OMFIT M3D-C1 Tutorial

B.C. Lyons
2/7/2017

This is a help document written in tutorial fashion that guides the user through the automated running of M3D-C1 using the autoC1 script through the OMFIT M3DC1 module.

Introduction

M3D-C1 is an implicit, three-dimensional, nonlinear simulation code that solves the two-fluid magnetohydrodynamics (MHD) equations in highly magnetized toroidal geometries. It also contains options to perform two-dimensional nonlinear or linear simulations, and to solve reduced sets of the MHD equations. The official users guide can be downloaded from http://w3.pppl.gov/~jardin/M3DC1/NEWDOC-latest.pdf.

This document outlines how to run M3D-C1 via OMFIT. It demonstrates how:

- to load and to preprocess a new equilibrium reconstruction for analysis,
- to load and to modify an M3D-C1 input namelist,
- to select types of calculations one wants to perform,
- and to start new calculations in an existing OMFIT M3D-C1 directory.

Running M3D-C1 through OMFIT is built on top of the autoC1 python utility, available for download from GitHub at https://github.com/bclyons12/autoC1.

Getting started

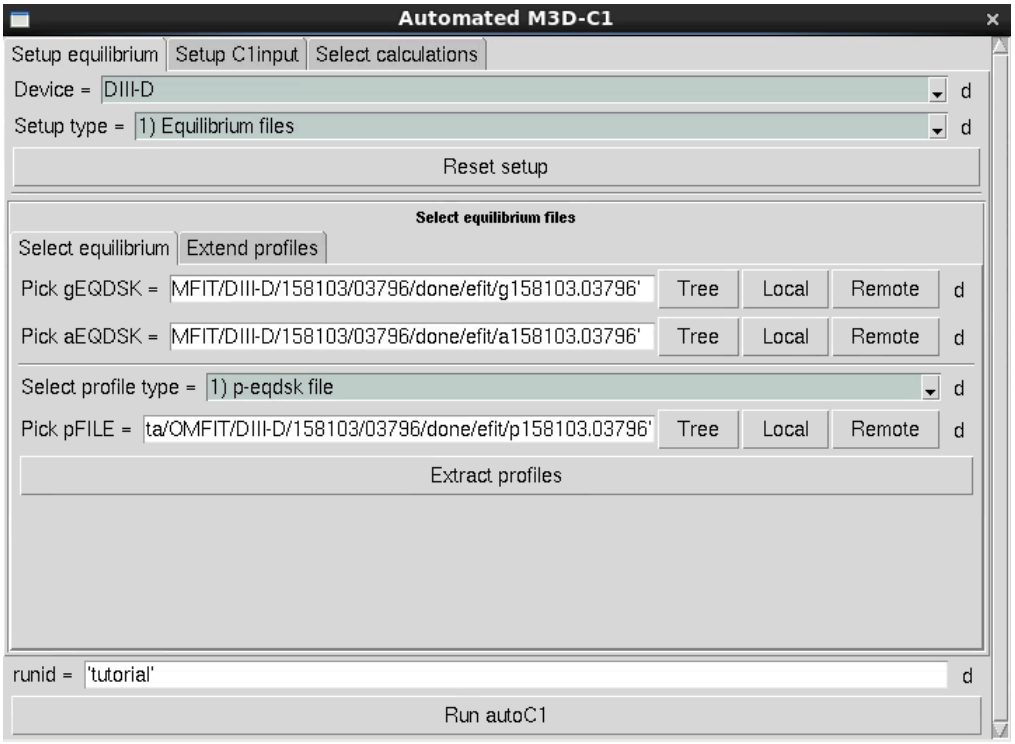
In order to perform automated M3D-C1 calculations, first launch OMFIT on your desired platform. From the menus, select `File` ---> `Import Module`, then load the `M3DC1` module from the list of Module IDs. From the OMFIT tree, double click `M3DC1` to launch the Automated M3D-C1 GUI. Note that the module name lack a hyphen compared to the name of the code. Detailed instructions for running OMFIT on various platforms can be found here: http://gafusion.github.io/OMFIT-source/usage.html.

Setting up an equilibrium

Under the `Setup equilibrium` tab, choose what `Device` you want to simulate from the dropdown menu. This will allow OMFIT to know what files it needs and autoC1 to know what templates to use. Next, choose what `Setup type` you want, running from `Equilibrium files` or running from an `Existing autoC1 directory`. If necessary, click the `Reset setup` button to start the setup over. This can be done at any step.

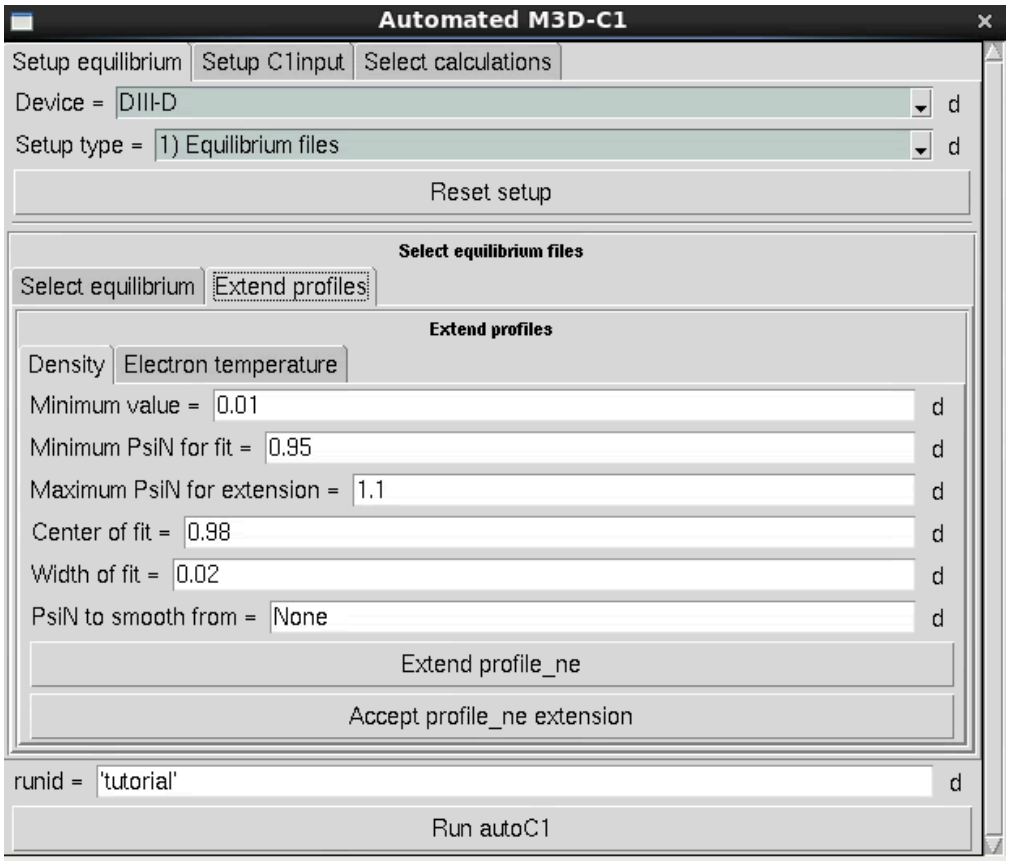
Running from equilibrium files

Selecting equilibrium files

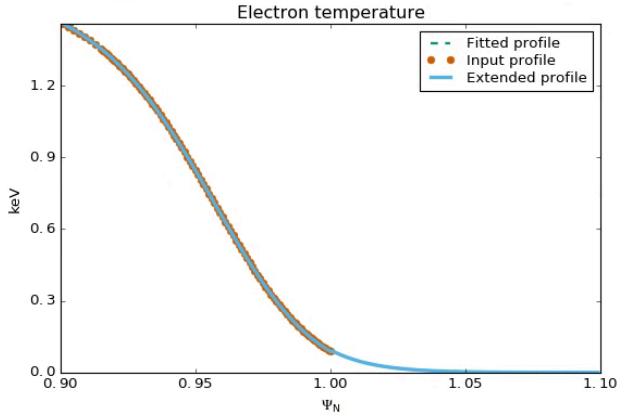


If you choose `Setup type = 1) Equilibrium files`, you must select the appropriate equilibrium files under the `Select equilibrium` subtab. All files can be loaded from either the OMFIT Tree itself or a local or remote path. A `gEQDSK` file is always included in the required equilibrium inputs and is assumed to be named like g<shot>.<time>. The numbers <shot> and <time> are used to define the working directory for this M3D-C1 calculation. Next, select what type of kinetic profiles format to load from the `Select profile type` menu and load the appropriate file(s). Unless `Select profile type = 0) M3D-C1 input files`, you must click the `Extract profiles` button in order to generate the profile_ne, profile_te, and profile_omega files that M3D-C1 itself will read.

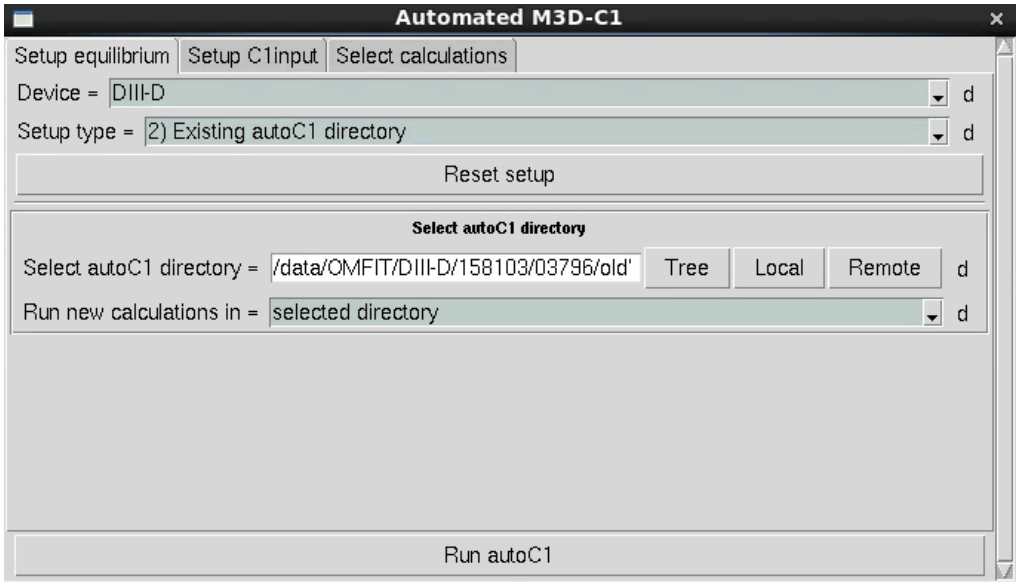
Extending profile data to open-field-line region



Next, the density and electron temperature profiles should be extended to small values in the open-field-line region using the `Extend profiles` subtab and the associated subsubtab. You can define initial parameters for the linear-tanh fit that will be used to extend that profile to the selected `Minimum value` at the selected `Maximum PsiN for extension`. Then, click the `Extend profile_<name>` button and examine the extended profile in the plot that appears. The final fit parameters are printed to the OMFIT Console, along with a warning if the fit has gone negative at any point. Keep modifying the initial fit parameters until the desired extension is achieved, then click the `Accept profile_<name> extension` button.

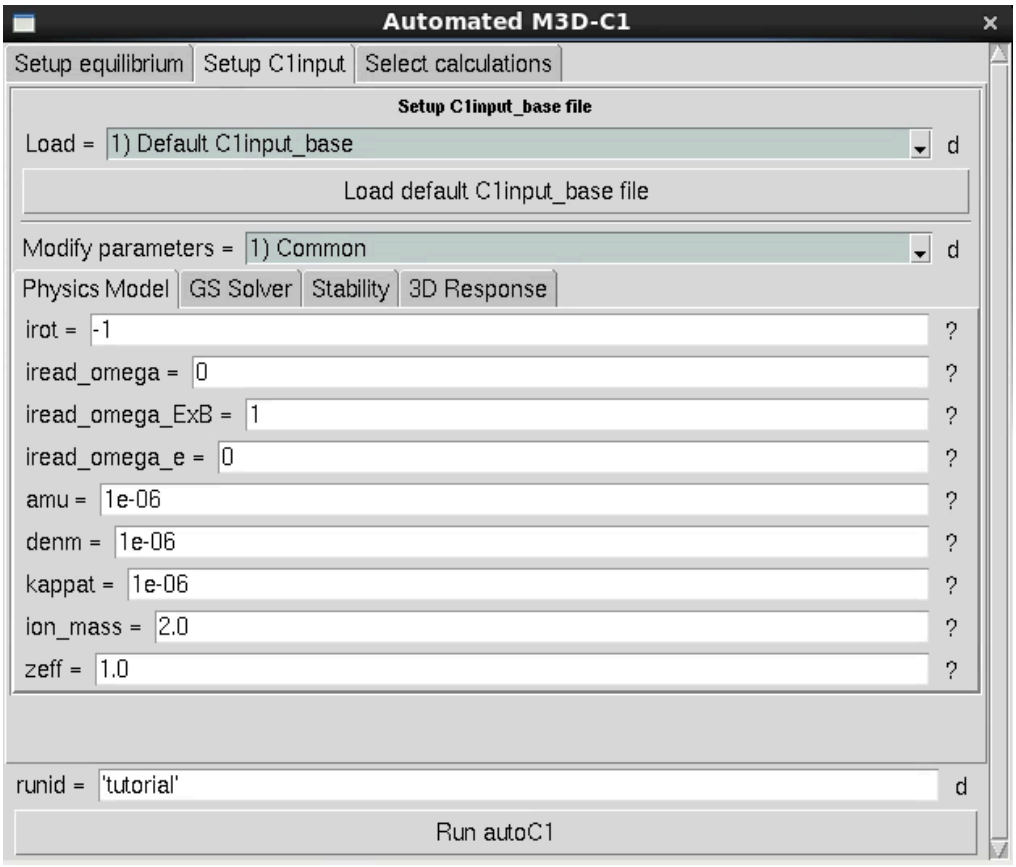


Running from an existing autoC1 directory



If you choose `Setup type = 2) Existing autoC1 directory`, you must choose a directory created by a previous run of autoC1 through OMFIT in the `Select autoC1 directory` entry box. Once the directory is given, OMFIT will load appropriate files from it so that it knows how to launch new calculations efficiently. In addition, you can choose to `Run new calculations in` either the `selected directory`, or in a `new directory`. Either way, choosing these options will tell autoC1 that the appropriate setup (e.g., mesh adaptation) has already been completed and you will be able to more quickly run new calculations.

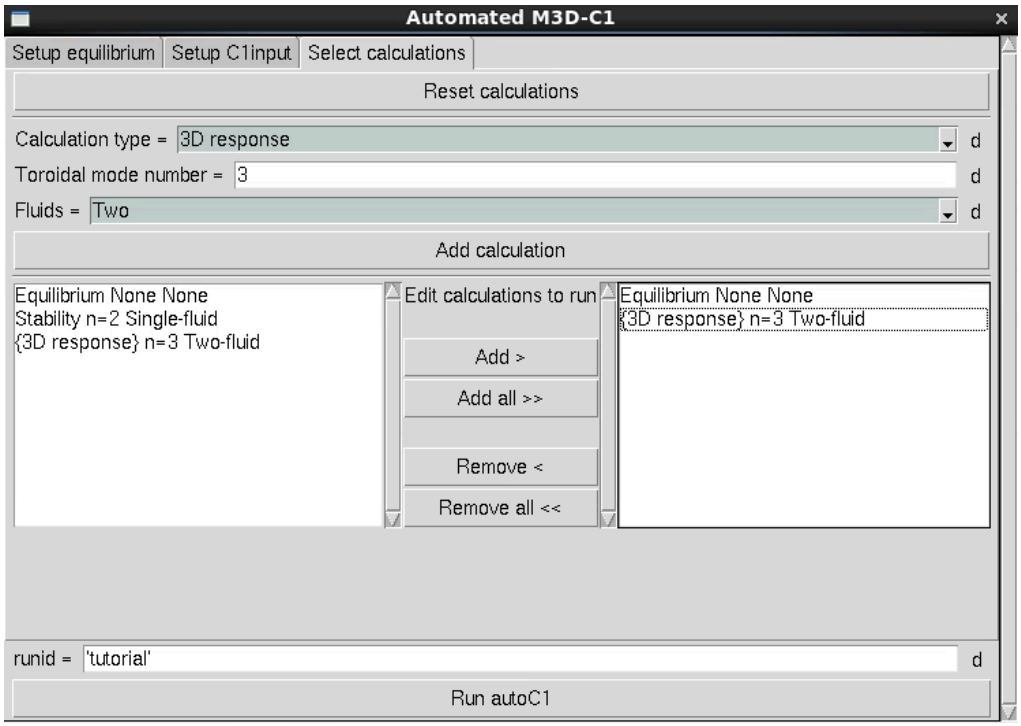
Setting up the input namelist



The next step is to load a C1input_base namelist. While autoC1 will take care of the details about certain parameters that need to be set for different kinds of runs, the input parameters in this namelist will be used for all the runs. One can choose to load a Default C1input_base file from the autoC1 templates (recommended for new users) or an existing C1input_base file from a past automated M3D-C1 run. If you choose `Load = 2) Exising C1input_base`, another box will appear allowing you to pick the desired file. If starting new calculations from an existing autoC1 directory, you'll also have a `Load = 3) Last C1input_base used` option.

After choosing the desired file and clicking the `Load <type> C1input_base` button, you will be able to modify the parameters loaded from that file. Choosing `Modify parameters = 1) Common` will give various commonly modified parameters, divided into subtabs depending on what those parameters are used for. Options are also given to modify any parameter already present in the C1input_base file and to add/modify/remove arbitrary parameters using a key-value pair.

Selecting calculations to run



The final step of the setup is to choose what types of calculations you want to run with the chosen equilibrium and C1input_base parameters. This is done under the `Select calculations` tab. Under the `Calculation type` menu, you can choose from

- `Equilibrium`: recalculate the equilibrium on the adapted M3D-C1 mesh,
- `Stability`: calculate the linear stability of the chosen `Toroidal mode number` for single- or two-fluid MHD,
- or `3D response`: calculate the linear response to external, 3D magnetic perturbations for the chose `Toroidal mode number` for single- or two-fluid MHD.

After specifying the desired options for a run, click the `Add calculation` button. It will then appear in both columns of the list editor. The left side gives the list of all calculation types that you have added. The right side gives the list of calculation you actually intend to run. The buttons in the middle can be used to add or to remove calculations from the run list.

Starting the autoC1 run

Once the setup is complete, define an appropriate `runid` in the box at the bottom of the GUI, if that box is present. Along with the `Device` and the <shot> and <time> from the `gEQDSK` file selected under the `Setup equilibrium` tab, this `runid` will define the working directory for the run. The definition of this working directory can be found in the OMFIT tree in:

```
OMFIT['M3DC1']['SETTINGS']['REMOTE_SETUP']['workDir']
```

Note that if you have the option to define a `runid`, this working directory will be overwritten if it already exists.

Once you're satisfied with all the options, click the `Run autoC1` button to begin the calculations on the remote server.

Exploring an autoC1 directory

Under construction

Examining M3D-C1 results

At the moment, all examination of M3D-C1 results should be done through the existing IDL post-processing routines. Instructions for using those scripts can be found here.

Python scripts for M3D-C1 post-processing are currently under development. This guide will be updated when they are usable through OMFIT.