

# Graflog User's guide

*Command line & Web-based tool to graph the results obtained with the CD++ toolkit.*

<http://www.sce.carleton.ca/faculty/wainer/wbgraf/index.html>

## Table of contents

1	Introduction .....	1
2	Command-line Graflog .....	1
2.1	Installation .....	1
2.2	How to obtain an output file.....	1
2.3	Running drawlog .....	2
2.4	Running graflog.....	2
2.5	The palette file .....	3
2.6	Important notes .....	4
2.7	Example: Fire model .....	4
3	Web-Based Graflog .....	5
3.1	Installation .....	5
3.2	How to obtain the results file.....	5
3.2.1	Downloading a results file from the site.....	5
3.2.2	Obtaining your own results file from a simulation.....	5
3.3	Executing the applet to graph the results.....	5
3.3.1	Choosing the Output file.....	5
3.3.2	Choosing the Model file .....	6
3.3.3	Defining the intervals.....	6
3.3.4	Defining the color palette.....	7
3.3.5	Viewing the results .....	7
3.3.6	Other options.....	8
4	More information .....	8

## 1 Introduction

This manual contains reference information about Graflog, a component of the CD++ toolkit. Graflog was developed to run in two versions: command-line and web-based. Both versions will be covered here.

## 2 Command-line Graflog

### 2.1 Installation

Uncompress the zip file into a working directory with WinZip. 386 or faster CPU, MS-DOS, a compatible VGA/SVGA display, 4MB RAM and 10MB hard disk available.

### 2.2 How to obtain an output file

When a simulation is executed, an output file must be generated. This file will be used later by *graflog* to create a dynamic execution graph. If we run the simulator of a model called *model.ma*, the simulation could be executed as:

```
simu -mmodel.ma -loutput.log ...
```

This generates a file *output.log* that can be used by *drawlog* (part of the CD++ toolkit). Drawlog will parse the file and convert it to a grid with numeric values on it (see details about the Drawlog tool in the CD++ User Manual).

### 2.3 Running drawlog

The Drawlog generates a text file readable by users. It includes borders of the planes being drawn, the simulation time, and other extra information. If you intend to draw the resulting graphs with Graflog (or the On-line Web-based Graflog), you should avoid the formatting of the outputs. This is done by using the switch *-f#*, which creates an output grid without any format at all. Consequently, only the values of each cell are displayed. It is important to consider that *graflog* only displays two-dimensional models. Therefore, the parameter *#* indicates the plane we want to show if an n-dimensional model was simulated.

If we consider our previous example, we could execute:

```
drawlog -mmodel.ma -loutput.log ... -f1 > model.drw
```

As we can see, the drawlog output has been redirected to the file *model.drw*. Once we have this file, we can graph the results properly with *graflog*. This file is viewable also with the on-line tool (see section 3).

### 2.4 Running graflog

To execute the *graflog*, you can use the following parameters:

**-x#** where *#* is an integer number, and it is the model's grid height

**-y#** where *#* is an integer number, and it is the model's grid width

**-e#** where *#* is the scale. Scale 1 means 1x1 pixel. Scale 2 means 2x2 pixels and so on.

**-fxxxx** where *xxxx* is the drawlog output file (in the above sample it is *model.drw*)

**-exxxx** where *xxxx* is the filename that contains the color palette.

**-s#** where *#* is the number of advances over the time. For example our model actualizes the complete grid every *t* units of time, and we execute the simulation for 10\*t units of time. If we execute *graflog -s2* then the graph will run until time 2\*t and then it will stop waiting for the user to press a key to continue. When a key is pressed, it will show the graph as usual, until time 4\*t arrives and it will stop again, and so on. Hence,

if the switch `-sI` is used, it will generate the graph in a step-by-step fashion. If this switch is omitted, the graph will continue until it reaches the end of the simulation.

`-t#` similar to the `-s` switch, but advances # units of time and then step-by-step.

`-v` displays the grid

Note: if ESC key is pressed, then the graph will be displayed without any user intervention, overriding the switches used for the execution.

## 2.5 The palette file

The palette file contains a table with the following format:

X1	Y1
X2	Y2
X3	Y3
...	...
X30	Y30

Where **X** is a range that represents the possible values of the cell and **Y** is the color assigned to the given range. Range's formats are described as `[###;###]` or `(###;###)` or `[###;###)` or `(###;###]`, where `###` is any real number as well as `+INF` and `-INF`

Examples:

`(-INF;20)`

`[-23;+INF]`

`[0;0]`

`[45;47.8994004)`

Up to 30 assignments can be made in a file. The valid colors are shown below.

black	0
blue	1
green	2
cyan	3
red	4
magenta	5
brown	6
light gray	7
dark gray	8
light blue	9

light green	10
light cyan	11
light red	12
light magenta	13
yellow	14
white	15

## 2.6 Important notes

1 – Default values:

```
x = 100
y = 100
e = 1
s = no interruptions
f = "salidaF.drw"
c = "color.txt"
```

2 – A color palette must always be used. Otherwise, no results will be displayed.

## 2.7 Example: Fire model

To execute the simulation:

```
simu -mFireII.ma -t02:00:00:000 -lFire.log
```

To obtain the output file using *drawlog*:

```
drawlog -mFireII.ma -cForestFire -lFire.log -f1 > output.drw
```

To generate the graph using *graflog*:

```
graflog -x30 -y30 -foutput.drw -cFire.col -e5 -s1
```

## **3 Web-Based Graflog**

### **3.1 Installation**

There is no installation of the applet on your system. However, you have to accept the applet's certificate in order to grant some file access permissions. It will read information about the model and the simulation from your file system. You just have to open your browser and go to:

<http://www.sce.carleton.ca/faculty/wainer/wbgraf/index.html>

Click on "Model Execution" and follow the instructions described in this manual. To run this application you need a Java-capable web browser such as Internet Explorer 3.0 or Netscape 3.0 or higher.

### **3.2 How to obtain the results file**

You have to obtain an output file to visualize the results later using the applet. This section describes briefly how to generate and also customize this file. You can choose between 2 ways of obtaining a result file to use with the applet, and both will be described here.

#### ***3.2.1 Downloading a results file from the site***

On the Cell-DEVS Website you will find several existing models. Many of them include a results file, and those can be used to graph the results with the applet.

#### ***3.2.2 Obtaining your own results file from a simulation***

The output file must be generated when running the simulation. Read details to generate these files in section 2. As a result, the output files obtained can be used to view the results with the applet.

### **3.3 Executing the applet to graph the results**

Once we have the results file obtained as described above, you can run the applet pressing the Model Execution link. You will be asked to accept a certificate. Do so, to grant access to the applet to the files you will be using. The steps required to view the simulation are described below.

#### ***3.3.1 Choosing the Output file***

Choose the output file containing the results of a Cell-DEVS simulation obtained as described in the previous section using the "Choose Output File" button. In the given example, that file would be *output.drw*.

### 3.3.2 *Choosing the Model file*

Select the model file (usually the extension used is ".ma"), which defines the model specification used for that model. The applet will use the model definition found there to draw the model grid.

### 3.3.3 *Defining the intervals*

Select the intervals chosen for each of the output values. Each range of values corresponds to a different color that will be used to display the cell's values. The ranges can be defined manually, loaded from a saved .pal file (it stores the colors and ranges and it's provided with some samples and may be even created by you) or by selecting the predefined ones.

To define the ranges manually:

- i) Click the "Choose Intervals + colors" button on the main window
- ii) Click on each textbox to write the minimum and maximum values associated to the interval
- iii) Repeat step ii) until all the intervals have been chosen
- iv) Click "Ok" button.

To use a previously saved .pal file, that includes the colors and ranges:

- i) Click the "Choose Intervals + colors" button on the main window
- ii) Click on the "Load from .pal" button
- iii) Choose the .pal you want to use
- iv) Click "Ok" button.
- vi) Now you may continue to section **3.4.5** (Viewing the results) because the colors have also been retrieved from the file.

To select one of the predefined ranges simply:

- i) Click the "Choose Intervals + colors" button on the main window
- ii) Select one of the 5 predefined intervals by clicking on them (from 1 to 5). You can see each of the predefined set of ranges.

These ranges must correspond to the different values stored in the output file, in order to properly group the values by colors. The predefined intervals have been chosen to apply correctly to some of the existing sample files, for your convenience.

### 3.3.4 *Defining the color palette*

You can select the colors palette manually, use a previously saved .pal file (stores the colors and ranges and it's provided with some samples or may be previously saved by yourself as described before) or select the predefined ones.

To define the colors manually:

- i) Click the "Choose Intervals + colors" button on the main window
- ii) Select the color you prefer to the first interval (you will see the selected color in the rectangle located on the lower-right corner of your screen. You can select the color in several ways:
  - clicking on the big palette box in the Color Dialog. The right-side bar allows you to specify the luminosity of the selected color by dragging or clicking.
  - typing the RGB (red, green, blue) in the text boxes. You can use the RGB to combine the three color intensities (valid values are from 0 to 255)..
  - typing the HSL (hue, saturation and luminosity) in the text boxes (valid values are from 0 to 255).
- iii) Click the button "Assign this color to the interval"
- iv) Repeat steps ii) and iii) until all the ranges have an associated color
- v) Click "Ok" button.

To use a previously saved .pal file, that includes the colors and ranges:

- i) Click the "Choose Intervals + colors" button on the main window
- ii) Click on the "Load from .pal" button
- iii) Choose the .pal you want to use
- iv) Click "Ok" button.

To select one of the predefined palettes simply:

- i) Click the "Choose Intervals + colors" button on the main window
- ii) Select one of the 5 predefined palettes by clicking on them (from 1 to 5)
- iii) Click "Ok" button.

Now, you can save the ranges and colors that are being used, by clicking the "Save to .pal" button in the Color Palette Window.

### 3.3.5 *Viewing the results*

To view the simulation, click the "Play" button.

You can always "Pause" the simulation, and even Stop and re-play it again many times.

### 3.3.6 *Other options*

*View grid:* shows/hides the associated grid

*Show values:* shows/hides the associated values for each cell

*Speed combo:* You can choose between three different speed rates (low/medium/high) to view the simulation at your preferred pace.

*Step increment:* When “Step by 1” is selected, the results will be shown completely, without skipping any frame. However, you can skip frames in the simulation by choosing a step increment greater than 1 from the combo list box.

## 4 **More information**

You can send comments, suggestions and report problems to: <gwainer@sce.carleton.ca>