layers by selecting them and using the EDIT menu (8).

When your sample is completed you can press Next --> (7) to move on to the next dialog MICROSCOPE AND SIMULATION PROPERTIES which is explained in Step 4.

### Step 3: Editing Layers

This dialog box will let you give a name and a chemical composition to a layer.

Start by entering the chemical composition in the COMPOSITION EDIT BOX (1).

Layer Chemical Composition

Name: PMMA (2) Add to Library (3)

Composition: COH (1)

Density: 1.188 (4)

Element	Z	Weight Fraction	Atomic Fraction
✓ C	6	0.599848	0.333333
✓ O	8	0.319614 (5)	0.133333
✓ H	1	0.080538	0.533333

☒ User Defined Density (7) Auto-Complete Element Fractions (6)

☒ User Defined Distributions (7)

Done

The elements must be written in Uppercase for the first letter and lowercase for the other, example (Fe2O3). Then give the layer a name (2), if no name is given, the composition will be chosen as the name.

The density (4) will **automatically be calculated** from the atomic weight and fractions, this calculation does not take into account the structure of the material and you may want to **\*\*change the density to it's actual value\***.

If you wish to **change** the atomic fractions or weight fractions manually then simply click (5) on the value you wish to change. To auto-complete an element as the remainder of the composition simply press the **AUTO-COMPLETE ELEMENT FRACTIONS BUTTON** (6) after selecting the element.

To save a composition for future use, give it a name and press **ADD TO LIBRARY BUTTON** (3). This will store the information so that next time you enter a name for a layer and it exists in the library it will set the layer to the saved layer.

To reset values to their automatically calculated states, just **un-check** the boxes **USER DEFINED DENSITY AND DISTRIBUTION** (7).

Repeat Step 3 for each layer and then move on to Step 4.

### Step 4: Setting up the Microscope and Simulation

This dialog lets you select the parameters of your simulation.

Microscope and Simulation Properties

Starting accelerating voltage (keV): 1

Ending accelerating voltage (keV): 1 (1)

Step (keV): 1

Number of electrons to simulate: 200 (2)

Polar angle of the incident beam (degree): 0 (3)

Beam diameter (nm): 10

Take-Off angle of X-Ray detector: 40 (4)

☒ Scan the beam to create an images

From: -10 To: 10 (5) Steps: 1

<-- Previous Cancel Finish (7) Next --> (6)

First, enter the **energy** used for your simulation (1).

The first box is where you set your **starting energy**, the second is the **energy where you want to stop the simulation** and the third box contains the **step in KeV** between each simulation.

If the ending energy is smaller or equal to the starting, then the step will be ignored and only one simulation at the starting energy will be done.

Second, enter the **number of electrons** you want to simulate (2). The bigger the number, the better the results but the longer the simulation.

Third is the **beam parameters** (3), the first box represents the **angle the electrons** will be fired into the sample and the **second box** contains the **width of the beam**.

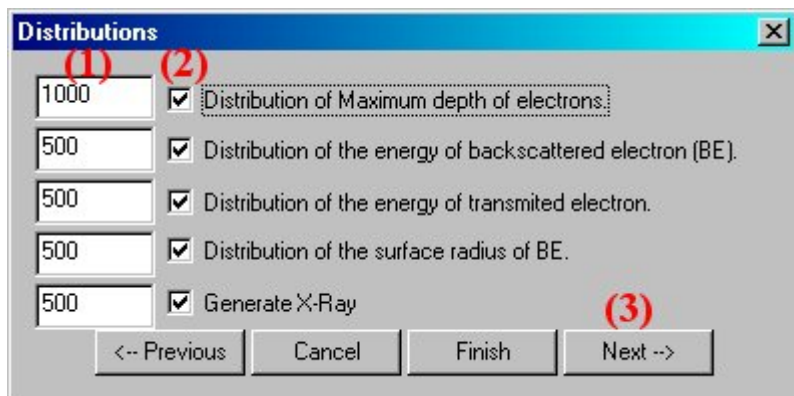
Fourth, define the **angle of your X-Ray detector** (4) with the positive X-axis of the sample being 0 and moving counterclockwise towards the negative X-axis being 180.

Finally, set the **limits of your scan** (5). The sample is centered at 0 nm. The step is in **nanometers**. If you do not wish to scan your sample, **uncheck** the box **SCAN BEAM TO CREATE AN IMAGE**. The **FROM** box then contains the **position of the beam on the sample**.

At this point you can already press **FINISH** (7) and move on to Step 8, but you can **continue through** the rest of the options by pressing **NEXT-->** (6) and moving on to Step 5.

## Step 5: Selecting Distributions

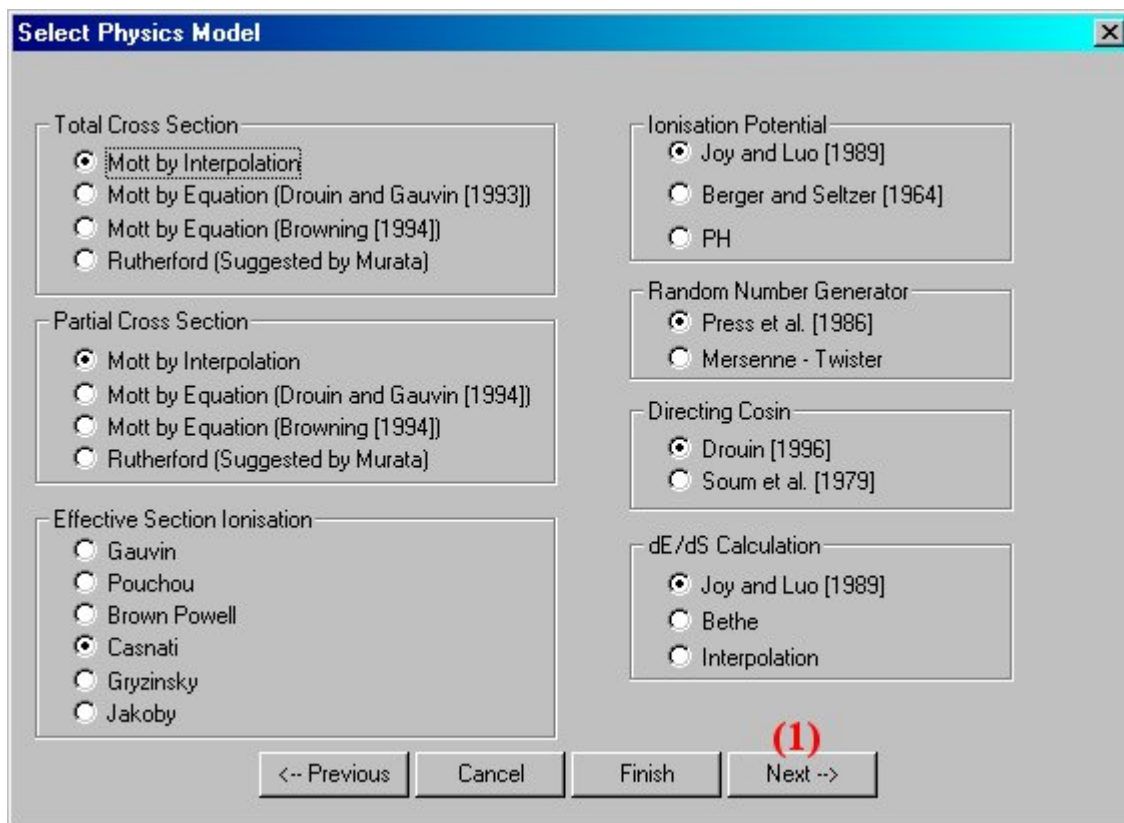
Here is where you set up your distribution.



Check the box for each distribution you wish to see after calculations (2). Enter the number of points you want each distribution to have (1).

Move on to the next step (3).

## Step 6: Choosing Physical Models



Unless you know which model does what, skip to the next step (1).

## Step 7: Options

(1): Conserving the trajectories takes up a lot of memory. Conserving NONE will not save any trajectories to memory which means that there is no way to view the trajectories after they have been calculated.

DISPLAYED ONLY will only keep the trajectories that have been displayed on the screen, this option is good if you want to see the trajectories and change some of their viewing parameters and ALL is used if you want to

keep all the trajectories in memory, display a selected few during calculation but after calculations it will display all of them.

(2): Number of electron trajectories to display on the screen during calculations.

(3): Minimal time between backups which are used to continue simulation in case of accidental termination of the simulation.

(4): The minimal energy, in KeV, at which a trajectory is terminated.

(5): If checked the trajectories will be displayed at regular intervals during the simulation, otherwise the first trajectories are displayed.

(6): Press FINISH when your simulation is ready to be calculated on go to Step 8.

## **Step 8: The End**

After pressing FINISH you will be asked if you want to begin the simulation, press YES if you wish to otherwise press NO.

[Back to Index](#)