Manual Of the GUI for the CD++ Tool

Authors: Jason Wu, Yolanda Wang (C.S. Dept.; Carleton University; 2001).

Supervisor: Prof.Gabriel A. Wainer (SCE Dept; Carleton University)

The manual describes in detail how to operate a GUI with several graphical inputs and outputs for DEVS models, including shows the execution results of Cell-DEVS models, Atomic-DEVS models and Coupled-DEVS models. This GUI is platform portable, it could be run on various environment such as JBuilder and JDK1.1 above. Java 1.3 version is needed to run the application. The version is Beta, therefore, if any errors are found, please report them to: gwainer@sce.carleton.ca.

To run the program, decompress the **finaldevs.tar** file (tar –xvf finaldevs.tar; or use Winzip under Windows environments). All source code and some necessary files will be in the directory **devs**; all the class files will be in **devs/class/**. Using the command line, set up the class path. For example, for MS-DOS:

set classpath=.;.\devs\class

Then press:

java devs.mainDEVS

Then the GUI will be shown on the screen.

If you experience any problems, read the manuals for your Java compiler

1. Introduction

This manual describes a GUI, which is used for creating atomic models and coupled models for the ND-C++ tool. The basic function of the GUI includes: create atomic model, create coupled model, retrieve parent class of the coupled model, and run external DOS-style command. The GUI also includes a simple text editor. The GUI is coded by Java, which enable it to run on various platforms (however, users can only run external command under Windows. The reason for this is explained in the following sections). The following sections explore the functions of the GUI with examples.

The design space is used for the user to define the model. Before creating the model, the user should select proper design space for the corresponding atomic model and coupled model by click on the tabs, which is indicated as “design space selector” in Fig 1. The information space helps the user to know the details of the model created on the design space.

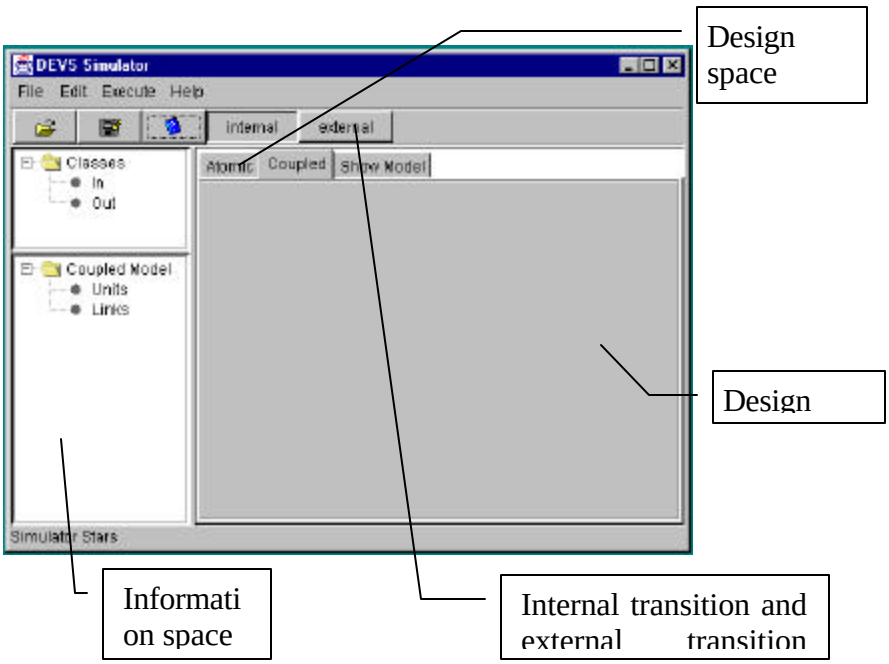


Fig 1

2. Create Atomic Model

The following section describe the basic steps that is needed to create an atomic model with the name “aex_a”.

- Select “Atomic” Tab from the Interface.
- Select “File|New” from the main menu of the Interface.
- Click right button of the mouse on the design space and a pop up menu will be shown. Fig 2 shows the result.

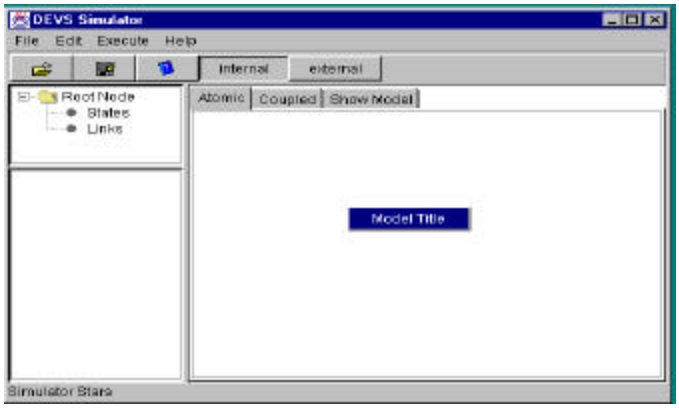


Fig 2

- Click the “Model Title” item. A “Set Title” dialog will be shown like Fig 3.

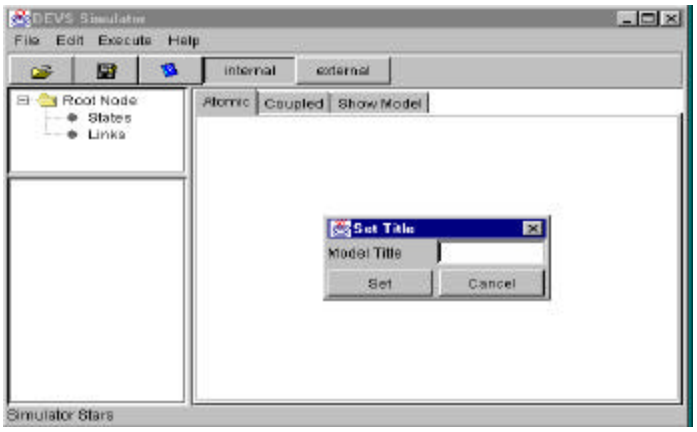


Fig 3

- Fill “aex_a” in the blank area and click “Set” button.
- Double click left mouse button on the design space, an unit will be drawn with blue color. Double click left mouse button on the unit, the unit will be selected with red color. Double click left mouse button on the selected unit, the unit will be un-selected and turns to be blue again.
- Click right mouse button on the unit, a popup menu will be shown with various functions connected to the unit. Select the “property” item, a “state properties” dialog will be shown. Fill the dialog, set state ID and TL for the state, click OK button.
- Click right mouse button on the unit and bring the popup menu. Select “add port” item and an “Add Port” dialog will be shown. Fill the dialog, select proper properties for the port, click OK button. Repeat the procedure and add all of the necessary ports to the unit. Fig 4 shows the result.

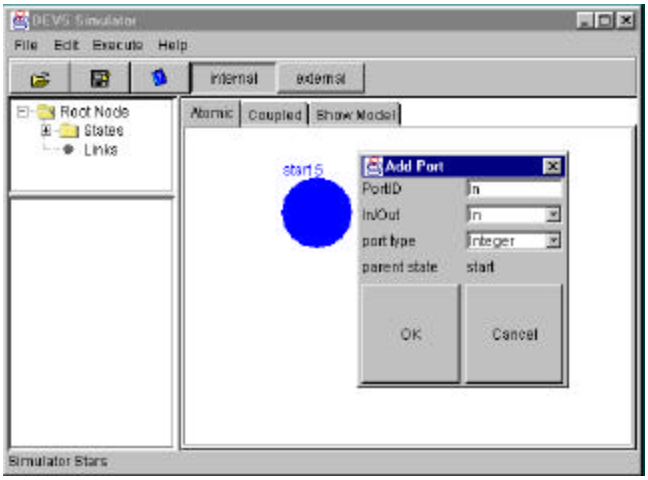


Fig 4

- Click on the nodes of the tree in the information space and expand the tree. The detailed information of the model can be shown like Fig 5.

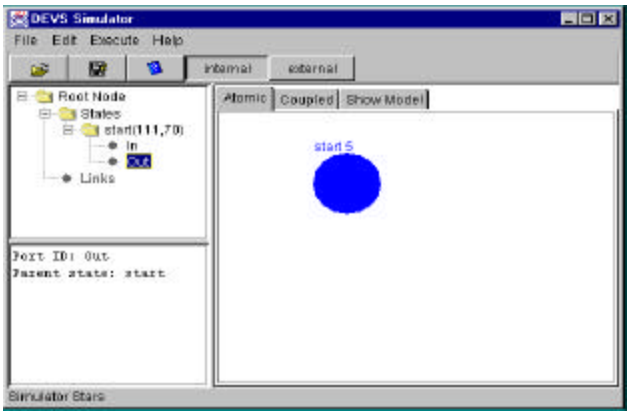


Fig 5

- After creating all of the necessary units according to the steps described in the above, we can draw links between units, which represent the transition functions. The links in the atomic model are divided into two category: internal and external, which represent internal transition and external transition. Before we draw a link, the user should selected desirable link type by clicking “internal” or “external” button on the toolbar of the Interface.
-
- To draw a link between two units, press the left button on one unit, hold the button and drag the mouse to another unit, release the button. A link with pre-selected type will be drawn. A small solid circle on the link represents the direction. Fig 6 shows the result.

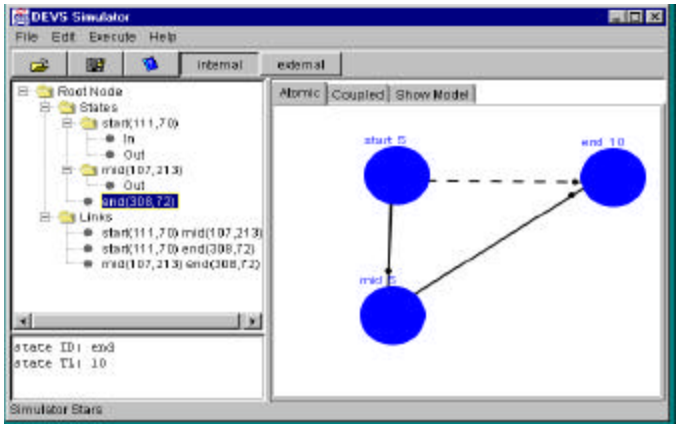


Fig 6

- After drawing the links, we need connect the link with the port. To do this, click the right on the target unit, select “Port@Link” item from the popup menu. A “Connect Port to Link” dialog will be shown. Select the target link and target port from the dialog, fill the value and click the connect button. Fig 7 shows the result.

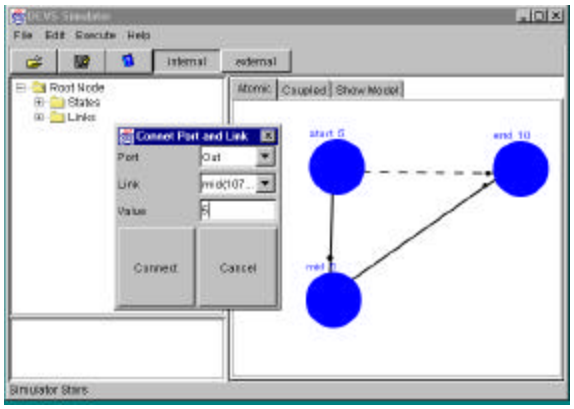


Fig 7

- To delete units or links, select the unit by double click on it or select the link by double click on the middle of the link, which will turn the selected units and links to be red. Then press the "Delete" key of the key board or select "Edit|Delete" from the main menu. All the selected links and units will be deleted. When an unit is deleted, all the links and ports connected to it will also be deleted. To delete a port from an unit, click right button on the unit the select the "delete port" item from the popup menu. A "Delete Port" dialog will be shown. Select the port to be deleted from the dialog and click the delete button.

- After finishing drawing the model, we need to save the model and export the model to be a standard ".ma" file that can used by the ND-C++ tools. To save the model, we select "File|Save" or "File|Save As" from the main menu, a file save dialog will be shown, select a desirable directory and give the file a name. The file name must use ".djf" as its extension.

- To export the model to be a standard ".ma" file, select "File|Export" from the main menu, a save file dialog will be shown. Give the file a name. The name of the file must be as same as the file that is saved as"xxx.djf" but with ".ma" as its extension. For example, a "aex_a.djf" file corresponds with a "aex_a.ma" file. Usually, we save the "xxx.ma" file into the same directory as the "xxx.djf" file.

- After we create the atomic model and save it to a file, we can open it later by selecting "File|Open", and open the corresponding ".djf" file. (note: the saved model can not be opened by reading the ".ma" file.)

3. Create Coupled Model

The basic steps to create a coupled model are as same as creating an atomic model. In the following section, we will only highlight the instinct procedure for the coupled model.

- The coupled model needs to be created on the coupled model work place, which can be selected by click on the “Coupled” tab above of the work place.
- Select “File|New” from the main menu will bring a new work place for the user to work on.
- To draw a coupled model, we need to input the necessary parent class first, since every unit in the coupled model is the instance of its parent class. To do this, select “File|Input” when we want to input the model, which is created by this GUI or has a standard “.ma” file. A file input dialog will help the user the select the desirable input file. Only the “.ma” file can be input as a parent class for drawing the units of the coupled model. If we want to input classes that are predefined in the “register.cpp” file, “File|Input from Register” should be selected and open the target “register.cpp” file with the help of the open file dialog. All the input classes are shown on the “classes tree” in the information space.
- To draw an unit of the coupled model, first, we need to select the parent class. The parent class can be selected by click the corresponding node of the “classes tree” in the information place. After a parent class is selected, we can draw an unit on the work place just as what we have done in the atomic model. Fig 8 shows the result.

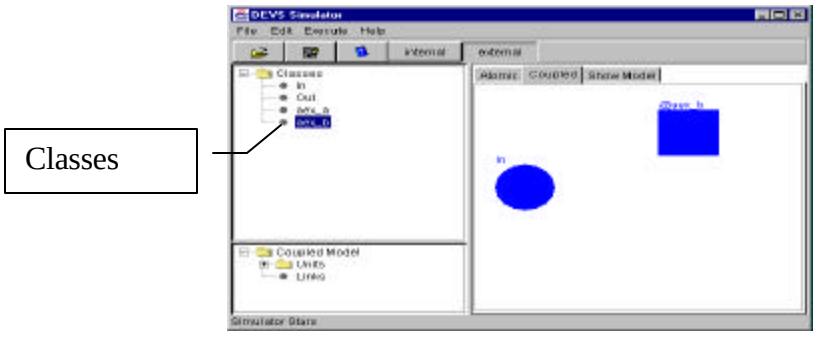


Fig 8

4. Retrieve the Parent Class of an Unit of the Coupled Model

Since every unit of the coupled model is an instance of one class, we can show the parent class model if the model is created by the GUI and saved as a “.djf” file.

- To show the parent class model of an unit of the coupled model, open the coupled model in the coupled model design space, and double click on the unit. Only one unit can be selected before we show the parent class model, which means only one unit is in red color after it is selected.
- Click the “Show Model” tab above of the design space, select “Edit|show parent model” from the main menu. The parent class model will be shown on the “Show Model” design space.
- If the parent class model has nested unit, which is a coupled model itself, we can show it by double click on the unit. When the unit is selected, its color turns to be red, select “Edit|show parent model” from the main menu. **Note: only one unit can be in**

the selected mode from both the "Coupled" space and "Show Model" space, which means only one unit of the two design spaces can be in the red color before we show the parent class model.

5. Run External "Simu" Command

If the user has an pre-defined ".ma" file, which describe a DEVS model, the model can be simulated by running the external "simu" command from the GUI. In order to begin the simulation, the following steps should be followed:

- Write an "xxx.bat" file as follows:
  start %1 %2 %3 %4 %5 %6 %7 %8 %9
  exit
  Save it. For example, we save it as "Run.bat".
- Select "Execute|Run" from the main menu, a "Run" dialog will be shown as Fig 9.
- Click the "Simu" button in the dialog, an open file dialog will be shown. Choose the external "simu.exe" file and open it.
- Select desirable parameters from the list box, and fill corresponding value in the value field.
- Click OK after assigning values to all necessary parameters. An open file dialog will shown again, choose the "xxx.bat" file we create in the first step and open it. The simulation will begin if we have a correct ".ma" file.

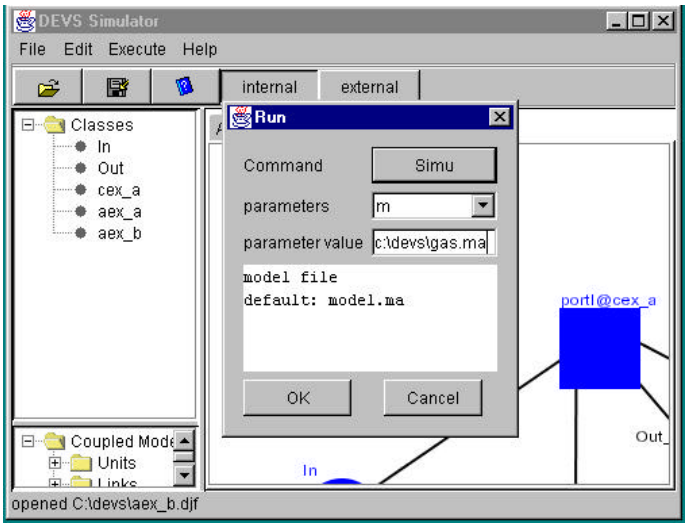


Fig 9

6. Run Text Editor

- To run the simple text editor, select "Execute|Run" from the main menu, a text editor will be shown.

- The user can also run a external text edit by selecting “Execute|Run”. In the “Run” dialog, click the “Simu” button and select the external text edit command. For example, select “NotePad.exe” for Windows application. And do not assign values to any parameter. The left procedures are as same as “Run Simu command” that has been described before.

Platform Independent Issues: as we have addressed at the beginning. The “Run external Simu command” function does not work in Unix environment. This problem is due to the bug in the Java language. In Java, the external command can only be launched by calling Runtime.exec(). However, to execute a DOS style external command in Windows, we need a “xxx.bat” file with format we have showed in previous sections. So, the program need a minor when run external simu command in Unix environment.

7. Graphical output of Cell-DEVS models

- Select "Animate" from the main menu of the main GUI, select the "CellAnimate" item.

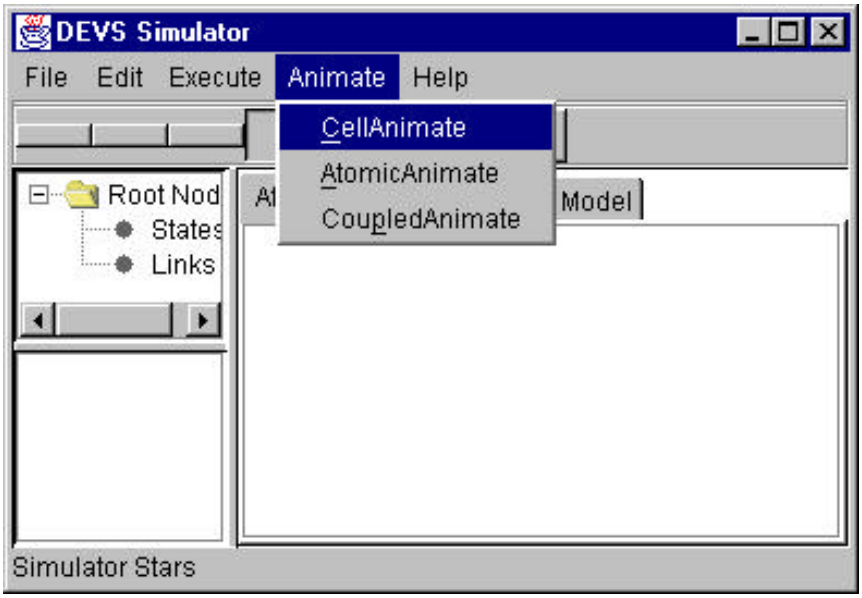


Fig cell-1

- A *cell animate* Dialog box will open.

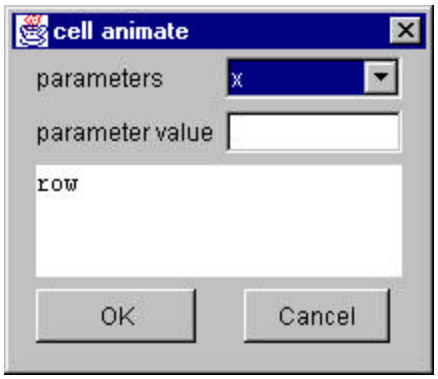


Fig cell-2

- Select each *parameters* and input their *parameter value* respectively. Some help information will be shown in the text area correspondingly. Here are some examples:

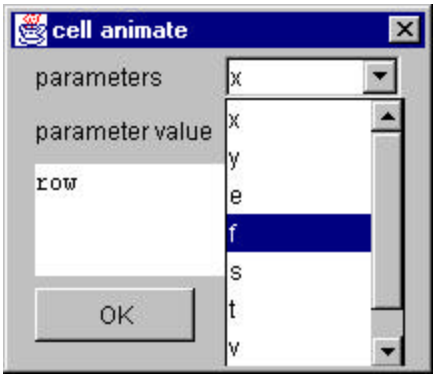


Fig cell-3: Select the "f" parameter

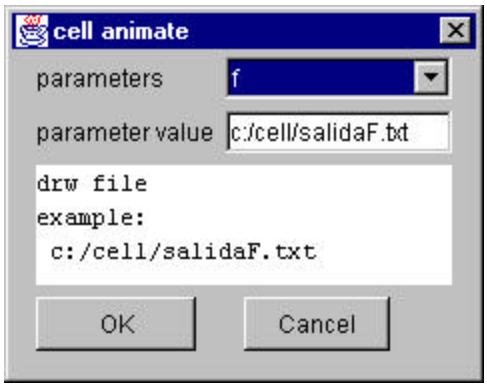


Fig cell-4: Input the "f" parameter value by inputting the drw file name and path. For example: c:/salidaF.txt

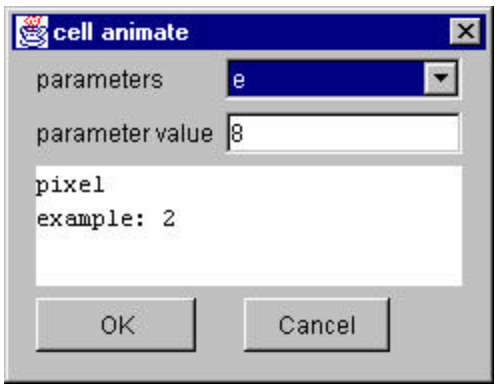


Fig cell-5: Input the "e" parameter value by inputting the pixel size you want. For example: 8

The possible parameters are:

x: the height of the grid. Int number, for example 20

y: the width of the grid. Int number, for example 20

e: the scale. if e is 1, means 1∗1 pixel; if e is 8, means 8∗8 pixel

f: the drw file which is produced by *drawlog*

s: ignore the middle times of the result. For example, if s is 10, means, after first show, then next show will be the time after 10∗**t,** and the third show will be after another 10∗**t,** and so on.

t: ignore the first # steps, then step by step. If t is 10, means ignore the first 10 steps, the step by step.

v: show with grid or not. Int number. 1: show grid. 0: not show grid.

m: Maximum step to show. Int number, for example 30.

- Click Ok button on the cell dialog box after inputting the parameters. Then the graphical output for the Cell will be shown.

Here in the following are some steps being showed:

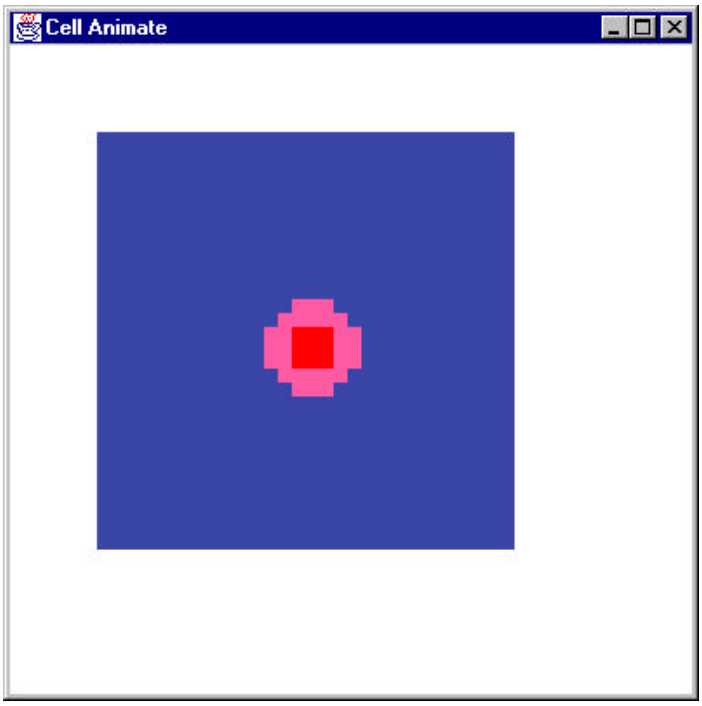


Fig cell-6

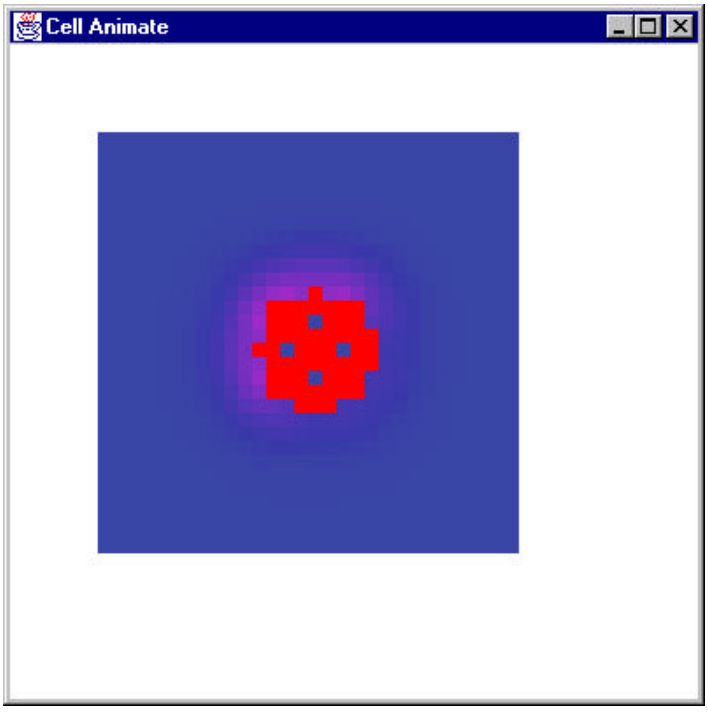


Fig cell-7

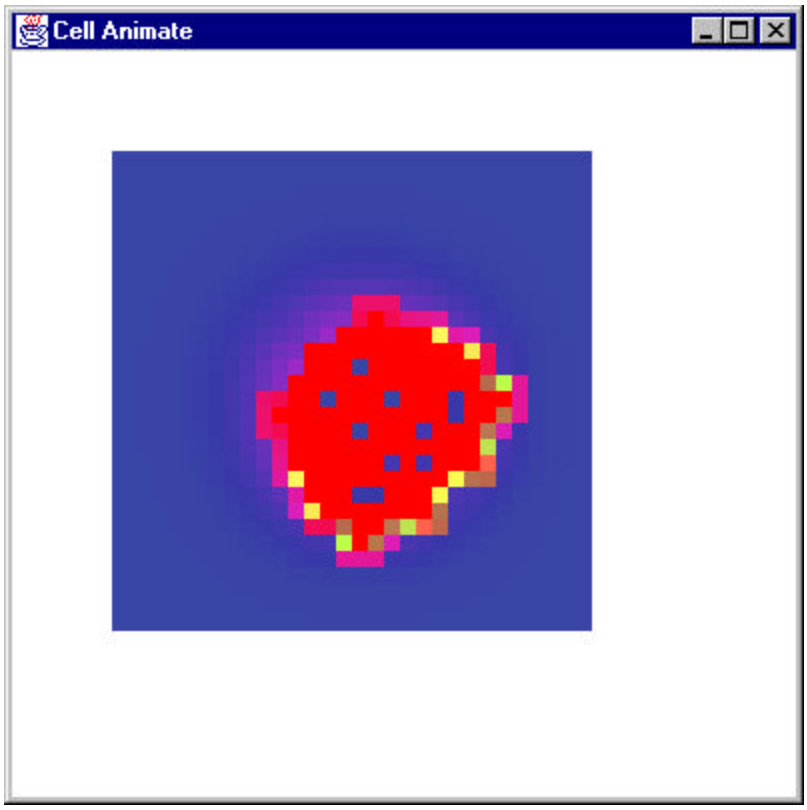


Fig cell-8

8. Graphical output of Atomic-DEVS models

- Select "Animate" from the main menu of the main GUI, select the "AtomicAnimate" item
- A *atomic animate* Dialog box will open.
- Select the *parameter* "f" and input the *parameter value*. Some help information will be shown in the text area. Here are some examples:

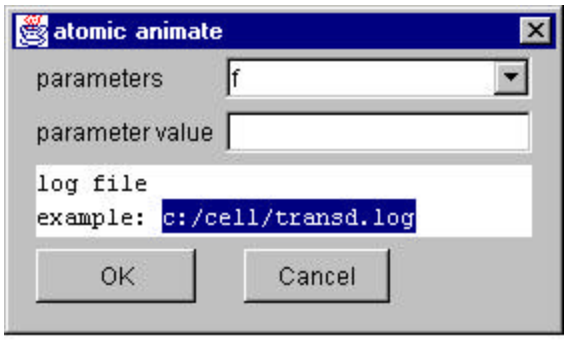


Fig atomic-1: input the log file name in the *parameter value* field. For example, c:/transd.log You could also copy the example file name in the text area by first selecting the contents, then press Ctrl-c, and put the cursor at the beginning of the *parameter value* area, finally press Ctrl-v, thus the contents are pasted on it.

- Click Ok button on the atomic dialog box after inputting the parameter. Then the graphical output for the Atomic will be shown.

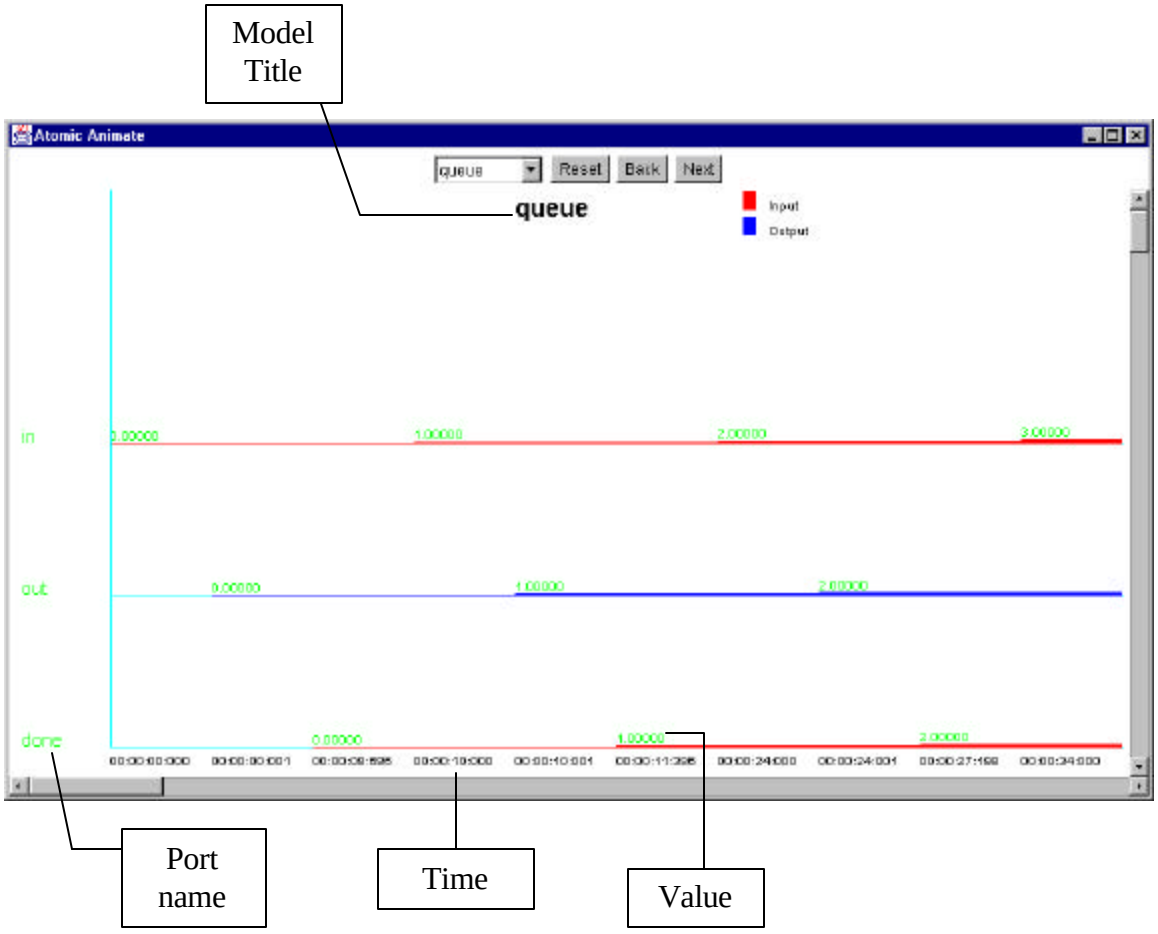


Fig atomic-2

- With "Next" button and "Back" button, you could see the graphical output of a specific model at different period of time until the end of the result.

  With "Reset" Button, the beginning of the output could be shown at any time. And you also can repeat it at any time.

- The popup list makes you chances of selecting or changing to any model you want to show at any time.

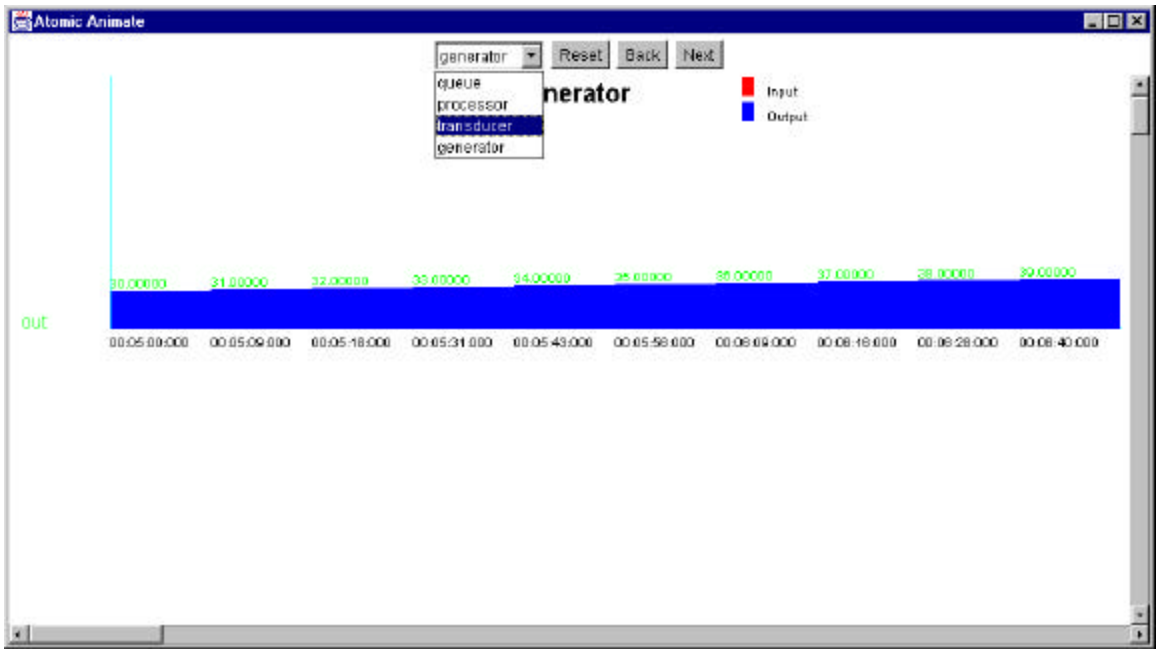


Fig atomic-3

The above is an example in *Fig atomic-3*, suppose a period of an output for the model "generator" is currently shown, but you want to change to see "transducer" model instead, then just select the transducer from the popup list , then you could see the output of transducer. And you could using "Next" or "Back" button to shown output of different time. Here is an example in the following:

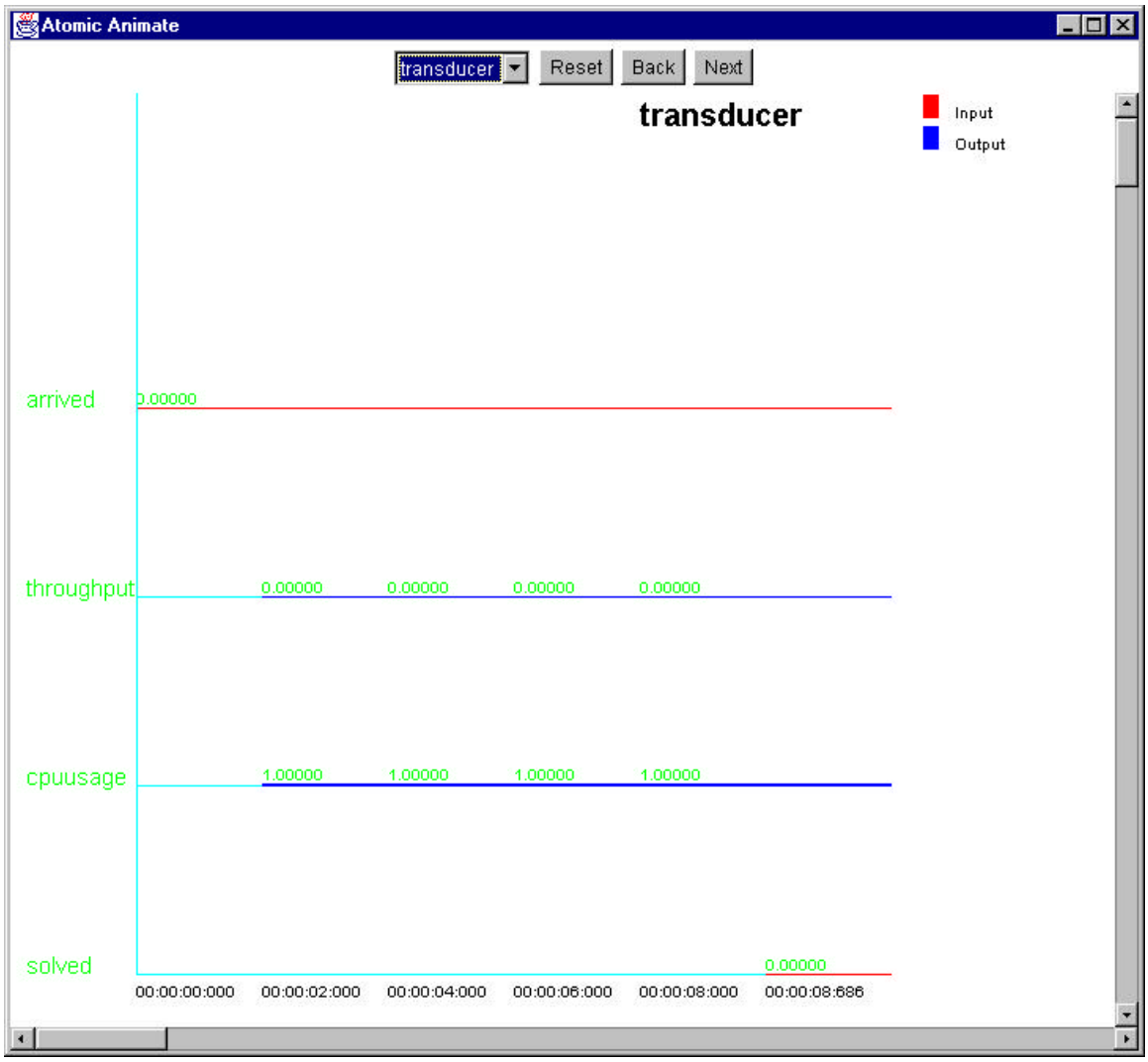


Fig atomic-4

9. Graphical output of Coupled-DEVS models

- Select "Animate" from the main menu of the main GUI, select the "CoupledAnimate" item.
- A *coupledc animate* Dialog box will open.
- Select the all the *parameters* and input their *parameter valuse respectively*. Some help information will be shown in the text area. Here are some examples:

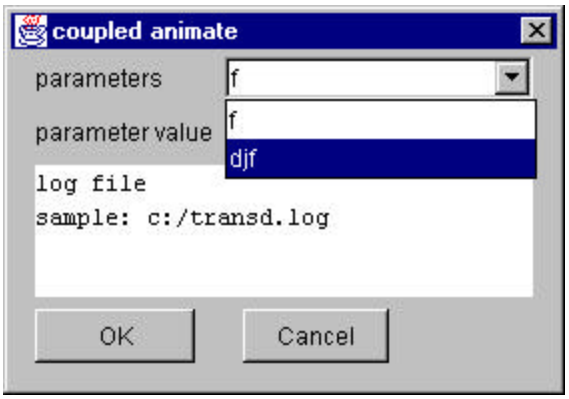
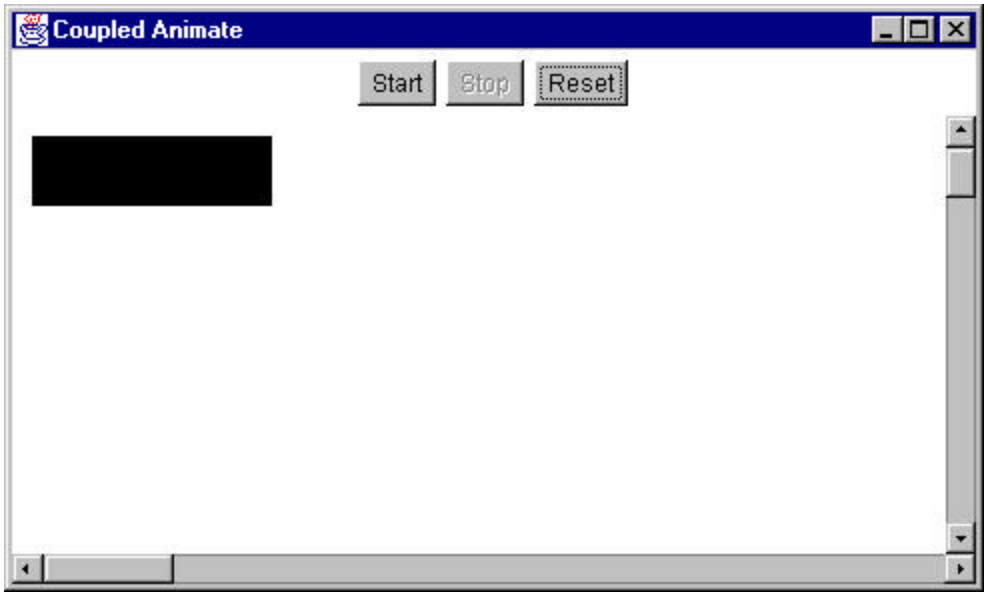


Fig coupled-1

Also you could use Ctrl-c to copy text and use Ctrl-v to paste it.

- Click Ok button on the coupled animate dialog box after inputting all the parameters. Then the graphical output for the Coupled will be shown.



- Press "Start" button, to begin:

Fig coupled-2

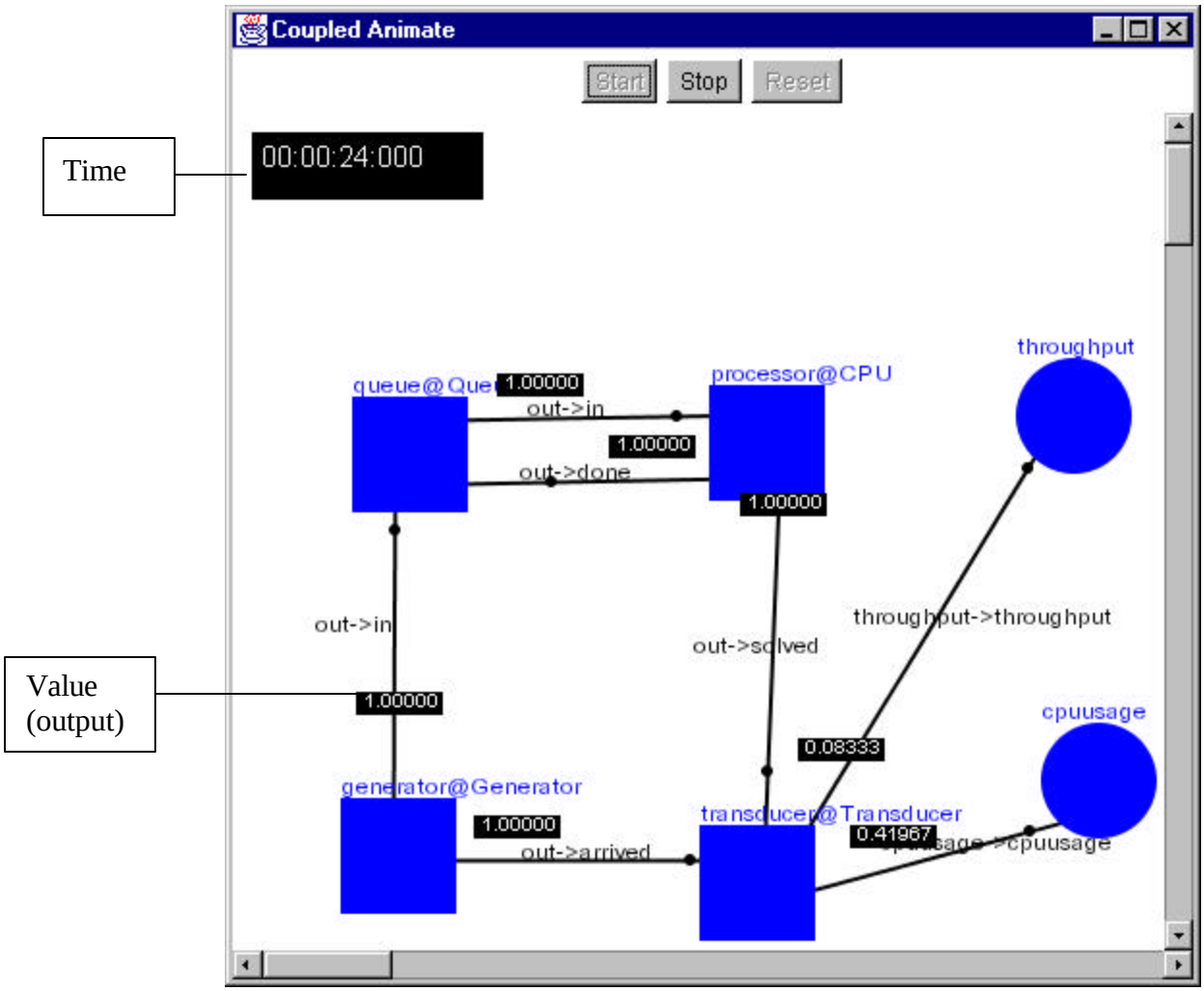


Fig coupled-3

At different same time, the change of output value for various models as well as all the links, models will be shown clearly.

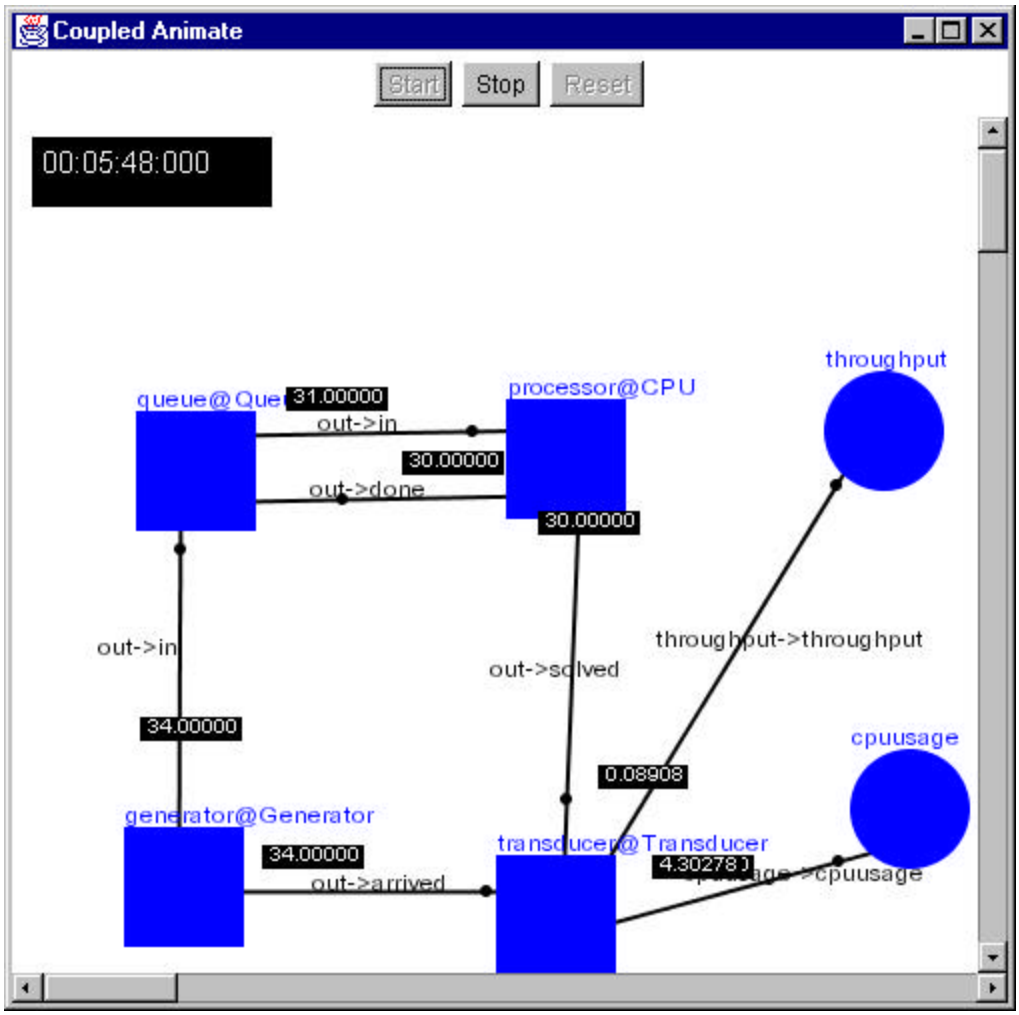


Fig coupled-4

- You could stop the simulation with "Stop" button at any time and continue by pressing "Start" button.

- If you want, click "Reset" button, the simulation will stop and the system will be reset. Then click the "Start" button to simulate from the very beginning.

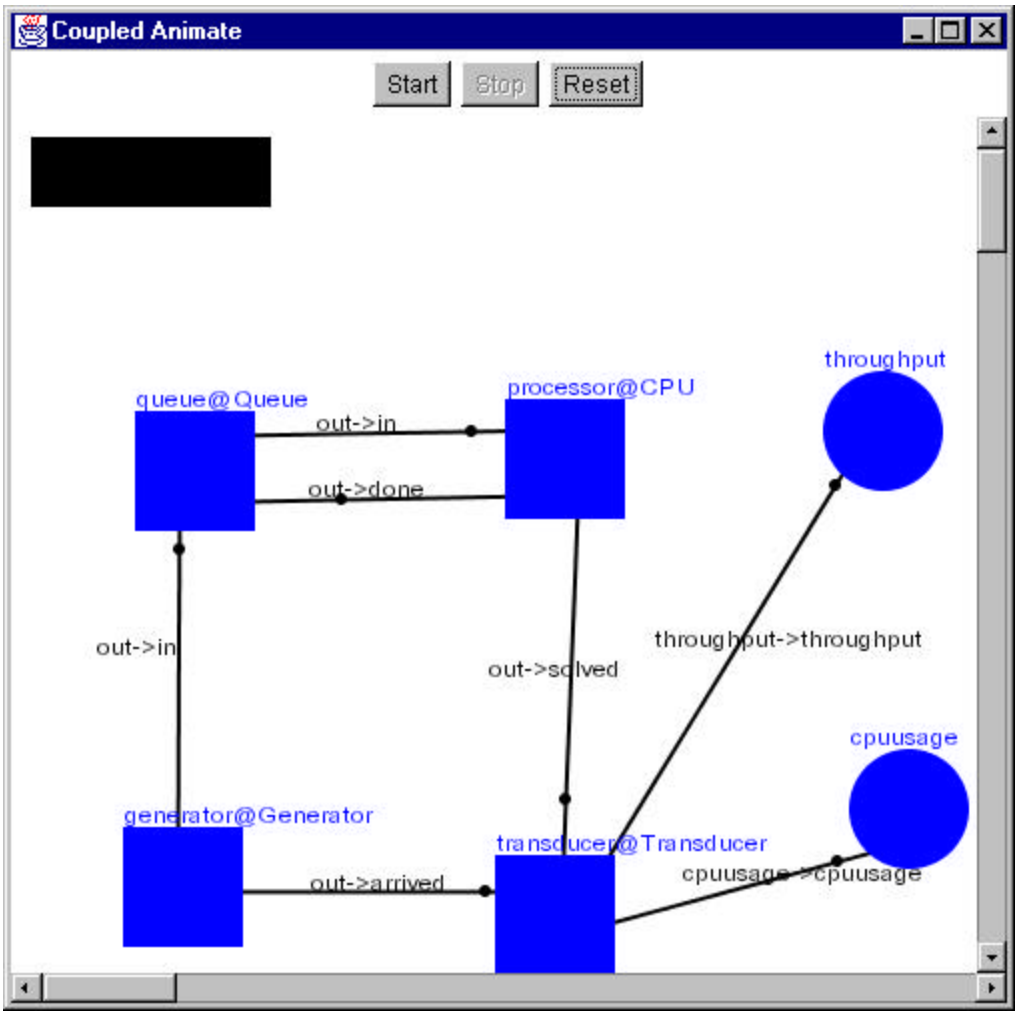


Fig coupled-5

10.Warning Message

Since in this project, for each model, there may need some input in dialog box. If the input is wrong, then some graphical output will not run. To avoid this, this project designs some warning message.

- For each dialog box of each model, if user forget to input some necessary parameter value and press a "Ok" button, warning message like the following will show:

Fig warning-1

If this happens, just close the warning window and back to the dialog window, check and input the parameters values which you have forgotten, and try again.

Note if this warning appears, you needn't input all the parameters again, the system is able to keep the already added parameters values (for instance, if you added the file name - option f -, and forgotten to add the djf file, then you should only input the parameters value of the djf when you try again).

- If inputting filename is needed in the dialog box, and you input the wrong file name or path, a warning will appear:

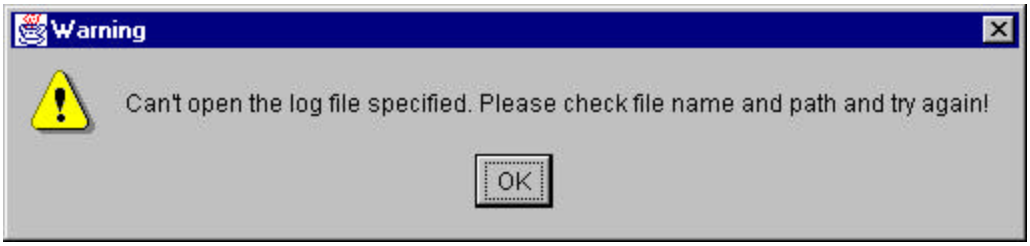


Fig warning-2

Also you need only to reinput the correct file name or path, and don't worry about other values you have already input. In the dialog box of each model, at any time, you could copy/paste the text by selecting the texts and using Ctrl-c to copy and ctrl-v to paste. The size of each dialog box could be enlarge or change as you like by putting the mouse on the border and drag.