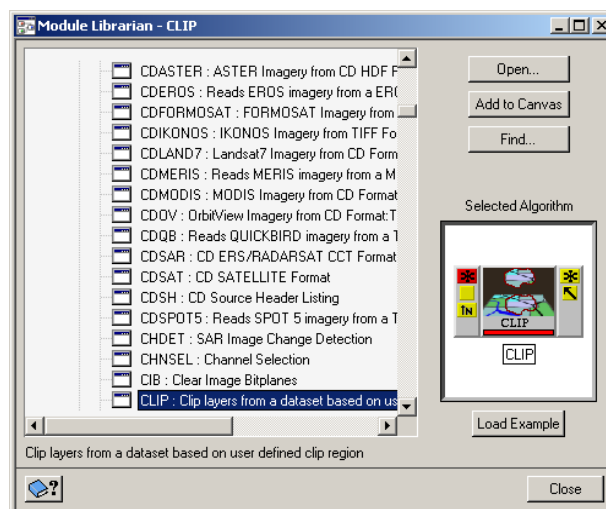


## TUTORIAL

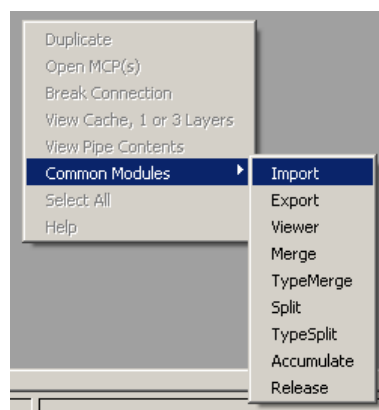
The new integration capabilities between Focus and the Modeler environment in Geomatica 10 now allows users to create custom models and utilize our batch processing abilities. This example will show you how you can create a model to clip all your input data layers to the same extents (such as an area of interest). All input layers will be used to create new layers based on the extents of the file that you wish to use as a Clip File. This and other models can be downloaded from our technical support modeler exchange website.

### Setup the model

1. From the Module Librarian window, find the clip module, then click on the Add to Canvas button, and click on the location of the Modeler workspace that you wish to position the module.



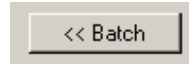
2. Right click in the Modeler workspace and choose Import from the Common Modules list. Now click again on the workspace to place the module into the model.



3. Right click on the Import Module that you just placed, and choose Duplicate to place a second Import Module into the model to use to open the clip file.

4. Finally add an Export module to the workspace from the Common Modules list. If necessary, rearrange your modules in the workspace until you are satisfied with the general location of each one.

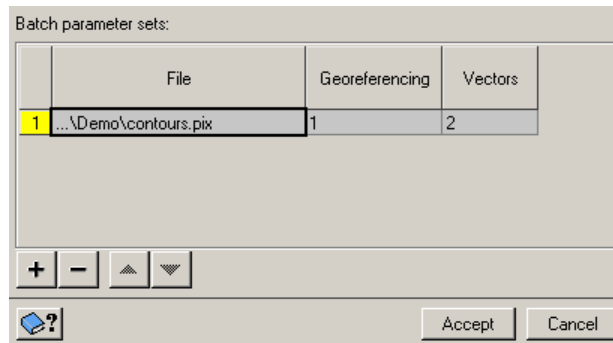
5. Select the first Input Module, right click and choose Open MCP(s) to open the module so that you can assign file names and parameters. Click the Batch button on the bottom to expand the window into Batch Mode.



6. Under the Input Params 1 tab, use the browse button to locate the first input File and then click Open.

7. In the Available data types section, select all of the available data types of the file that you wish to include.

8. Click the + button on the bottom to add another input file. Click on the layer that was just added to the window. Now go back to the top of the window and use the browse button to locate the second image. Repeat until all input images have been selected.



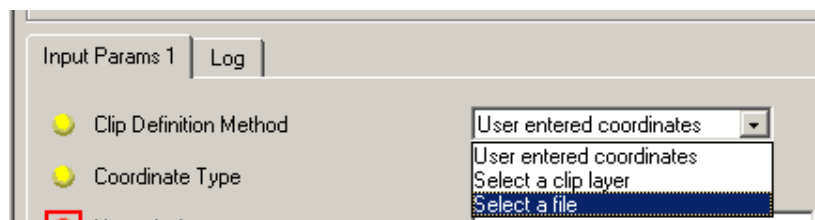
9. When all input layers have been set up, then you can click the Accept button to continue.

10. Select the second Input Module, right click and choose Open MCP(s) to open the module so that you can assign file names and parameters.

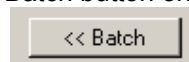
11. Under the Input Params 1 tab, browse to locate the File to be used as the clip file and then click Open, click the available data types and then click the Accept button to continue.

12. Select the Clip Module, right click and choose Open MCP(s) to open the module so that you can assign file names and parameters.

13. Under the Input Params 1 tab choose Select a File as your Clip Definition Method and clip Accept button.



14. Select the Export Module, right click and choose Open MCP(s) to open the module so that you can assign file names and parameters. Click the Batch button on the bottom to expand the window.



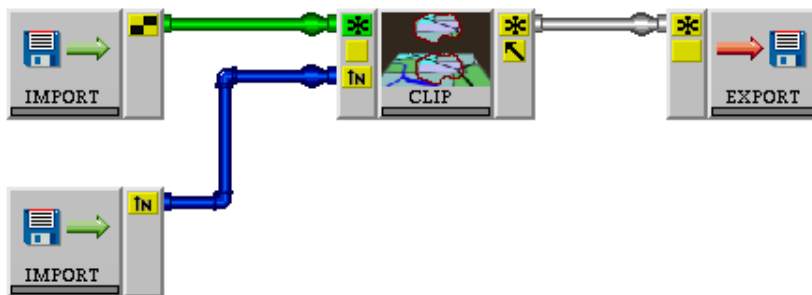
15. Under the Input Params 1 tab, browse to location that you wish to create the File to and give the output file a new filename. Next press the + button to add another output file and new file name for each one of the input files that you wish to batch process.

16. Click Accept to continue.

17. Next join the modules together with pipes. On the right side of the IMPORT module, click the mixed port glyph. Then click on the input port glyph that will be located on the left side of the Clip module and will be colored red until it is properly connected.

18. On the right side of the second IMPORT module, click the mixed port glyph and then click the input Clip layer port glyph that will be located on the left side of the Clip module just below the one that you just connected.

19. On the right side of the Clip module, click the mixed port glyph and then click on the input port glyph that will be located on the left side of the Export module. Your finished model should look like the one below. (You can save your model to a .MOD file from the main menu).



## Run the model in batch mode

1. Once all the modules have been properly setup (with input and output file names specified) and all modules have been connected with pipes then you can run the model in batch mode. To run the model in batch mode, all you need to do is use the Run Batch button.

