OOMMF
User's Guide

October 30, 2002

This manual documents release 1.2a3.

WARNING: In this alpha release, the documentation may not be up to date.

Abstract

This manual describes OOMMF (Object Oriented Micromagnetic Framework), a public domain micromagnetics program developed at the National Institute of Standards and Technology. The program is designed to be portable, flexible, and extensible, with a user-friendly graphical interface. The code is written in C++ and Tcl/Tk. Target systems include a wide range of Unix platforms, Windows NT, and Windows 9X.

Contents

Disclaimer

This software was developed at the National Institute of Standards and Technology by employees of the Federal Government in the course of their official duties. Pursuant to Title 17, United States Code, Section 105, this software is not subject to copyright protection and is in the public domain.

OOMMF is an experimental system. NIST assumes no responsibility whatsoever for its use by other parties, and makes no guarantees, expressed or implied, about its quality, reliability, or any other characteristic.

We would appreciate acknowledgement if the software is used. When referencing OOMMF software, we recommend citing the NIST technical report, M. J. Donahue and D. G. Porter, "OOMMF User's Guide, Version 1.0," **NISTIR 6376**, National Institute of Standards and Technology, Gaithersburg, MD (Sept 1999).

Commercial equipment and software referred to on these pages are identified for informational purposes only, and does not imply recommendation of or endorsement by the National Institute of Standards and Technology, nor does it imply that the products so identified are necessarily the best available for the purpose.

1 Overview of OOMMF

The goal of the OOMMF[1] (Object Oriented Micromagnetic Framework) project in the Information Technology Laboratory (ITL) at the National Institute of Standards and Technology (NIST) is to develop a portable, extensible public domain micromagnetic program and associated tools. This code will form a completely functional micromagnetics package, but will also have a well documented, flexible programmer's interface so that people developing new code can swap their own code in and out as desired. The main contributors to OOMMF are Mike Donahue and Don Porter.

In order to allow a programmer not familiar with the code as a whole to add modifications and new functionality, we feel that an object oriented approach is critical, and have settled on C++ as a good compromise with respect to availability, functionality, and portability. In order to allow the code to run on a wide variety of systems, we are writing the interface and glue code in Tcl/Tk. This enables our code to operate across a wide range of Unix platforms, Windows NT, and Windows 9X.

The code may be modified at 3 distinct levels. At the top level, individual programs interact via well-defined protocols across network sockets. One may connect these modules together in various ways from the user interface, and new modules speaking the same protocol can be transparently added. The second level of modification is at the Tcl/Tk script level. Some modules allow Tcl/Tk scripts to be imported and executed at run time, and the top level scripts are relatively easy to modify or replace. At the lowest level, the C++ source is provided and can be modified, although at present the documentation for this is incomplete (cf. the "OOMMF Programming Manual").

The first portion of OOMMF released was a magnetization file display program called **mmDisp**. A working release[2] of the complete OOMMF project was first released in October, 1998. It included a problem editor, a 2D micromagnetic solver, and several display widgets, including an updated version of **mmDisp**. The solver can be controlled by an interactive interface (Sec. 10.1), or through a sophisticated batch control system (Sec. 10.2). This solver was originally based on a micromagnetic code that Mike Donahue and Bob McMichael had previously developed. It utilizes a Landau-Lifshitz ODE solver to relax 3D spins on a 2D mesh of square cells, using FFT's to compute the self-magnetostatic (demag) field. Anisotropy, applied field, and initial magnetization can be varied pointwise, and arbitrarily shaped elements can be modeled.

The current development version, OOMMF 1.2, includes Oxs, the OOMMF eXtensible Solver. Oxs offers users of OOMMF the ability to extend Oxs with their own modules. The details of programming an Oxs extension module are found in the OOMMF Programming Manual [3]. The extensible nature of the Oxs solver means that its capabilities may be varied as necessary for the problem to be solved. Oxs modules distributed as part of OOMMF support full 3D simulations suitable for modeling layered materials.

[1] http://math.nist.gov/oommf/
[2] http://math.nist.gov/oommf/software.html
[3] http://math.nist.gov/oommf/doc/

If you want to receive e-mail notification of updates to this project, register your e-mail address with the "μMAG Announcement" mailing list:

http://www.ctcms.nist.gov/~rdm/email-list.html.

The OOMMF developers are always interested in your comments about OOMMF. See the Credits (Sec. 22) for instructions on how to contact them, and for information on referencing OOMMF.

2 Installation

2.1 Requirements

OOMMF software is written in C++ and Tcl. It uses the Tcl-based Tk Windowing Toolkit to create graphical user interfaces that are portable to many varieties of Unix as well as Microsoft Windows 9X/NT.

Tcl and Tk must be installed before installing OOMMF. Tcl and Tk are available for free download [4] from the Tcl Developer Xchange[5]. We recommend the latest stable versions of Tcl and Tk concurrent with this release of OOMMF. OOMMF requires at least Tcl version 7.5 and Tk version 4.1 on Unix platforms, and requires at least Tcl version 8.0 and Tk version 8.0 on Microsoft Windows platforms. OOMMF software does not support any alpha or beta versions of Tcl/Tk, and each release of OOMMF may not work with later releases of Tcl/Tk. Check the release dates of both OOMMF and Tcl/Tk to ensure compatibility.

A Tcl/Tk installation includes two shell programs. The names of these programs may vary depending on the Tcl/Tk version and the type of platform. The first shell program contains an interpreter for the base Tcl language. In the OOMMF documentation we refer to this program as `tclsh`. The second shell program contains an interpreter for the base Tcl language extended by the Tcl commands supplied by the Tk toolkit. In the OOMMF documentation we refer to this program as `wish`. Consult your Tcl/Tk documentation to determine the actual names of these programs on your platform (for example, `tclsh83.exe` or `wish8.0`).

OOMMF applications communicate via TCP/IP network sockets. This means that OOMMF requires support for networking, even on a stand-alone machine. At a minimum, OOMMF must be able to access the loopback interface so that the host can talk to itself using TCP/IP.

OOMMF applications that use Tk require a windowing system and a valid display. On Unix systems, this means that an X server must be running. If you need to run OOMMF applications on a Unix system without display hardware or software, you may need to start the application with command line option **`-tk 0`** (see Sec. 5) or use the Xvfb[6] virtual frame buffer.

The OOMMF source distribution unpacks into a directory tree containing about 800 files and directories, occupying approximately 10 MB of storage. The amount of disk space needed for compiling and linking varies greatly between platforms; allow an additional 15 MB to 80 MB for the build. Removing intermediate object modules (cf. the `pimake` "objclean" target, in Reducing Disk Space Usage, Sec. 2.3.1, below) reduces the final space requirement for source + binary executables to between 15 MB and 25 MB. The OOMMF distribution containing Windows executables unpacks into a directory tree occupying about 15 MB of storage. **Note:** On a non-compressed `FAT16` file system on a large disk, OOMMF may

[4] http://purl.org/tcl/home/software/tcltk/choose.html
[5] http://purl.org/tcl/home/
[6] http://www.itworld.com/AppDev/1461/UIR000330xvfb/

take up much more disk space. This is because on such systems, the minimum size of any file is large, as much as 32 KB. Since this is much larger than many files in the OOMMF distribution require, a great deal of disk space is wasted.

To build OOMMF software from source code, you will need a C++ compiler capable of handling C++ templates, C++ exceptions, and (for the OOMMF eXtensible Solver) the C++ Standard Template Library. You will need other software development utilities for your platform as well. We do development and test builds on the following platforms, although porting to others should not be difficult:

Platform	Compilers
AIX	VisualAge C++ (xlC), Gnu gcc
Alpha/Linux	Compaq C++, Gnu gcc
Alpha/Windows NT	Microsoft Visual C++
Intel/Linux	Gnu gcc
Intel/Windows NT, 95, 98	Microsoft Visual C++, Intel C++, Cygwin gcc, Borland C++
MIPS/IRIX 6 (SGI)	MIPSpro C++, Gnu gcc
SPARC/Solaris	Sun Workshop C++, Gnu gcc

System Notes:

- **Windows:** Versions of the Microsoft Visual C++ compiler earlier than 5.0 will not build the Oxs (3D) solver.
- **HP-UX:** The older HP cfront compiler will not build the Oxs (3D) solver.

2.2 Basic Installation

Follow the instructions in the following sections, in order, to prepare OOMMF software for use on your computer.

2.2.1 Download

The latest release of the OOMMF software may be retrieved from the OOMMF download page[7]. Each release is available in two formats. The first format is a gzipped tar file containing an archive of all the OOMMF source code. The second format is a `.zip` compressed archive containing source code and pre-compiled executables for Microsoft Windows 9X/NT running on an x86-based microprocessor system. Each Windows binary distribution is compatible with only a particular sequence of releases of Tcl/Tk. For example, a Windows binary release for Tcl/Tk 8.3.x is compatible with Tcl/Tk 8.3.0, 8.3.1, ... Other release formats, e.g., pre-compiled executables for Microsoft Windows NT running on a Compaq Alpha Systems RISC-based microprocessor system, and/or compatible with older versions of Tcl/Tk, may be made available upon request.

[7] http://math.nist.gov/oommf/software.html

For the first format, unpack the distribution archive using gunzip and tar:

```
gunzip -c oommf12a3_20021029.tar.gz | tar xvf -
```

For the other format(s), you will need a utility program to unpack the `.zip` archive. This program must preserve the directory structure of the files in the archive, and it must be able to generate files with names not limited to the old MSDOS 8.3 format. Some very old versions of the pkzip utility do not have these properties. One utility program which is known to work is UnZip[8].

Using your utility, unpack the `.zip` archive, e.g.

```
unzip oommf12a3_20021029_84.zip
```

For either distribution format, the unpacking sequence creates a subdirectory `oommf` which contains all the files and directories of the OOMMF distribution. If a subdirectory named `oommf` already existed (say, from an earlier OOMMF release), then files in the new distribution overwrite those of the same name already on the disk. Some care may be needed in that circumstance to be sure that the resulting mix of files from an old and a new OOMMF distribution combine to create a working set of files.

2.2.2 Check Your Platform Configuration

After downloading and unpacking the OOMMF software distribution, all the OOMMF software is contained in a subdirectory named `oommf`. Start a command line interface (a shell on Unix, or the MS-DOS Prompt on Microsoft Windows), and change the working directory to the directory `oommf`. Find the Tcl shell program installed as part of your Tcl/Tk installation. In this manual we call the Tcl shell program `tclsh`, but the actual name of the executable depends on the release of Tcl/Tk and your platform type. Consult your Tcl/Tk documentation.

In the root directory of the OOMMF distribution is a file named `oommf.tcl`. It is the bootstrap application (Sec. 5) which is used to launch all OOMMF software. With the command line argument `+platform`, it will print a summary of your platform configuration when it is evaluated by `tclsh`. This summary describes your platform type, your C++ compiler, and your Tcl/Tk installation. As an example, here is the typical output on a Linux/Alpha system:

```
$ tclsh oommf.tcl +platform
oommf.tcl 1.2.0.3  info:
OOMMF release 1.2.0.3
Platform Name: linalp
Tcl name for OS: Linux 2.2.20
C++ compiler: /usr/bin/g++
Tcl configuration file: /usr/local/lib/tclConfig.sh
```

[8] http://www.info-zip.org/pub/infozip/UnZip.html

```
tclsh: /usr/local/bin/tclsh8.4
Tcl release: 8.4.1 (config) 8.4.1 (running)
Tk configuration file: /usr/local/lib/tkConfig.sh
wish: /usr/local/bin/wish8.4
Tk release: 8.4.1 (config) 8.4.1 (running)
```

If `oommf.tcl +platform` doesn't print a summary similar to the above, it should instead print an error message describing why it can't. For example, if your Tcl installation is older than release 7.5, the error message will report that fact. Follow the instructions provided and repeat until `oommf.tcl +platform` successfully prints a summary of the platform configuration information.

The first line of the example summary reports that OOMMF recognizes the platform by the name `linalp`. OOMMF software recognizes many of the more popular computing platforms, and assigns each a platform name. The platform name is used by OOMMF in index and configuration files and to name directories so that a single OOMMF installation can support multiple platform types. If `oommf.tcl +platform` reports the platform name to be "unknown", then you will need to add some configuration files to help OOMMF assign a name to your platform type, and associate with that name some of the key features of your computer. See the section on "Managing OOMMF platform names" (Sec. 2.3.4) for further instructions.

The second line reports what C++ compiler will be used to build OOMMF from its C++ source code. If you downloaded an OOMMF release with pre-compiled binaries for your platform, you may ignore this line. Otherwise, if this line reports "none selected", or if it reports a compiler other than the one you wish to use, then you will need to tell OOMMF what compiler to use. To do that, you must edit the appropriate configuration file for your platform. Continuing the example above, one would edit the file `config/cache/linalp.tcl`. Editing instructions are contained within the file. On other platforms the name `linalp` in `config/cache/linalp.tcl` should be replaced with the platform name OOMMF reports for your platform. For example, on a Windows machine using an x86 processor, the corresponding configuration file is `config/cache/wintel.tcl`.

The next three lines describe the Tcl configuration OOMMF finds on your platform. The first line reports the name of the configuration file installed as part of Tcl, if any. Conventional Tcl installations on Unix systems and within the Cygwin environment on Windows have such a file, usually named `tclConfig.sh`. The Tcl configuration file records details about how Tcl was built and where it was installed. On Windows platforms, this information is recorded in the Windows registry, so it is normal to have `oommf.tcl +platform` report "none found". If `oommf.tcl +platform` reports "none found", but you know that an appropriate Tcl configuration file is present on your system, you can tell OOMMF where to find the file by setting the environment variable OOMMF_TCL_CONFIG to its absolute filename. (For information about setting environment variables, see your operating system documentation.) In unusual circumstances, OOMMF may find a Tcl configuration file which doesn't correctly describe your Tcl installation. In that case, use the environment variable OOMMF_TCL_CONFIG to instruct OOMMF to use a different file that you specify, and

edit that file to include a correct description of your Tcl installation.

The second line describing your Tcl installation reports the absolute pathname of the `tclsh` program. If this differs from the `tclsh` you used to evaluate `oommf.tcl +platform`, there may be something wrong with your Tcl configuration file. Note that the same `tclsh` program might be known by several absolute pathnames if there are symbolic links in your Tcl installation. If `oommf.tcl +platform` reports that it cannot find a `tclsh` program, yet you know where an appropriate one is installed on your system, you can tell OOMMF where to find the `tclsh` program by setting the environment variable OOMMF_TCLSH to its absolute location.

The third line describing your Tcl installation reports its release number according to two sources. First is the release number recorded in the Tcl configuration file. Second is the release number of the `tclsh` program used to evaluate `oommf.tcl +platform`. If these numbers do not match, it may indicate something is wrong with your Tcl configuration file. If you have multiple releases of Tcl installed under a common root directory on your computer, there can be only one Tcl configuration file. It is important that you use the Tcl release that corresponds to the Tcl configuration file.

The next three lines describe the Tk configuration OOMMF finds on your platform. They are analogous to the three lines describing the Tcl configuration. The environment variables OOMMF_TK_CONFIG and OOMMF_WISH may be used to tell OOMMF where to find the Tk configuration file and the `wish` program, respectively.

Finally, the output of `oommf.tcl +platform` may include warnings about possible problems with your Tcl/Tk installation. For example, if you are missing important header files, or if your Tcl/Tk installation is thread-enabled (which OOMMF does not support).

If `oommf.tcl +platform` indicates problems with your Tcl/Tk installation, it may be easiest to re-install Tcl/Tk taking care to perform a conventional installation. OOMMF deals best with conventional Tcl/Tk installations. If you do not have the power to re-install an existing broken Tcl/Tk installation (perhaps you are not the sysadmin of your machine), you might still install your own copy of Tcl/Tk in your own user space. In that case, if your private Tcl/Tk installation makes use of shared libraries, take care that you do whatever is necessary on your platform to be sure that your private `tclsh` and `wish` find and use your private shared libraries instead of those from the system Tcl/Tk installation. This might involve setting an environment variable (such as LD_LIBRARY_PATH). If you use a private Tcl/Tk installation, you also want to be sure that there are no environment variables like TCL_LIBRARY or TK_LIBRARY that still refer to the system Tcl/Tk installation.

Other Configuration Issues A few other configurations should be checked on Windows platforms. First, note that absolute filenames on Windows makes use of the backslash (\) to separate directory names. On Unix and within Tcl the forward slash (/) is used to separate directory names in an absolute filename. In this manual we usually use the Tcl convention of forward slash as separator. In portions of the manual pertaining only to MS Windows we use the backslash as separator. There may be instructions in this manual which do not work exactly as written on Windows platforms. You may need to replace forward slashes

with backward slashes in pathnames when working on Windows.

OOMMF software needs networking support that recognizes the host name `localhost`. It may be necessary to edit a file which records that `localhost` is a synonym for the loopback interface (127.0.0.1). If a file named `hosts` exists in your system area (for example, `C:\Windows\hosts`), be sure it includes an entry mapping 127.0.0.1 to `localhost`. If no `hosts` file exists, but a `hosts.sam` file exists, make a copy of `hosts.sam` with the name `hosts`, and edit the copy to have the `localhost` entry.

In recent releases of Tcl/Tk (version 8.0.3 and later) the directory which holds the `tclsh` and `wish` programs also holds several `*.dll` files that OOMMF software needs to find to run properly. Normally when the OOMMF bootstrap application (Sec. 5) or **mmLaunch** (Sec. 6) is used to launch OOMMF programs, they take care of making sure the necessary `*.dll` files can be found. As an additional measure, you might want to add the directory which holds the `tclsh` and `wish` programs to the list of directories stored in the PATH environment variable. All the directories in the PATH are searched for `*.dll` files needed when starting an executable.

2.2.3 Compiling and Linking

If you downloaded a distribution with pre-compiled executables, you may skip this section.

When building OOMMF software from source code, be sure the C++ compiler reported by `oommf.tcl +platform` is properly configured. In particular, if you are running on a Windows system, please read carefully the notes in Advanced Installation, Sec. 2.3.5, pertaining to your compiler.

The compiling and linking of the C++ portions of OOMMF software are guided by the application pimake (Sec. 16.9) ("Platform Independent Make") which is distributed as part of the OOMMF release. To begin building OOMMF software with `pimake`, first change your working directory to the root directory of the OOMMF distribution:

```
cd .../path/to/oommf
```

If you unpacked the new OOMMF release into a directory `oommf` which contained an earlier OOMMF release, use `pimake` to build the target `upgrade` to clear away any source code files which were in a former distribution but are not part of the latest distribution:

```
tclsh oommf.tcl pimake upgrade
```

Next, build the target `distclean` to clear away any old executables and object files which are left behind from the compilation of the previous distribution:

```
tclsh oommf.tcl pimake distclean
```

Next, to build all the OOMMF software, run `pimake` without specifying a target:

```
tclsh oommf.tcl pimake
```

On some platforms, you cannot successfully compile OOMMF software if there are OOMMF programs running. Check that all OOMMF programs have terminated (including those in the background) before trying to compile and link OOMMF.

When `pimake` calls on a compiler or other software development utility, the command line is printed, so that you may monitor the build process. Assuming a proper configuration for your platform, `pimake` should be able to compile and link all the OOMMF software without error. If `pimake` reports errors, please first consult Troubleshooting (Sec. 20) to see if a fix is already documented. If not, please send both the *complete* output from `pimake` and the output from `oommf.tcl +platform` to the OOMMF developers when you e-mail to ask for help.

2.2.4 Installing

The current OOMMF release does not support an installation procedure. For now, simply run the executables from the directories in which they were unpacked/built.

2.2.5 Using OOMMF Software

To start using OOMMF software, run the OOMMF bootstrap application (Sec. 5). This may be launched from the command line interface:

```
tclsh oommf.tcl
```

If you prefer, you may launch the OOMMF bootstrap application `oommf.tcl` using whatever graphical "point and click" interface your operating system provides. By default, the OOMMF bootstrap application will start up a copy of the OOMMF application **mmLaunch** (Sec. 6) in a new window.

If you publish material created with the aid of OOMMF, please refer to Credits (Sec. 22) for citation information.

2.2.6 Reporting Problems

If you encounter problems when installing or using OOMMF, please report them to the OOMMF developers. See Troubleshooting (Sec. 20) for detailed instructions.

2.3 Advanced Installation

The following sections provide instructions for some additional installation options.

2.3.1 Reducing Disk Space Usage

To delete the intermediate files created when building the OOMMF software from source code, use pimake (Sec. 16.9) to build the target `objclean` in the root directory of the OOMMF distribution.

```
tclsh oommf.tcl pimake objclean
```

Running your platform `strip` utility on the OOMMF executable files should also reduce their size somewhat.

2.3.2 Local Customizations

OOMMF software supports local customization of some of its features. All OOMMF programs load the file `config/options.tcl`, which contains customization commands as well as editing instructions. As it is distributed, `config/options.tcl` directs programs to also load the file `config/local/options.tcl`, if it exists. Because future OOMMF releases may overwrite the file `config/options.tcl`, permanent customizations should be made by copying `config/options.tcl` to `config/local/options.tcl` and editing the copy. It is recommended that you leave in the file `config/local/options.tcl` only the customization commands necessary to change those options you wish to modify. Remove all other options so that overwrites by subsequent OOMMF releases are allowed to change the default behavior.

Notable available customizations include the choice of which network port the host service directory application (Sec. 4) uses, and the choice of what program is used for the display of help documentation. By default, OOMMF software uses the application **mmHelp** (Sec. 15), which is included in the OOMMF release, but the help documentation files are standard HTML, so any web browser (for example, Netscape Navigator or Microsoft Internet Explorer) may be used instead. Complete instructions are in the file `config/options.tcl`.

2.3.3 Optimization

In the interest of successful compilation of a usable software package "out of the box," the default configuration for OOMMF does not attempt to achieve much in terms of optimization. However, in each platform's configuration file (for example, `config/cache/wintel.tcl`), there are alternative values for the configuration's optimization flags, available as comments. If you are familiar with your compiler's command line options, you may experiment with other choices as well. You can edit the platform configuration file to replace the default selection with another choice that provides better computing performance. For example, in `config/cache/wintel.tcl`, alternative optimization flags for the MSVC++ compiler are defined with the line:

```
$config SetValue program_compiler_c++_option_opt {format "/G5 /Ox"}
```

The extensible solver, Oxs, can be compiled with debugging support for extensive runtime code checks. This will significantly reduce computation performance. In the standard OOMMF distributions, these checks should be disabled. You may verify this by checking that the following line appears in the file `config/options.tcl`:

```
Oc_Option Add * Platform cflags {-def NDEBUG}
```

To enable these checks, either comment/remove this line, or else add to the `config/local/options.tcl` file a "cflags" option line without "-def NDEBUG", such as

```
Oc_Option Add * Platform cflags {-warn 1}
```

The `config/local/options.tcl` file may be created if it does not already exist.

2.3.4 Managing OOMMF Platform Names

OOMMF software classifies computing platforms into different types using the scripts in the directory `config/names` relative to the root directory of the OOMMF distribution. Each type of computing platform is assigned a unique name. These names are used as directory names and in index and configuration files so that a single OOMMF installation may contain platform-dependent sections for many different types of computing platforms.

To learn what name OOMMF software uses to refer to your computing platform, run

```
tclsh oommf.tcl +platform
```

in the OOMMF root directory.

Changing the name OOMMF assigns to your platform First, use pimake (Sec. 16.9) to build the target `distclean` to clear away any compiled executables built using the old platform name.

```
tclsh oommf.tcl pimake distclean
```

Then, to change the name OOMMF software uses to describe your platform from `foo` to `bar`, simply rename the file

`config/names/foo.tcl` to `config/names/bar.tcl`

and

`config/cache/foo.tcl` to `config/cache/bar.tcl`.

After renaming your platform type, you should recompile your executables using the new platform name.

Adding a new platform type If `oommf.tcl +platform` reports the platform name `unknown`, then none of the scripts in `config/names/` recognizes your platform type. As an example, to add the platform name `foo` to OOMMF's vocabulary of platform names, create the file `config/names/foo.tcl`. The simplest way to proceed is to copy an existing file in the directory `config/names` and edit it to recognize your platform.

The files in `config/names` include Tcl code like this:

```
Oc_Config New _ \
  [string tolower [file rootname [file tail [info script]]]] {
    # In this block place the body of a Tcl proc which returns 1
    # if the machine on which the proc is executed is of the
    # platform type identified by this file, and which returns 0
    # otherwise.
    #
    # The usual Tcl language mechanism for discovering details
    # about the machine on which the proc is running is to
    # consult the global Tcl variable 'tcl_platform'.  See the
    # existing files for examples, or contact the OOMMF
    # developers for further assistance.
}
```

After creating the new platform name file `config/names/foo.tcl`, you also need to create a new platform cache file `config/cache/foo.tcl`. A reasonable starting point is to copy the file `config/cache/unknown.tcl` for editing. Contact the OOMMF developers for assistance.

Please consider contributing your new platform recognition and configuration files to the OOMMF developers for inclusion in future releases of OOMMF software.

Resolving platform name conflicts If the script `oommf.tcl +platform` reports "Multiple platform names are compatible with your computer", then there are multiple files in the directory `config/names/` that return 1 when run on your computer. For each compatible platform name reported, edit the corresponding file in `config/names/` so that only one of them returns 1. Experimenting using `tclsh` to probe the Tcl variable `tcl_platform` should assist you in this task. If that fails, you can explicitly assign a platform type corresponding to your computing platform by matching its hostname. For example, if your machine's host name is `foo.bar.net`:

```
Oc_Config New _ \
  [string tolower [file rootname [file tail [info script]]]] {
    if {[string match foo.bar.net [info hostname]]} {
        return 1
    }
    # Continue with other tests...
}
```

Contact the OOMMF developers if you need further assistance.

2.3.5 Microsoft Windows Options

This section lists installation options for Microsoft Windows.

Using Microsoft Visual C++ If you are building OOMMF software from source using the Microsoft Visual C++ command line compiler, `cl.exe`, it is necessary to run `vcvars32.bat` to set up the path and some environment variables. This file is distributed as part of Visual C++. You may want to set up your system so this batch file gets run automatically when you boot the system, or open a command prompt. See your compiler and system documentation for details.

Using the Cygwin toolkit The Cygwin Project[9] is a free port of the GNU development environment to Windows NT and 9X, which includes the GNU C++ compiler gcc and a port of Tcl/Tk. OOMMF has been successfully built and tested within the Cygwin environment, and sample `config/names/cygtel.tcl` and `config/cache/cygtel.tcl` files are included in the OOMMF distribution. Use `cygtclsh` as your `tclsh` program when configuring, building, and launching OOMMF software.

Note that OOMMF software determines whether it is running with the Cygwin versions of Tcl/Tk by examining the environment variables OSTYPE and TERM. If either is set to a value beginning with `cygwin`, the Cygwin environment is assumed. If you are using the Cygwin environment with a different values for both OSTYPE and TERM, you will have to modify `config/names/cygtel.tcl` accordingly.

Using Borland C++ OOMMF has been successfully built and tested using the Borland C++ command line compiler[10] version 5.5. However, a couple preparatory steps are necessary before building OOMMF with this compiler.

1. Properly complete bcc55 compiler installation.

   Be sure to read the `readme.txt` file in the `BCC55` subdirectory of the Borland install directory. In particular, check that the `bcc32.cfg` and `ilink32.cfg` configuration files exist in the `BIN` directory, and have appropriate contents. If you omit this step you will get error messages during the OOMMF build process relating to the inability of the Borland compiler to find system header files and libraries.

2. Create Borland compatible Tcl and Tk libraries.

   The import libraries distributed with Tcl/Tk, release 8.0.3 and later, are not compatible with the Borland C++ linker. However, the command line utilities `impdef` and `implib`, which are distributed with the Borland compiler, can be used to create suitable libraries from the Tcl/Tk DLL's. In the Tcl/Tk library directory (typically `C:\Tcl\lib` or `"C:\Program Files\Tcl\lib"`), issue the following commands

   ```
   impdef -a tcl83bc.def ..\bin\tcl83.dll
   implib tcl83bc.lib tcl83bc.def
   ```

[9] http://sourceware.cygnus.com/cygwin/
[10] http://www.inprise.com/bcppbuilder/freecompiler/

to create the Borland compatible import library `tcl83bc.lib`. Repeat with "tk" in place of "tcl" to create `tk83bc.lib`. The "-a" switch requests `impdef` to add a leading underscore to function names. This is sufficient for the DLL's shipped with Tcl/Tk 8.3, but other releases may require additional tweaking. The module definition file output by `impdef`, e.g., `tcl83bc.def` above, is a plain text file. You may need to edit this file to add or modify entries.

3. Edit `config\cache\wintel.tcl`

   At a minimum, you will have to change the `program_compiler_c++` value to point to the Borland C++ compiler. The sample `wintel.tcl` cache file assumes the librarian `tlib` and the linker `ilink32` are in the execution path, and that the Borland compatible import libraries made above are in the Tcl/Tk library directory. If this is not the case then you will have to make the additional modifications. Also, you may need to add the "-o" switch to the linker command to force ordinal usage of the Borland compatible Tcl/Tk libraries produced in the previous step.

After this, continue with the instructions in Sec. 2.2.3, Compiling and Linking.

Setting the TCL_LIBRARY environment variable If you encounter difficulties during OOMMF start up, you may need to set the environment variable TCL_LIBRARY.

On Windows NT Bring up the Control Panel (e.g., by selecting **Settings|Control Panel** off the Start menu), and select **System**. Go to the **Environment** tab, and enter TCL_LIBRARY as the Variable, and the name of the directory containing `init.tcl` for the Value, e.g.,

```
%SystemDrive%\Program Files\Tcl\lib\tcl8.0
```

Click **Set** and **OK** to finish.

On Windows 9x Edit the file `autoexec.bat`. Add a line such as the following:

```
set TCL_LIBRARY=C:\Program Files\Tcl\lib\tcl8.0
```

Checking .tcl file association on Windows NT As part of the Tcl/Tk installation, files with the `.tcl` extension are normally associated with the `wish` application. This allows Tcl scripts to be launched from Windows Explorer by double-clicking on their icon, or from the NT command line without specifying the `tclsh` or `wish` shells. If this is not working, you may check your installation from the NT command line as follows. First, run the command "`assoc .tcl`". This should return the file type associated with the `.tcl` extension, e.g., `TclScript`. Next, use the `ftype` command to check the command line associated with that file type, e.g.,

```
C:\> ftype TclScript
 "C:\Program Files\Tcl\bin\wish84.exe" "%1" %2 %3 %4 %5 %6 %7 %8 %9
```

Note that the quotes are required as shown to protect spaces in pathnames. If either `assoc` or `ftype` are incorrect, view the command line help information ("`assoc /?`" and "`ftype /?`") for details on making changes.

Adding an OOMMF shortcut to your desktop Right mouse click on the desktop to bring up the configuration dialog, and select **New|Shortcut**. Enter the command line necessary to bring up OOMMF, e.g.,

```
tclsh84 c:\oommf\oommf.tcl
```

Click **Next>** and enter `OOMMF` for the shortcut name. Select **Finish**.

At this point the shortcut will appear on your desktop with either the tclsh or wish icons. Right mouse click on the icon and select **Properties**. Select the **ShortCut** tab, and bring up **Change Icon...** Under **File Name:** enter the OOMMF icon file, e.g.,

```
C:\oommf\oommf.ico
```

Click **OK**. Back on the **Shortcut** tab, change the **Run:** selection to Minimized. Click **OK** to exit the Properties dialog box. Double clicking on the OOMMF icon should now bring up the OOMMF application **mmLaunch**.

3 Quick Start: Example OOMMF Session

STEP 1: Start up the mmLaunch window.

- At the command prompt, when you are in the OOMMF root directory, type

  `tclsh oommf.tcl`

  (The name of the Tcl shell, rendered here as `tclsh`, may vary between systems. This matter is discussed in Sec. 2.1.) Alternatively, you may launch `oommf.tcl` using whatever "point and click" interface is provided by your operating system.
- This will bring up a small window labeled **mmLaunch**. It will come up in background mode, so you will get another prompt in your original window, even before the **mmLaunch** window appears.

STEP 2: Gain access to other useful windows.

- On the **mmLaunch** window, check the **localhost** box, causing a menu of user account boxes to appear. Check the box corresponding to the account you want to compute on. (On Windows 9X systems, the user account list may consist of only the faux "oommf" user account.) This gives a menu of options:
  - **mmArchive:** to auto-save scalar and vector field data
  - **mmDataTable:** to display current values of scalar outputs
  - **mmDisp:** to display vector fields
  - **mmGraph:** to form x-y plots
  - **mmProbEd:** to grab/modify a problem for **mmSolve2D**
  - **mmSolve2D:** to control the 2D solver
  - **Oxsii:** to control the 3D solver
- Click on **mmDisp**, **mmGraph**, and/or **mmDataTable**, depending on what form of output you want. Use **mmArchive** to save data to disk.

STEP 3a: Run a 2D problem.

Load problem:

- On the **mmLaunch** window, click on the **mmProbEd** button.
- On the **mmProbEd** window, make menu selection **File|Open...** An **Open File** dialog window will appear. On this window:
  - Double click in the **Path** subwindow to change directories. Several sample problems can be found in the directory `oommf/app/mmpe/examples`.
  - To load a problem, double click on a `*.mif` file (e.g., prob1.mif) from the list above the **Filter:** subwindow.

- Modify the problem as desired by clicking on buttons from the main **mmProbEd** window (e.g., **Material Parameters**), and fill out the pop-up forms. A completely new problem may be defined this way.

Initialize solver:

- On the **mmLaunch** window, click on the **mmSolve2D** button to launch an instance of the program **mmSolve2D**.
- Wait for the new solver instance to appear in the **Threads** column in the **mmLaunch** window.
- Check the box next to the **mmSolve2D** entry in the **Threads** column. A window containing an **mmSolve2D** interface will appear.
- On the **mmSolve2D** window:
  - Check **Problem Description** under **Inputs**.
  - Check **mmProbEd** under **Source Threads**.
  - Click **LoadProblem**.
  - A status line will indicate the problem is loading.
  - When the problem is fully loaded, more buttons appear.
  - Check **Scheduled Outputs**.
  - For each desired output (**TotalField**, **Magnetization**, and/or **DataTable**), specify the frequency of update:
    1. Check desired output. This will exhibit the possible output destinations under the Destination Threads heading. Output applications such as **mmDisp**, **mmGraph**, and/or **mmDataTable** must be running to appear in this list.
    2. Check the box next to the desired Destination Thread. This will exhibit Schedule options.
    3. Choose a schedule:
       - **Iteration:** fill in number and check the box.
       - **ControlPoint:** fill in number and check the box.
       - **Interactive:** whenever you click corresponding Interactive output button.

Start calculation:

- On the **mmSolve2D** window, start the calculation with **Run** (which runs until problem completion) or **Relax** (which runs until the next control point is reached).
- If you requested mmDataTable output, check the boxes for the desired quantities on the **mmDataTable** window under the **Data** menu, so that they appear and are updated as requested in your schedule.
- Similarly, check the box for the desired X, Y1, and Y2 quantities on the **mmGraph** window(s) under the **X**, **Y1** and **Y2** menus.

Save results:

- Vector field data (magnetization and effective field) may be interactively written to disk using **mmDisp**, or may be automatically saved via scheduled output to **mmArchive**. For example, to save the magnetization state at each control point, start up an instance of **mmArchive** and select the **ControlPoint** check box for **mmArchive** on the **Magnetization** schedule in the solver. This may be done before starting the calculation. (Control points are points in the simulation where the applied field is stepped. These are typically equilibrium states, but depending on the input `*.mif` file, may be triggered by elapsed simulation time or iteration count.)
- **DataTable** data may be saved by sending scheduled output from the solver to **mmArchive**, which will automatically save all the data it receives. Alternatively, **mmGraph** can perform this function. Schedule output to **mmGraph** as desired, and use either the interactive or automated save functionality of **mmGraph** (Sec. 12). You can setup the solver data scheduling before the calculation is started, but must wait for the first data point to configure **mmGraph** before saving any data. As a workaround, you may configure **mmGraph** by sending it the initial solver state interactively, and then use the **Options|clear Data** menu item in **mmGraph** to remove the initializing data point.

Midcourse control:

- On the **mmSolve2D** window, buttons can stop and restart the calculation:
  - **Reset:** Return to beginning of problem.
  - **LoadProblem:** Restart with a new problem.
  - **Run:** Apply a sequence of fields until all complete.
  - **Relax:** Run the ODE at the current applied field until the next control point is reached.
  - **Pause:** Click anytime to stop the solver. Continue simulation from paused point with **Run** or **Relax**.
  - **Field−:** Apply the previous field again.
  - **Field+:** Apply the next field in the list.
- Output options can be changed and new output windows opened.

STEP 3b: Run a 3D problem.

Launch solver:

- On the **mmLaunch** window, click on the **Oxsii** button to launch an instance of the program **Oxsii**.
- Wait for the new solver instance to appear in the **Threads** column in the **mmLaunch** window.
- Check the box next to the **Oxsii** entry in the **Threads** column. A window containing an **Oxsii** interface will appear.

Load problem:

- On the **Oxsii** window, select the **File|Load...** menu option. A **Load Problem** dialog box will appear. On this window:
  - Double click in the **Path** subwindow to change directories. Several sample problems can be found in the directory `oommf/app/oxs/examples`.
  - To load a problem, double click on a `*.mif` file (e.g., stdprob1.mif) from the list above the **Filter:** subwindow.

  There is currently no analogue to **mmProbEd** for the 3D solver, so the input files must be composed by hand. For details, see the MIF 2.1 (Sec. 17.1) and Oxs_Ext Child Class (Sec. 7.3) documentation. Note that MIF 1.x (i.e., 2D problem) files are not readable by **Oxsii**.
- The status line in the **Oxsii** interface window will indicate the problem is loading.
- When the problem is fully loaded, the status line will show "Pause", and the top row of buttons (**Reload**, **Reset**, ...) will become active. Also, the Output list will fill with available outputs.
- Set up scheduled outputs. For each desired output
  1. Select the source from the Output list.
  2. Select the receiver from the Destination list.
  3. Specify the frequency of update:
     - **Step:** fill in number and check the box.
     - **Stage:** fill in number and check the box.

  The items in the Output list will vary depending on the problem that was loaded. The items in the Destination list reflect the OOMMF data display and archiving programs currently running.

Start calculation:

- On the **Oxsii** window, start the calculation with **Run**, **Relax**, or **Step**.
- If you requested mmDataTable output, check the boxes for the desired quantities on the **mmDataTable** window under the **Data** menu, so that they appear and are updated as requested in your schedule.
- Similarly, check the box for the desired X, Y1, and Y2 quantities on the **mmGraph** window(s) under the **X**, **Y1** and **Y2** menus.

Save results:

- Vector field data (magnetization and effective field) may be interactively written to disk using **mmDisp**, or may be automatically saved via scheduled output to **mmArchive**. For example, to save the magnetization state at the end of each problem stage, start up an instance of **mmArchive** and select the **Stage** check box for the `Magnetization` output, `mmArchive` destination pair. (Stages denote points in the simulation where some significant event occurs, such as when an

equilibrium is reached or some preset simulation time index is met. These criteria are set by the input MIF file.)

- Tabular data may be saved by sending scheduled output from the solver to **mmArchive**, which will automatically save all the data it receives. Alternatively, **mmGraph** can perform this function. Schedule output to **mmGraph** as desired, and use either the interactive or automated save functionality of **mmGraph** (Sec. 12). You can setup the solver data scheduling before the calculation is started, but must wait for the first data point to configure **mmGraph** before saving any data. As a workaround, you may configure **mmGraph** by sending it the initial solver state interactively, and then use the **Options|clear Data** menu item in **mmGraph** to remove the initializing data point.

Midcourse control:

- On the **Oxsii** window, buttons can stop and restart the calculation:
  - **Reload:** Reload the same file from disk.
  - **Reset:** Return to problem start.
  - **Run:** Step through all stages until all complete.
  - **Relax:** Run until the current stage termination criteria are met.
  - **Step:** Do one solver iteration and then pause.
  - **Pause:** Click anytime to stop the solver. Continue simulation from paused point with **Run**, **Relax** or **Step**.
  - **Stage:** Interactively change the current stage index by either typing the desired stage number (counting from 0) into the **Stage** entry box or by moving the associated slider.
- Output options can be changed and new output windows opened. The **Send** button in the **Oxsii** Schedule subwindow is used to interactively send output to the selected Output + Destination pair.

STEP 4: Exit OOMMF.

- Individual OOMMF applications can be terminated by selecting the **File|Exit** menu item from their interface window.
- Selecting **File|Exit** on the **mmLaunch** window will close the **mmLaunch** window, and also the interface windows for any **mmArchive**, **mmSolve2D**, and **Oxsii** applications. However, those applications will continue to run in the background, and their interfaces may be re-displayed by starting a new **mmLaunch** instance.
- To automatically kill all OOMMF applications, select the **File|Exit All OOMMF** option from the **mmLaunch** menu bar.

4 OOMMF Architecture Overview

Before describing each of the applications which comprise the OOMMF software, it is helpful to understand how these applications work together. OOMMF is not structured as a single program. Instead it is a collection of programs, each specializing in some task needed as part of a micromagnetic simulation system. An advantage of this modular architecture is that each program may be improved or even replaced without a need to redesign the entire system.

The OOMMF programs work together by providing services to one another. The programs communicate over Internet (TCP/IP) connections, even when the programs are running on a common host. An advantage of this design is that distributed operation of OOMMF programs over a networked collection of hosts is supported in the basic design, and will be available in a future release.

When two OOMMF applications are in the relationship that one is requesting a service from the other, it is convenient to introduce some clarifying terminology. Let us refer to the application that is providing a service as the "server application" and the application requesting the service as the "client application." Note that a single application can be both a server application in one service relationship and a client application in another service relationship.

Each server application provides its services on a particular Internet port, and needs to inform potential client applications how to obtain its service. Each client application needs to be able to look up possible providers of the service it needs. The intermediary which brings server applications and client applications together is another application called the "account service directory." There may be at most one account service directory application running under the user ID of each user account on a host. Each account service directory keeps track of all the services provided by OOMMF server applications running under its user account on its host and the corresponding Internet ports at which those services may be obtained. OOMMF server applications register their services with the corresponding account service directory application. OOMMF client applications look up service providers running under a particular user ID in the corresponding account server directory application.

The account service directory applications simplify the problem of matching servers and clients, but they do not completely solve it. OOMMF applications still need a mechanism to find out how to obtain the service of the account service directory! Another application, called the "host service directory" serves this function. Only one copy of the host service directory application runs on each host. Its sole purpose is to tell OOMMF applications where to obtain the services of account service directories on that host. Because only one copy of this application runs per host, it can provide its service on a well-known port which is configured into the OOMMF software. By default, this is port 15136. OOMMF software can be customized (Sec. 2.3.2) to use a different port number.

The account service directory applications perform another task as well. They launch other programs under the user ID for which they manage service registration. The user controls the launching of programs through the interface provided by the application **mm-**

Launch (See Sec. 6), but it is the account service directory application that actually spawns a subprocess for the new application. Because of this architecture, most OOMMF applications are launched as child processes of an account service directory application. These child processes inherit their environment from their parent account service directory application, including their working directory, and other key environment variables, such as `DISPLAY`. Each account service directory application sets its working directory to the root directory of the OOMMF distribution. Future releases of OOMMF software will likely be based on a revised architecture which alleviates these restrictions.

These service directory applications are vitally important to the operation of the total OOMMF micromagnetic simulation system. However, it would be easy to overlook them. They act entirely "behind the scenes" without a user interface window. Furthermore, they are never launched by the user. When any server application needs to register its service, if it finds that these service directory applications are not running, it launches new copies of them. In this way the user can be sure that if any OOMMF server applications are running, then so are the service directory applications needed to direct clients to its service. After all server applications terminate, and there are no longer any services registered with a service directory application, it terminates as well. Similarly, when all service directory applications terminate, the host service directory application exits.

In the sections which follow, the OOMMF applications are described in terms of the services they provide and the services they require.

5 Command Line Launching

Some of the OOMMF applications are platform-independent Tcl scripts. Some of them are Tcl scripts that require special platform-dependent interpreters. Others are platform-dependent, compiled C++ applications. It is likely that some of them will change status in later releases of OOMMF. Each of these types of application requires a different command line for launching. Rather than require all OOMMF users to manage this complexity, we provide a pair of programs that provide simplified interfaces for launching OOMMF applications.

The first of these is used to launch OOMMF applications from the command line. Because its function is only to start another program, we refer to this program as the "bootstrap application." The bootstrap application is the Tcl script `oommf.tcl`. In its simplest usage, it takes a single argument on the command line, the name of the application to launch. For example, to launch **mmGraph** (Sec. 12), the command line is:

```
tclsh oommf.tcl mmGraph
```

The search for an application matching the name is case-insensitive. (Here, as elsewhere in this document, the current working directory is assumed to be the OOMMF root directory. For other cases, adjust the pathname to `oommf.tcl` as appropriate.) As discussed in Sec. 2.1, the name of the Tcl shell, rendered here as `tclsh`, may vary between systems.

If no command line arguments are passed to the bootstrap application, by default it will launch the application **mmLaunch** (Sec. 6).

Any command line arguments to the bootstrap application that begin with the character '`+`' modify its behavior. For a summary of all command line options recognized by the bootstrap application, run:

```
tclsh oommf.tcl +help
```

The command line arguments `+bg` and `+fg` control how the bootstrap behaves after launching the requested application. It can exit immediately after launching the requested application in background mode (`+bg`), or it can block until the launched application exits (`+fg`). Each application registers within the OOMMF system whether it prefers to be launched in foreground or background mode. If neither option is requested on the command line, the bootstrap launches the requested application in its preferred mode.

The first command line argument that does not begin with the character `+` is interpreted as a specification of which application should be launched. As described above, this is usually the simple name of an application. When a particular version of an application is required, though, the bootstrap allows the user to include that requirement as part of the specification. For example:

```
tclsh oommf.tcl "mmGraph 1.1"
```

will guarantee that the instance of the application mmGraph it launches is of at least version 1.1. If no copy of mmGraph satisfying the version requirement can be found, an error is reported.

The rest of the command line arguments that are not recognized by the bootstrap are passed along as arguments to the application the bootstrap launches. Since the bootstrap recognizes command line arguments that begin with `+` and most other applications recognize command line arguments that begin with `-`, confusion about which options are provided to which programs can be avoided. For example,

```
tclsh oommf.tcl +help mmGraph
```

prints out help information about the bootstrap and exits without launching mmGraph. However,

```
tclsh oommf.tcl mmGraph -help
```

launches mmGraph with the command line argument `-help`. mmGraph then displays its own help message.

All OOMMF applications accept the standard options listed below. Some of the OOMMF applications accept additional arguments when launched from the command line, as documented in the corresponding sections of this manual. When an option argument is specified as `<0|1>`, `0` typically means off, no or disable, and `1` means on, yes or enable.

-version Display the version of the application and exit.

-help Display a help message and exit.

-tk <0|1> Disable or enable Tk. Tk must be enabled for an application to display graphical widgets. However, when Tk is enabled, on Unix platforms the application is dependent on an X Windows server. If the X Windows server dies, it will kill the application. Long-running applications that do not inherently use display widgets support disabling of Tk with `-tk 0`. Other applications that must use display widgets are unable to run with the option `-tk 0`. To run applications that require `-tk 1` on a Unix system with no display, one might use Xvfb[11] .

-cwd directory Set the current working directory of the application.

-console Display a console widget in which Tcl commands may be interactively typed into the application. Useful for debugging.

In addition, those applications which enable Tk accept additional Tk options, such as `-display`. See the Tk documentation for details.

The bootstrap application should be infrequently used by most users. The application **mmLaunch** (Sec. 6) provides a more convenient graphical interface for launching applications. The main uses for the bootstrap application are launching **mmLaunch**, launching **pimake**, launching programs which make up the OOMMF Batch System (Sec. 10.2) and other programs that are inherently command line driven, and in circumstances where the user wishes to precisely control the command line arguments passed to an OOMMF application or the environment in which an OOMMF application runs.

[11] http://www.itworld.com/AppDev/1461/UIR000330xvfb/

Platform Issues

On most Unix platforms, if `oommf.tcl` is marked executable, it may be run directly, i.e., without specifying `tclsh`. This works because the first few lines of the `oommf.tcl` Tcl script are:

```
#!/bin/sh
# \
exec tclsh "$0" ${1+"$@"}
```

When run, the first `tclsh` on the execution path is invoked to interpret the `oommf.tcl` script. If the Tcl shell program cannot be invoked by the name `tclsh` on your computer, edit the first lines of `oommf.tcl` to use the correct name. Better still, use symbolic links or some other means to make the Tcl shell program available by the name `tclsh`. The latter solution will not be undone by file overwrites from OOMMF upgrades.

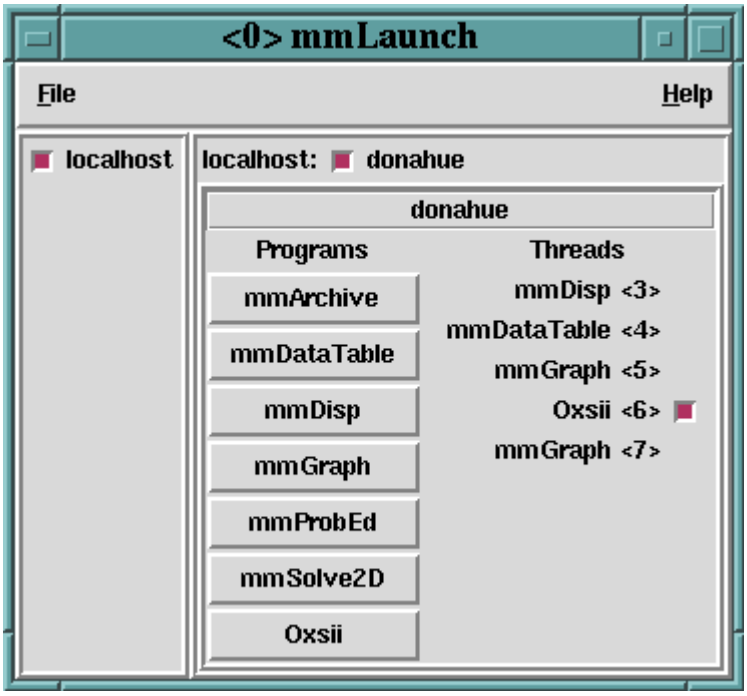
If in addition, the directory `.../path/to/oommf` is in the execution path, the command line can be as simple as:

```
oommf.tcl appName
```

from any working directory.

On Windows platforms, because `oommf.tcl` has the file extension `.tcl`, it is normally associated by Windows with the `wish` interpreter. The `oommf.tcl` script has been specially written so that either `tclsh` or `wish` is a suitable interpreter. This means that simply double-clicking on an icon associated with the file `oommf.tcl` (say, in Windows Explorer) will launch the bootstrap application with no arguments. This will result in the default behavior of launching the application **mmLaunch**, which is suitable for launching other OOMMF applications. (If this doesn't work, refer back to the Windows Options section in the installation instructions, Sec. 2.3.5.)

6 OOMMF Launcher/Control Interface: mmLaunch



Overview

The application **mmLaunch** launches, monitors, and controls other OOMMF applications. It is the OOMMF application which is most closely connected to the account service directory and host service directory applications that run behind the scenes. It also provides user interfaces to any applications, notably **Oxsii** (Sec. 7.1) and **mmSolve2D** (Sec. 10.1), that do not have their own user interface window.

Launching

mmLaunch should be launched using the bootstrap application (Sec. 5). The command line is

```
tclsh oommf.tcl mmLaunch [standard options]
```

Controls

Upon startup, **mmLaunch** displays a panel of checkbuttons, one for each host service directory to which it is connected. In the current release of OOMMF there is only one checkbutton—**localhost**. Future releases of **mmLaunch** will be able to connect to remote

hosts as well. If there is no host service directory running on the localhost when **mmLaunch** is launched, **mmLaunch** will start one. In that circumstance, there may be some delay before the **localhost** check button appears.

Toggling the **localhost** checkbutton toggles the display of an interface to the host service directory. The host service directory interface consists of a row of checkbuttons, one for each account service directory registered with the host service directory. Each checkbutton is labeled with the user ID of the corresponding account service directory. For most users, there will be only one checkbutton, labeled with the user's own account ID, except on Windows 9X, where the dummy account ID "oommf" may be displayed instead. If there is no account service directory running for the account under which **mmLaunch** was launched, **mmLaunch** will start one. In that circumstance, there may be some delay before the account checkbutton appears.

Toggling an account checkbutton toggles the display of an interface to the corresponding account service directory. The account service directory interface consists of two columns. The **Programs** column contains buttons labeled with the names of OOMMF applications that may be launched under the account managed by this account service directory. Clicking on one of these buttons launches the corresponding application. Only one click is needed, though there will be some delay before the launched application displays a window to the user. Multiple clicks will launch multiple copies of the application. Note: The launching is actually handled by the account service directory application (Sec. 4), which sets the initial working directory to the OOMMF root directory.

The **Threads** column is a list of all the OOMMF applications currently running under the account that are registered with the account service directory. The list includes both the application name and an ID number by which multiple copies of the same application may be distinguished. This ID number is also displayed in the title bar of the corresponding application's user interface window. When an application exits, its entry is automatically removed from the Threads list.

Any of the running applications that do not provide their own interface window will be displayed in the **Threads** list with a checkbutton. The checkbutton toggles the display of an interface which **mmLaunch** provides on behalf of that application. The only OOMMF applications currently using this service are the 3D solvers **Oxsii** and **Boxsi** (Sec. 7), the 2D solvers **mmSolve2D** and **batchsolve** (Sec. 10), and the archive application **mmArchive** (Sec. 14). These interfaces are described in the documentation for the corresponding applications.

The menu selection **File|Exit** terminates the **mmLaunch** application, and the **File|Exit All OOMMF** selection terminates all applications in the Threads list, and then exits **mmLaunch**. The menu **Help** provides the usual help facilities.

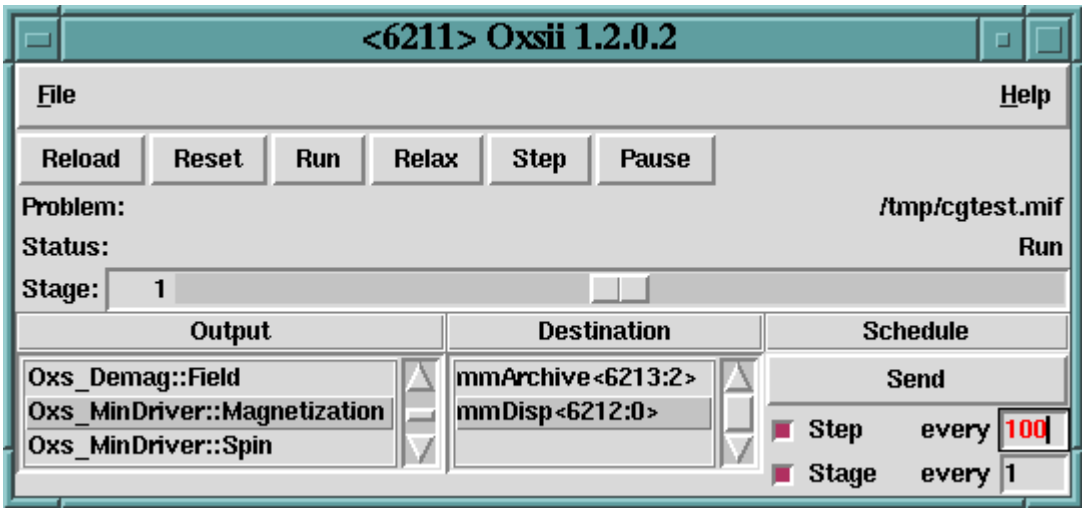
7 OOMMF eXtensible Solver

The Oxs (OOMMF eXtensible Solver) is an extensible micromagnetic computation engine capable of solving problems defined on three-dimensional grids of rectangular cells holding three-dimensional spins. There are two interfaces provided to Oxs: the interactive interface Oxsii (Sec. 7.1) intended to be controlled primarily through a graphical user interface, and the batch mode Boxsi (Sec. 7.2), which has extended command line controls making it suitable for use in shell scripts.

Problem definition for Oxs is accomplished using input files in the MIF 2.1 format (Sec. 17.1). This is an extensible format; the standard OOMMF modules are documented in Sec. 7.3 below.

Note on Tk dependence: Some MIF 2 problem descriptions rely on external image files; examples include those using the `Oxs_ImageAtlas` class Sec. 7.3.1, or those using the MIF 2 `ReadFile` command with the `image` translation specification Sec. 17.1.1. If the image file is not in the PPM P3 (text) format, then the **any2ppm** application may be launched to read and convert the file. Since **any2ppm** requires Tk, at the time the image file is read a valid display must be available. See the **any2ppm** documentation (Sec. 16.1) for details.

7.1 OOMMF eXtensible Solver Interactive Interface: Oxsii



Overview

The application **Oxsii** is the graphical, interactive user interface to the Oxs micromagnetic computation engine. Within the OOMMF architecture (see Sec. 4), **Oxsii** is both a server and a client application. **Oxsii** is a client of data table display and storage applications, and vector field display and storage applications. **Oxsii** is the server of a solver control service for which the only client is **mmLaunch** (Sec. 6). It is through this service that **mmLaunch** provides a user interface window (shown above) on behalf of **Oxsii**.

A micromagnetic problem is communicated to **Oxsii** via a MIF 2.1 file, which defines a collection of Oxs_Ext objects that comprise the problem model. The problem description includes a segmentation of the lifetime of the simulation into stages. Stages mark discontinuous changes in model attributes, such as applied fields, and also serve to mark coarse grain simulation progress. **Oxsii** provides controls to advance the simulation, stopping between iterations, between stages, or only when the run is complete. Throughout the simulation, the user may save and display intermediate results, either interactively or via scheduling based on iteration and stage counts.

Launching

Oxsii may be started either by selecting the **Oxsii** button on mmLaunch, or from the command line via

```
tclsh oommf.tcl oxsii [standard options] [-parameters params] \
   [-pause <0|1>] [-nice <0|1>] [-exitondone <0|1>] [miffile]
```

where

-parameters params Sets MIF 2.1 format (Sec. 17.1) file parameters. The *params* argument should be a list with an even number of arguments, corresponding to name + value pairs. This list must be quoted so it is presented to **Oxsii** as a single item on the command line. The quoting mechanism is shell/operating system specific. Refer to your system documentation for details.

-nice <0|1> If enabled (i.e., 1), then the program will drop its scheduling priority after startup. The default is 1, i.e., to yield scheduling priority to other applications.

-pause <0|1> If enabled (i.e., 1), then the program automatically pauses after loading the specified *miffile*. The default is 0, i.e., to automatically move into "Run" mode once the problem is loaded. This switch has no effect if miffile is not specified.

-exitondone <0|1> Whether to exit after solution of the problem is complete. Default is to simply await the interactive selection of another problem to be solved.

miffile Load and solve the problem found in *miffile*, which must be in the MIF 2.1 format. Optional.

All the above switches are optional.

Since **Oxsii** does not present any user interface window of its own, it depends on **mmLaunch** to provide an interface on its behalf. The entry for an instance of **Oxsii** in the **Threads** column of any running copy of **mmLaunch** has a checkbutton next to it. This button toggles the presence of a user interface window through which the user may control that instance of **Oxsii**.

Inputs

Unlike **mmSolve2D** (Sec. 10.1), **Oxsii** loads problem specifications directly from disk (via the `File|Load...` menu selection), rather than through **mmProbEd** (Sec. 8) or **FileSource** (Sec. 9). Also, input files for **Oxsii** must be in the MIF 2.1 (Sec. 17.1) format, as opposed to the older MIF 1.1 format used by the 2D solver. There are sample MIF 2.1 files in the directory `oommf/app/oxs/examples`. The command line tool **mifconvert** (Sec. 16.7) can be used as an aid for converting MIF 1.1 files to the MIF 2.1 format. MIF files may be edited with any plain text editor.

Outputs

Once a problem has been loaded, the scroll box under the Output heading will fill with a list of available outputs. The contents of this list will depend upon the `Oxs_Ext` objects specified in the input MIF file. Refer to the documentation for those objects for specific details (Sec. 7.3). To send output from **Oxsii** to another OOMMF application, highlight the desired selection under the Output heading, make the corresponding selection under the Destination heading, and then specify the output timing under the Schedule heading. Outputs may be scheduled by the step or stage, and may be sent out interactively by pressing the **Send** button. The initial output configuration is set by `Destination` and `Schedule` commands in the input MIF file (Sec. 17.1.1).

Outputs fall under two general categories: scalar (single-valued) outputs and vector field outputs. The scalar outputs are grouped together as the `DataTable` entry in the Output scroll box. Scalar outputs include such items as total and component energies, average magnetization, stage and iteration counts, max torque values. When the `DataTable` entry is selected, the Destination box will list all OOMMF applications accepting datatable-style input, i.e., all currently running **mmDataTable** (Sec. 11), **mmGraph** (Sec. 12), and **mmArchive** (Sec. 14) processes.

The vector field outputs include pointwise magnetization, various total and partial magnetic fields, and torques. Unlike the scalar outputs, the vector field outputs are listed individually in the Output scroll box. Allowed destinations for vector field output are running instances of **mmDisp** (Sec. 13) and **mmArchive** (Sec. 14). Caution is advised when scheduling vector field output, especially with large problems, because the output may run many megabytes.

Controls

The **File** menu button holds 5 entries: Load, Show Console, Close Interface, Clear Schedule and Exit Oxsii. **File|Load...** launches a dialog box that allows the user to select an input MIF problem description file. **File|Show Console** brings up a Tcl shell console running off the **Oxsii** interface Tcl interpreter. This console is intended primary for debugging purposes. In particular, output from MIF Report commands (Sec. 17.1.1) may be viewed here. **File|Close Interface** will remove the interface window from the display, but leaves the

solver running. This effect may also be obtained by deselecting the **Oxsii** interface button in the **Threads** list in **mmLaunch**. **File|Clear Schedule** will disable all currently active output schedules, exactly as if the user clicked through the interactive schedule interface one output and destination at a time and disabled each schedule-enabling checkbutton. The final entry, **File|Exit Oxsii**, terminates the **Oxsii** solver and closes the interface window.

The **Help** menu provides the usual help facilities.

The row of buttons immediately below the menu bar provides simulation progress control. These buttons become active once a problem has been loaded. The first button, **Reload**, re-reads the most recent problem MIF input file, re-initializes the solver, and pauses. **Reset** is similar, except the file is not re-read. The remaining four buttons, **Run**, **Relax**, **Step** and **Pause** place the solver into one of four *run-states*. In the Pause state, the solver sits idle awaiting further instructions. If **Step** is selected, then the solver will move forward one iteration and then Pause. In Relax mode, the solver takes at least one step, and then runs until it reaches a stage boundary, at which point the solver is paused. In Run mode, the solver runs until the end of the problem is reached. Interactive output is available in all modes; the scheduled outputs occur appropriately as the step and stage counts advance.

Directly below the progress control buttons are two display lines, showing the name of the input MIF file and the current run-state. Below the run-state `Status` line is the stage display and control bar. The simulation stage may be changed at any time by dragging the scroll bar or by typing the desired stage number into the text display box to the left of the scroll bar. Valid stage numbers are integers from 0 to $N - 1$, where N is the number of stages specified by the MIF input file.

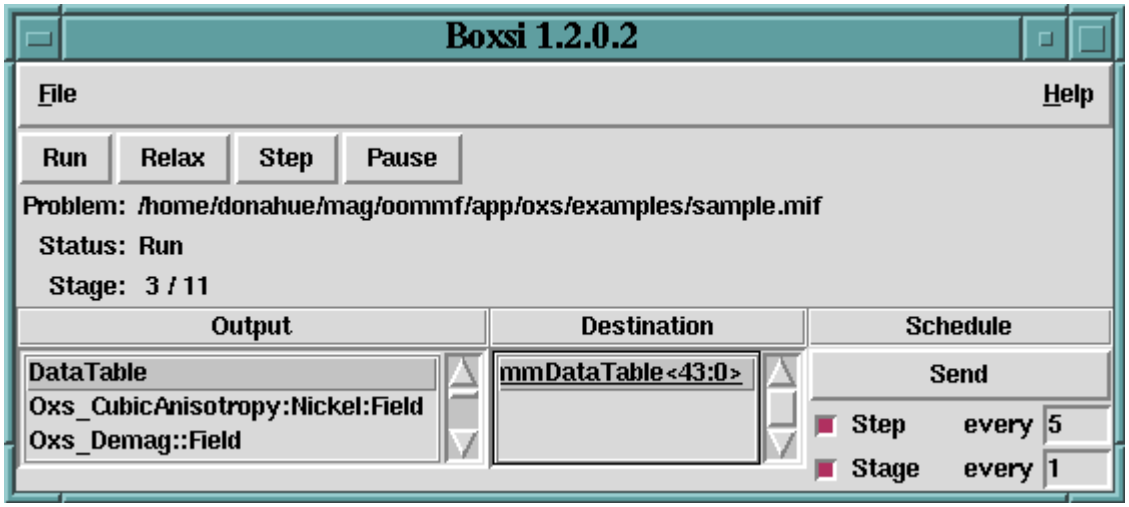
Details

The simulation model construction is governed by the Specify blocks in the input MIF file. Therefore, all aspects of the simulation are determined by the specified Oxs_Ext classes (Sec. 7.3). Refer to the appropriate Oxs_Ext class documentation for simulation and computational details.

Known Bugs

A bug in the network traffic handling code of Tcl on Windows 9X systems can sometimes interfere with communications between the control interface of **Oxsii** and the actual computation engine. If **Oxsii** is sending out data to two or more data display services every iteration, the network traffic from that data can "crowd out" the receipt of control messages from the control interface. You may observe this as a long delay between the time you click the **Pause** button and the time the solver stops iterating. This bug first appeared in Tcl release 8.0.3, and remained through Tcl release 8.1.1. It is fixed in Tcl releases 8.2 and later, which we recommend for OOMMF users on Windows 9X systems. Other platforms do not have this problem.

7.2 OOMMF eXtensible Solver Batch Interface: boxsi



Overview

The application **Boxsi** provides a batch mode interface to the Oxs micromagnetic computation engine. A restricted graphical interface is provided, but **Boxsi** is primarily intended to be controlled by command line arguments, and launched by the user either directly from the shell prompt or from inside a batch file.

Within the OOMMF architecture (see Sec. 4), **Boxsi** is both a server and a client application. It is a client of data table display and storage applications, and vector field display and storage applications. **Boxsi** is the server of a solver control service for which the only client is **mmLaunch** (Sec. 6). It is through this service that **mmLaunch** provides a user interface window (shown above) on behalf of **Boxsi**.

A micromagnetic problem is communicated to **Boxsi** through a MIF 2.1 file specified on the command line and loaded from disk.

Launching

Boxsi must be started from the command line. The syntax is

```
tclsh oommf.tcl boxsi [standard options] [-parameters params] \
   [-pause <0|1>] [-nice <0|1>] [-exitondone <0|1>] [-kill tags] miffile
```

where

-parameters params Sets MIF 2.1 format (Sec. 17.1) file parameters. The *params* argument should be a list with an even number of arguments, corresponding to name + value pairs. This list must be quoted so it is presented to **Boxsi** as a single item on the command line. The quoting mechanism is shell/operating system specific. Refer to your system documentation for details.

-nice <0|1> If enabled (i.e., 1), then the program will drop its scheduling priority after startup. The default is 0, i.e., to retain its original scheduling priority.

-pause <0|1> If enabled (i.e., 1), then the program automatically pauses after loading the specified problem file. The default is 0, i.e., to automatically move into "Run" mode once the problem is loaded.

-exitondone <0|1> Whether to exit after solution of the problem is complete, or to await the interactive selection of the **File|Exit** command. The default is 1, i.e., automatically exit when done.

-kill tags On termination, sends requests to other applications to shutdown too. The *tags* argument should be either a list of destination tags (which are declared by `Destination` commands, Sec. 17.1.1) from the input MIF file, or else the keyword `all`, which is interpreted to mean all the destination tags.

miffile Load and solve the problem found in *miffile*, which must be in the MIF 2.1 format. Required.

Although **Boxsi** cannot be launched by **mmLaunch**, nonetheless a limited graphical interactive interface for **Boxsi** is provided through **mmLaunch**, in the same manner as is done for **Oxsii**. Each running instance of **Boxsi** is included in the **Threads** list of **mmLaunch**, along with a checkbutton. This button toggles the presence of a user interface window.

Inputs

Boxsi loads problem specifications directly from disk as requested on the command line. The format for these files is the MIF 2.1 (Sec. 17.1) format, the same as used by the **Oxsii** interactive interface. The older MIF 1.1 (Sec. 17.2) used by the 2D solver **mmSolve2D** cannot be read by **Boxsi**, but the command line tool **mifconvert** (Sec. 16.7) can be used as an aid for converting MIF 1.1 files to the MIF 2.1 format. Sample MIF 2.1 files can be found in the directory `oommf/app/oxs/examples`.

Outputs

The lower panel of the **Boxsi** interactive interface presents Output, Destination, and Schedule sub-windows that display the current output configuration and allow interactive modification of that configuration. These controls are identical to those in the **Oxsii** user interface; refer to the **Oxsii** documentation (Sec. 7.1) for details. The only difference between **Boxsi** and **Oxsii** with respect to outputs is that in practice **Boxsi** tends to rely primarily on `Destination` and `Schedule` commands in the input MIF file (Sec. 17.1.1) to setup the output configuration. The interactive output interface is used for incidental runtime monitoring of the job.

Controls

The runtime controls provided by the **Boxsi** interactive interface are a restricted subset of those available in the **Oxsii** interface. If the runtime controls provided by **Boxsi** are found to be insufficient for a given task, consider using **Oxsii** instead.

The **File** menu holds 4 entries: Show Console, Close Interface, Clear Schedule, and Exit Oxsii. **File|Show Console** brings up a Tcl shell console running off the **Boxsi** interface Tcl interpreter. This console is intended primary for debugging purposes. **File|Close Interface** will remove the interface window from the display, but leaves the solver running. This effect may also be obtained by deselecting the **Boxsi** interface button in the **Threads** list in **mmLaunch**. **File|Clear Schedule** will disable all currently active output schedules, exactly as if the user clicked through the interactive schedule interface one output and destination at a time and disabled each schedule-enabling checkbutton. The final entry, **File|Exit Boxsi**, terminates the **Boxsi** solver and closes the interface window. Note that there is no **File|Load...** menu item; the problem specification file must be declared on the **Boxsi** command line.

The **Help** menu provides the usual help facilities.

The row of buttons immediately below the menu bar provides simulation progress control. These buttons—**Run**, **Relax**, **Step** and **Pause**—become active once the micromagnetic problem has been initialized. These buttons allow the user to change the run state of the solver. In the Pause state, the solver sits idle awaiting further instructions. If **Step** is selected, then the solver will move forward one iteration and then Pause. In Relax mode, the solver takes at least one step, and then runs until it reaches a stage boundary, at which point the solver is paused. In Run mode, the solver runs until the end of the problem is reached. When the problem end is reached, the solver will either pause or exit, depending upon the setting of the `-exitondone` command line option.

Normally the solver progresses automatically from problem initialization into Run mode, but this can be changed by the `-pause` command line switch. Interactive output is available in all modes; the scheduled outputs occur appropriately as the step and stage counts advance.

Directly below the run state control buttons are three display lines, showing the name of the input MIF file, the current run-state, and the current stage number/maximum stage number. Both stage numbers are 0-indexed.

Details

As with **Oxsii**, the simulation model construction is governed by the Specify blocks in the input MIF file, and all aspects of the simulation are determined by the specified Oxs_Ext classes (Sec. 7.3). Refer to the appropriate Oxs_Ext class documentation for simulation and computational details.

Known Bugs

Boxsi suffers from the same Windows 9X Tcl network traffic bug as **Oxsii**. OOMMF users on Windows 9X systems are encouraged to upgrade to Tcl 8.2 or later.

7.3 Standard Oxs_Ext Child Classes

An Oxs simulation is built as a collection of `Oxs_Ext` (Oxs Extension) objects. These are defined via Specify blocks in the input MIF 2.1 file (Sec. 17.1). The reader will find the information and sample MIF file, Fig. 5, provided in that section to be a helpful adjunct to the material presented below.

This section describes the `Oxs_Ext` classes available in the standard OOMMF distribution, including documentation of their Specify block initialization strings. The standard `Oxs_Ext` objects, i.e., those that are distributed with OOMMF, can be identified by the `Oxs_` prefix in their names. Additional `Oxs_Ext` classes may be available on your system. Check local documentation for details.

In the following presentation, the `Oxs_Ext` classes are organized into 8 categories: atlases, meshes, energies, evolvers, drivers, scalar field objects, vector field objects, and MIF support classes. The following `Oxs_Ext` classes are currently available:

- Atlases
  `Oxs_BoxAtlas` `Oxs_ImageAtlas`
  `Oxs_MultiAtlas` `Oxs_ScriptAtlas`
- Meshes
  `Oxs_RectangularMesh`
- Energies
  `Oxs_CubicAnisotropy` `Oxs_Demag`
  `Oxs_Exchange6Ngbr` `Oxs_ExchangePtwise`
  `Oxs_FixedZeeman` `Oxs_RandomSiteExchange`
  `Oxs_ScriptUZeeman` `Oxs_SimpleDemag`
  `Oxs_StageZeeman` `Oxs_TransformZeeman`
  `Oxs_TwoSurfaceExchange` `Oxs_UniaxialAnisotropy`
  `Oxs_UniformExchange` `Oxs_UZeeman`
- Evolvers
  `Oxs_CGEvolve` `Oxs_EulerEvolve`
- Drivers
  `Oxs_MinDriver` `Oxs_TimeDriver`
- Scalar Field Objects
  `Oxs_AtlasScalarField` `Oxs_LinearScalarField`
  `Oxs_RandomScalarField` `Oxs_ScriptScalarField`
  `Oxs_UniformScalarField`

- Vector Field Objects
  `Oxs_AtlasVectorField` `Oxs_FileVectorField`
  `Oxs_PlaneRandomVectorField` `Oxs_RandomVectorField`
  `Oxs_ScriptVectorField` `Oxs_UniformVectorField`
- MIF Support Classes
  `Oxs_LabelValue`

7.3.1 Atlases

Geometric volumes of spaces are specified in Oxs via *atlases*, which divide their domain into one or more disjoint subsets called *regions*. Each atlas definition also specifies a bounding box, i.e., an axes parallel rectangular parallelepiped that contains all the regions. The most commonly used atlases are the simple `Oxs_BoxAtlas` and the compound `Oxs_MultiAltas`.

Oxs_BoxAtlas: An axes parallel rectangular parallelepiped, containing a single region that is coterminous with the atlas itself. The specify block has the form

```
Specify Oxs_BoxAtlas:atlasname {
    xrange { xmin xmax }
    yrange { ymin ymax }
    zrange { zmin zmax }
    name regionname
}
```

where *xmin, xmax, ...* are coordinates in meters, specifying the extents of the volume being defined. The *regionname* label specifies the name assigned to the region contained in the atlas. The **name** entry is optional; if not specified then the region name is taken from the object instance name, i.e., *atlasname*.

Oxs_ImageAtlas: This class is designed to allow an image file to be used to define regions in terms of colors in the image. It is intended for use in conjunction with the `Oxs_AtlasScalarField` and `Oxs_AtlasVectorField` classes in circumstances where a small number of distinct species (materials) are being modeled. This is a generalization of the mask file functionality of the 2D solver (Sec. 17.2.3).

For situations requiring continuous variation in material parameters, the script field classes should be used in conjunction with the `ReadFile` MIF extension command. See the `ColorField` sample proc in the `ReadFile` documentation in Sec. 17.1.1 for an example of this technique.

The `Oxs_ImageAtlas` Specify block has the following form:

```
Specify Oxs_ImageAtlas:name {
    xrange { xmin xmax }
    yrange { ymin ymax }
```

```
        zrange { zmin zmax }
        viewplane view
        image pic
        colormap {
            color-1 region_name
            color-2 region_name
            ...
            color-n region_name
        }
        matcherror max_color_distance
    }
```

The **xrange**, **yrange**, **zrange** entries specify the extent of the atlas, in meters. The **viewplane** *view* value should be one of the 3 two-letter codes `xy`, `zx` or `yz`, which specify the mapping of the horizontal and vertical axes of the image respectively to axes in the simulation. The image is scaled as necessary along each dimension to match the atlas extents along the corresponding axes. The image is overlaid through the entire depth of the perpendicular dimension, i.e., along the axis absent from the `viewplane` specification. The `Oxs_ImageAtlas` class can be used inside a `Oxs_MultiAtlas` object to specify regions in a multilayer structure.

The **image** entry specifies the name of the image file to use. If the file path is relative, then it will be taken with respect to the directory containing the MIF file. The image format may be any of those recognized by `any2ppm` (Sec. 16.1). The file will be read directly by Oxs if it is in the P3 or P6 PPM formats, otherwise `any2ppm` will be automatically launched to perform the conversion.

The **colormap** value is an even length list of color + region name pairs. The colors may be specified in any of several ways. The most explicit is to use one of the Tk numeric formats, `#rgb`, `#rrggbb`, `#rrrgggbbb` or `#rrrrggggbbbb`, where each r, g, and b is one hex digit (i.e., 0-9 or A-F) representing the red, green and blue components of the color, respectively. For example, `#F00` is bright (full-scale) red, `#800` would be a darker red, while `#FF0` and `#FFFF00` would both be bright yellow. Refer to the `Tk_GetColor` documentation for details. For shades of gray the special notation `grayD` or `greyD` is available, where D is a decimal value between 0 and 100, e.g., `grey0` is black and `grey100` is white. Alternatively, one may use any of the symbolic names defined in the `oommf/config/colors.def` file, such as `red`, `white` and `skyblue`. When comparing symbolic names, spaces and capitalization are ignored. The list of symbolic names can be extended by adding additional files to the `Color filename` option in the `options.tcl` customization file (Sec. 2.3.2). Finally, one *color* in the `colormap` list may optionally be the special keyword "default," which essentially represents all colors not in the `colormap` list.

Each of the specified colors should be distinct, but the region names are allowed to be repeated as desired. The region names may be chosen arbitrarily, except the special

keyword "universe" is reserved for points not in any of the regions. This includes all points outside the atlas bounding box defined by the `xrange`, `yrange`, `zrange` entries, but may also include points inside that boundary.

Pixels in the image are assigned to regions by comparing the color of the pixel to the list of colors specified in `colormap`. If the pixel color is closer to a `colormap` color than *max_color_distance*, then the colors are considered matched. If a pixel color matches exactly one `colormap` color, then the pixel is assigned to the corresponding region. If a pixel color matches more than one `colormap` color, the pixel is assigned to the region corresponding to the closest match. If a pixel color doesn't match any of the `colormap` colors, then it is assigned to the *default region*, which is the region paired with the "default" keyword. If `default` does not explicitly appear in the `colormap` colors list, then `universe` is made the default region.

To calculate the distance between two colors, each color is first converted to a scaled triplet of floating point red, green, and blue values, (r, g, b), where each component lies in the interval $[0, 1]$, with $(0, 0, 0)$ representing black and $(1, 1, 1)$ representing white. For example, $(0, 0, 1)$ is bright blue. Given two colors in this representation, the distance is computed using the standard Euclidean norm with uniform weights, i.e., the distance between (r_1, g_1, b_1) and (r_2, g_2, b_2) and is

$$\sqrt{(r_1 - r_2)^2 + (g_1 - g_2)^2 + (b_1 - b_2)^2}.$$

Since the difference in any one component is at most 1, the distance between any two colors is at most $\sqrt{3}$.

As explained above, two colors are considered to match if the distance between them is less than the specified **matcherror** value. If *max_color_distance* is sufficiently small, then it may easily happen that a pixel's color does not match any of the specified region colors, so the pixel would be assigned to the default region. On the other hand, if *max_color_distance* is larger than $\sqrt{3}$, then all colors will match, and no pixels will be assigned to the default region. If `matcherror` is not specified, then the default value for *max_color_distance* is 3, which means all colors match.

The following example should help clarify these matters.

```
Specify Oxs_ImageAtlas:atlas {
    xrange { 0 400e-9 }
    yrange { 0 200e-9 }
    zrange { 0  20e-9 }
    image  mypic.gif
    viewplane "xy"
    colormap {
        blue   cobalt
        red    permalloy
        green  universe
```

```
        default cobalt
    }
    matcherror .1
}
```

Blue pixels get mapped to the "cobalt" region and red pixels to the "permalloy" region. Green pixels are mapped to the "universe" non-region, which means they are considered to be outside the atlas entirely. This is a fine point, but comes into play when atlases with overlapping bounding boxes are brought together inside an `Oxs_MultiAtlas`. To which region would an orange pixel be assigned? The scaled triplet representation for orange is $(1, 0.647, 0)$, so the distance to blue is 1.191, the distance to red is 0.647, and the distance to green is 1.06. Thus the closest color is red, but 0.647 is outside the `matcherror` setting of 0.1, so orange doesn't match any of the colors and is hence assigned to the default region, which in this case is cobalt. On the other hand, if `matcherror` had been set to say 1, then orange and red would match and orange would be assigned to the permalloy region.

Pixels with colors that are equidistant to and match more than one color in the colormap will be assigned to one of the closest color regions. The user should not rely on any particular selection, that is to say, the explicit matching procedure in this case is not defined.

Oxs_MultiAtlas: This atlas is built up as an ordered list of other atlases. The set of regions defined by the `Oxs_MultiAtlas` is the union of the regions of all the atlases contained therein. The sub-atlases need not be disjoint, however each point is assigned to the region in the first sub-atlas in the list that contains it, so the regions defined by the `Oxs_MultiAtlas` are effectively disjoint.

The `Oxs_MultiAtlas` specify block has the form

```
Specify Oxs_MultiAtlas:name {
    atlas    atlas_1_spec
    atlas    atlas_2_spec
    ...
    xrange { xmin xmax }
    yrange { ymin ymax }
    zrange { zmin zmax }
}
```

Each *atlas_spec* may be either a reference to an atlas defined earlier and outside the current Specify block, or else an inline, embedded atlas definition. The bounding box **xrange**, **yrange** and **zrange** specifications are each optional. If not specified the corresponding range for the atlas bounding box is taken from the minimal bounding box containing all the sub-atlases.

If the atlases are not disjoint, then the regions as defined by an `Oxs_MultiAtlas` can be somewhat different from those of the individual component atlases. For example, suppose `regionA` is a rectangular region in `atlasA` with corner points (5,5,0) and (10,10,10), and `regionB` is a rectangular region in `atlasB` with corner points (0,0,0) and (10,10,10). When composed in the order `atlasA`, `atlasB` inside an `Oxs_MultiAtlas`, `regionA` reported by the `Oxs_MultiAtlas` will be the same as `regionA` reported by `atlasA`, but `regionB` as reported by the `Oxs_MultiAtlas` will be the "L" shaped volume of those points in `atlasB`'s `regionB` not inside `regionA`. If the `Oxs_MultiAtlas` is constructed with `atlasB` first and `atlasA` second, then `regionB` as reported by the `Oxs_MultiAtlas` would agree with that reported by `atlasB`, but `regionA` would be empty.

NOTE: The `attributes` key label (cf. Sec. 17.1.2.5) is not supported by this class.

Oxs_ScriptAtlas: An atlas where the regions are defined via a Tcl script. The specify block has the form

```
Specify Oxs_ScriptAtlas:name {
    xrange { xmin xmax }
    yrange { ymin ymax }
    zrange { zmin zmax }
    regions { rname_1 rname_2 ... rname_n }
    script_args { args_request }
    script Tcl_script
}
```

Here *xmin, xmax, ...* are coordinates in meters, specifying the extents of the axes-parallel rectangular parallelepiped enclosing the total volume being identified. This volume is subdivided into n sub-regions, using the names as given in the **regions** list. The **script** is used to assign points to the various regions. Appended to the script are the arguments requested by **script_args**, in the manner explained in the User Defined Support Procedures section (Sec. 17.1.2.6) of the MIF 2.1 file format documentation. The value *args_request* should be a subset of {`relpt rawpt minpt maxpt span`}. If `script_args` is not specified, the default value `relpt` is used. When executed, the return value from the script should be an integer in the range 1 to n, indicating the user-defined region in which the point lies, or else 0 if the point is not in any of the n regions. Region index 0 is reserved for the implicit "universe" region, which is all-encompassing. The following example may help clarify the discussion:

```
proc Octs { cellsize x y z xmin ymin zmin xmax ymax zmax } {
    global RegionArray
    set xindex [expr {int(floor(($x-$xmin)/$cellsize))}]
    set yindex [expr {int(floor(($y-$ymin)/$cellsize))}]
    set zindex [expr {int(floor(($z-$zmin)/$cellsize))}]
```

```
    set octant [expr {1+$xindex+2*$yindex+4*$zindex}]
    if {$octant<1 || $octant>8} {
       return 0
    }
    return $octant
}
Specify Oxs_ScriptAtlas:octant {
    xrange {-20e-9 20e-9}
    yrange {-20e-9 20e-9}
    zrange {-20e-9 20e-9}
    regions { VIII V VII VI IV I III II }
    script_args { rawpt minpt maxpt }
    script { Octs 20e-9 }
}
```

This atlas divides the rectangular volume between $(-20, -20, -20)$ and $(20, 20, 20)$ (nm) into eight regions, corresponding to the standard octants, I through VIII. The `Octs` Tcl procedure returns a value between 1 and 8, with 1 corresponding to octant VIII and 8 to octant II. The canonical octant ordering starts with I as the $+x, +y, +z$ space, proceeds counterclockwise in the $+z$ half-space, and concludes in the $-z$ half-space with V directly beneath I, VI beneath II, etc. The ordering computed algorithmically in `Octs` starts with 1 for the $-x, -y, -z$ space, 2 for the $+x, -y, -z$ space, 3 for the $-x, +y, -z$ space, etc. The conversion between the two systems is accomplished by the ordering of the `regions` list.

7.3.2 Meshes

Meshes define the discretization impressed on the simulation. There should be exactly one mesh declared in a MIF 2.1 file. The only standard mesh available at present is

```
Specify Oxs_RectangularMesh:name {
    cellsize { xstep ystep zstep }
    atlas atlas_spec
}
```

This creates an axes parallel rectangular mesh across the entire space covered by **atlas**. The mesh sample rates along each axis are specified by **cellsize** (in meters). The mesh is cell-based, with the center of the first cell one half step in from the minimal extremal point (xmin,ymin,ymax) for *atlas_spec*. The *name* is commonly set to "mesh", in which case the mesh object may be referred to by other `Oxs_Ext` objects by the short name `:mesh`.

7.3.3 Energies

The following subsections describe the available energy terms. In order to be included in the simulation energy and field calculations, each energy term must be declared in its own,

top-level Specify block, i.e., energy terms should not be declared inline inside other `Oxs_Ext` objects. There is no limitation on the number of energy terms that may be specified in the input MIF file. Many of these terms have spatially varying parameters that are initialized via *field_object_spec* entries (Sec. 7.3.6) in their Specify initialization block (see Sec. 17.1.2.2).

Outputs: For each magnetization configuration, two standard outputs are provided by all energy terms: the scalar output "Energy," which is the total energy in joules contributed by this energy term, and the vector field output "Field," which is the pointwise field in A/m. If the code was compiled with the macro `NDEBUG` not defined, then there will be an additional scalar output, "Calc count," which counts the number of times the term has been calculated in the current simulation. This is intended for debugging purposes only; this number should agree with the "Energy calc count" value provided by the evolver.

- **Anisotropy Energy**

  Oxs_UniaxialAnisotropy: Uniaxial magneto-crystalline anisotropy. Specify block takes 2 parameters, crystalline anisotropy constant **K1** (in J/m^3) and anisotropy direction **axis**. The axis direction is an easy axis if K1>0, or is the normal to the easy plane if $K1 < 0$. Both may be varied cellwise across the mesh. The anisotropy constant is initialized with a scalar field object, and the axis direction with an vector field object. The axis directions should be unit vectors. The energy computed by this term is non-negative in all cases.

  Oxs_CubicAnisotropy: Cubic magneto-crystalline anisotropy. Specify block takes 3 parameters, crystalline anisotropy constant **K1** (in J/m^3) and anisotropy directions **axis1** and **axis2**. The third anisotropy axis is computed as the vector product, axis1 × axis2. The axis directions are easy axes if K1>0, or hard axes if K1<0. All may be varied cellwise across the mesh. K1 is initialized with a scalar field object, and the axis directions with vector field objects. The axis directions should be unit vectors. The second axis, `axis2`, will be adjusted if necessary to be orthogonal to `axis1`. For each cell, if K1>0 then the computed energy will be non-negative, else if K1<0 then the computed energy will be non-positive.

- **Exchange Energy**

  Oxs_Exchange6Ngbr: Standard 6-neighbor exchange energy. The exchange energy density contribution from cell i is given by

$$E_i = \sum_{j \in N_i} A_{ij} \frac{\mathbf{m}_i \cdot (\mathbf{m}_i - \mathbf{m}_j)}{\Delta_{ij}^2} \tag{1}$$

  where N_i is the set consisting of the 6 cells nearest to cell i, A_{ij} is the exchange coefficient between cells i and j in J/m, and Δ_{ij} is the discretization step size between cell i and cell j (in meters).

  The Specify block for this term has the form

```
Specify Oxs_Exchange6Ngbr:name {
    default_A value
    atlas atlas_spec
    A {
        region-1 region-1 A_11
        region-1 region-2 A_12
        ...
        region-m region-n A_mn
    }
}
```

The **A** block specifies A_{ij} values on a region by region basis, where the regions are labels declared by *atlas_spec*. This allows for specification of A both inside a given region (e.g., A_{ii}) and along interfaces between regions (e.g., A_{ij}). By symmetry, if A_{ij} is specified, then the same value is automatically assigned to A_{ji} as well. The **default_A** value is applied to any otherwise unassigned A_{ij}.

Although one may specify A_{ij} for any pair of regions i and j, it is only required and only active if the region pair are in contact. If long-range exchange interaction is required, use `Oxs_TwoSurfaceExchange`.

Oxs_UniformExchange: Similar to `Oxs_Exchange6Ngbr`, except the exchange constant A is uniform across all space. The Specify block is very simple, consisting of the label **A** with the desired exchange coefficient value in J/m. Since `A` is not spatially varying, it is initialized with a simple constant (as opposed to a scalar field object).

Oxs_ExchangePtwise: The exchange coefficient A_i is specified on a point-by-point (or cell-by-cell) basis, as opposed to the pairwise specification model used by `Oxs_Exchange6Ngbr`. The exchange energy at a cell i is computed across its nearest 6 neighbors, N_i, using the formula

$$E_i = \sum_{j \in N_i} A_{ij,\text{eff}} \frac{\mathbf{m}_i \cdot (\mathbf{m}_i - \mathbf{m}_j)}{\Delta_{ij}^2}$$

where Δ_{ij} is the discretization step size from cell i to cell j in meters, and

$$A_{ij,\text{eff}} = \frac{2A_iA_j}{A_i + A_j},$$

with $A_{ij,\text{eff}} = 0$ if A_i and A_j are 0.

Note that $A_{ij,\text{eff}}$ satisfies the following properties:

$$\begin{aligned} A_{ij,\text{eff}} &= A_{ji,\text{eff}} \\ A_{ij,\text{eff}} &= A_i \qquad \text{if } A_i = A_j \\ \lim_{A_i \downarrow 0} A_{ij,\text{eff}} &= 0. \end{aligned}$$

Additionally, if A_i and A_j are non-negative,

$$\min(A_i, A_j) \leq A_{ij,\text{eff}} \leq \max(A_i, A_j).$$

Evaluating the exchange energy with this formulation of $A_{ij,\text{eff}}$ is equivalent to finding the minimum possible exchange energy between cells i and j under the assumption that A_i is constant in each of the two cells. Similar considerations are made in computing the exchange energy for a 2D variable thickness model [13].

The Specify block for `Oxs_ExchangePtwise` has the form

```
Specify Oxs_ExchangePtwise:name {
    A scalarfield_spec
}
```

where *scalarfield_spec* is an arbitrary scalar field object (Sec. 7.3.6) returning the desired exchange coefficient in J/m.

Oxs_TwoSurfaceExchange: Provides long-range bilinear and biquadratic exchange. Typically used to simulate RKKY-style coupling across non-magnetic spacers in spinvalves. The specify block has the form

```
Specify Oxs_TwoSurfaceExchange:name {
    sigma value
    sigma2 value
    surface1 {
        atlas atlas_spec
        region region_label
        scalarfield scalarfield_spec
        scalarvalue fieldvalue
        scalarside sign
    }
    surface2 {
        atlas atlas_spec
        region region_label
        scalarfield scalarfield_spec
        scalarvalue fieldvalue
        scalarside sign
    }
}
```

Here **sigma** and **sigma2** are the bilinear and biquadratic surface (interfacial) exchange energies, in J/m^2. Either is optional, with default value 0.

The **surface1** and **surface2** sub-blocks describe the two interacting surfaces. Each description consists of 5 name-values pairs, which must be listed in the order shown. In each sub-block, *atlas_spec* specifies an atlas, and *region_label* specifies a region in that atlas. These bound the extent of the desired surface. The

following **scalarfield**, **scalarvalue** and **scalarside** entries define a discretized surface inside the bounding region. Here *scalarfield_spec* references a scalar field object, *fieldvalue* should be a floating point value, and *sign* should be a single character, either '−' or '+'. If *sign* is '−', then any point for which the scalar field object takes a value less than or equal to the `scalarvalue` value is considered to be "inside" the surface. Conversely, if *sign* is '+', then any point for which the scalar field object has value greater than or equal to the `scalarvalue` value is considered to be "inside" the surface. The discretized surface determined is the set of all points on the problem mesh that are in the bounding region, are either on the surface or lie on the "inside" side of the surface, and have a neighbor that is on the "outside" side of the surface.

In this way, 2 discrete lists of cells representing the two surfaces are obtained. Each cell from the first list (representing `surface1`) is then matched with the closest cell from the second list (i.e., from `surface2`). Note the asymmetry in this matching process: each cell from the first list is included in exactly one match, but there may be cells in the second list that are included in many match pairs, or in none. If the two surfaces are of different sizes, then in practice typically the smaller will be made the first surface, because this will usually lead to fewer multiply-matched cells, but this designation is not required.

The resulting exchange energy density at cell i on one surface from matching cell j on the other is given by

$$E_{ij} = \frac{\sigma\left[1 - \mathbf{m}_i \cdot \mathbf{m}_j\right] + \sigma_2\left[1 - (\mathbf{m}_i \cdot \mathbf{m}_j)^2\right]}{\Delta_{ij}}$$

where σ and σ_2, respectively, are the bilinear and biquadratic surface exchange coefficients between the two surfaces, in $\mathrm{J/m^2}$, $\mathbf{m}_i$ and $\mathbf{m}_j$ are the normalized, unit spins (i.e., magnetization directions) at cells i and j, and Δ_{ij} is the discretization cell size in the direction from cell i towards cell j, in meters. Note that if σ is negative, then the surfaces will be anti-ferromagnetically coupled. Likewise, if σ_2 is negative, then the biquadratic term will favor orthogonal alignment.

The following example produces an antiferromagnetic exchange coupling between the lower surface of the "top" layer and the upper surface of the "bottom" layer, across a middle "spacer" layer. The simple `Oxs_LinearScalarField` object is used here to provide level surfaces that are planes orthogonal to the z-axis. In practice this example might represent a spinvalve, where the top and bottom layers would be composed of ferromagnetic material and the middle layer could be a copper spacer.

```
Specify Oxs_MultiAtlas:atlas {
    atlas { Oxs_BoxAtlas {
        name top
        xrange {0 500e-9}
```

```
        yrange {0 250e-9}
        zrange {6e-9 9e-9}
    } }
    atlas { Oxs_BoxAtlas {
        name spacer
        xrange {0 500e-9}
        yrange {0 250e-9}
        zrange {3e-9 6e-9}
    } }
    atlas { Oxs_BoxAtlas {
        name bottom
        xrange {0 500e-9}
        yrange {0 250e-9}
        zrange {0 3e-9}
    } }
}

Specify Oxs_LinearScalarField:zheight {
    vector {0 0 1}
    norm   1.0
}

Specify Oxs_TwoSurfaceExchange:AF {
    sigma -1e-4
    surface1 {
               atlas  :atlas
              region  bottom
         scalarfield  :zheight
         scalarvalue  3e-9
          scalarside  -
    }
    surface2 {
               atlas  :atlas
              region  top
         scalarfield  :zheight
         scalarvalue  6e-9
          scalarside  +
    }
}
```

Oxs_RandomSiteExchange: A randomized exchange energy. The Specify block has the form

```
Specify Oxs_RandomSiteExchange:name {
```

`linkprob` *probability*
`Amin` *A_lower_bound*
`Amax` *A_upper_bound*
}

Each adjacent pair of cells i, j, is given **linkprob** probability of having a non-zero exchange coefficient A_{ij}. Here two cells are adjacent if they lie in each other's 6-neighborhood. If a pair is found to have a non-zero exchange coefficient, then A_{ij} is drawn uniformly from the range [`Amin`, `Amax`]. The exchange energy is computed using (1), the formula used by the `Oxs_Exchange6Ngbr` energy object. The value A_{ij} for each pair of cells is determined during problem initialization, and is held fixed thereafter. The limits *A_lower_bound* and *A_upper_bound* may be any real numbers; negative values may be used to weaken the exchange interaction arising from other exchange energy terms. The only restriction is that *A_lower_bound* must not be greater than *A_upper_bound*. The `linkprob` value *probability* must lie in the range $[0, 1]$.

- **Self-Magnetostatic Energy**

  Oxs_Demag: Standard demagnetization energy term, which is built on the assumption that the magnetization is constant in each cell, and computes the average demagnetization field through the cell using formulae from [2, 12] and convolution via the Fast Fourier Transform. The Specify initialization string should be an empty string, typically denoted by {}.

  Oxs_SimpleDemag: This is the same as the `Oxs_Demag` object, except that the implementation does not use any of the of the symmetries inherent in the demagnetization kernel, or special properties of the Fourier Transform when applied to a real (non-complex) function. As a result, the source code for this implementation is considerably simpler than for `Oxs_Demag`, but the run time performance and memory usage are poorer. `Oxs_SimpleDemag` is included for validation checks, and as a base for user-defined demagnetization implementations. The Specify initialization string for `Oxs_SimpleDemag` is an the empty string, i.e., {}.

- **Zeeman Energy**

  Oxs_UZeeman: Uniform (homogeneous) applied field energy. The specify block for this term takes an optional **multiplier** entry, and a required field range list **Hrange**. The field range list should be a compound list, with each sublist consisting of 7 elements: the first 3 denote the start field for the range, the next 3 denote the end field for the range, and the last element specifies the number of (linear) steps through the range. If the step count is 0, then the range consists of the start field only. If the step count is bigger than 0, then the start field is skipped over if and only if it is the same field that ended the previous range (if any).

The fields specified in the range entry are nominally in A/m, but these values are multiplied by `multiplier`, which may be used to effectively change the units. For example,

```
Specify Oxs_UZeeman {
    multiplier 795.77472
    Hrange {
        { 0 0 0 10 0 0 2 }
        { 10 0 0 0 0 0 1 }
    }
}
```

The applied field steps between 0 mT, 5 mT, 10 mT and back to 0 mT. (Note that 795.77472=$0.001/\mu_0$.)

Oxs_FixedZeeman: Non-uniform, non-time varying applied field. This can be used to simulate a biasing field. The specify block takes one required parameter, which defines the field, and one optional parameter, which specifies a multiplication factor.

```
Specify Oxs_FixedZeeman:name {
    field vector_field_spec
    multiplier multiplier
}
```

The default value for *multiplier* is 1.

Oxs_ScriptUZeeman: Spatially uniform applied field, potentially varying as a function of time and stage, determined by a Tcl script. The Specify block has the form

```
Specify Oxs_ScriptUZeeman:name {
    script_args { args_request }
    script Tcl_script
    multiplier multiplier
}
```

Here **script** indicates the Tcl script to use. The script is called once each iteration. Appended to the script are the arguments requested by **script_args**, in the manner explained in the User Defined Support Procedures section (Sec. 17.1.2.6) of the MIF 2.1 file format documentation. The value *args_request* should be a subset of {`stage stage_time total_time`}. If `script_args` is not specified, the default argument list is the complete list in the aforementioned order. The units for the time arguments are seconds.

The return value from the script should be a 6-tuple of numbers, {`Hx`, `Hy`, `Hz`, `dHx`, `dHy`, `dHz`}, representing the applied field and the time derivative of the applied field. The field as a function of time must be differentiable for the duration of each stage. Discontinuities are permitted between stages.

The field and its time derivative are multiplied by the **multiplier** value before use. The final field value should be in A/m; if the Tcl script returns the field in T, then a `multiplier` value of $1/\mu_0$ (approx. 795774.72) should be applied to convert the Tcl result into A/m. The default value for `multiplier` is 1.

The following example produces a sinusoidally varying field of frequency 1 GHz and amplitude 800 A/m, directed along the x-axis.

```
Specify Oxs_ScriptUZeeman {
   script_args total_time
   script SineField
}

proc SineField { total_time } {
    set PI [expr {4*atan(1.)}]
    set Amp 800.0
    set Freq [expr {1e9*(2*$PI)}]
    set Hx [expr {$Amp*sin($Freq*$total_time)}]
    set dHx [expr {$Amp*$Freq*cos($Freq*$total_time)}]
    return [list $Hx 0 0 $dHx 0 0]
}
```

Oxs_TransformZeeman: Essentially a combination of the `Oxs_FixedZeeman` and `Oxs_ScriptUZeeman` classes, where an applied field is produced by applying a spatially uniform, but time and stage varying linear transform to a spatially varying but temporally static field. The transform is specified by a Tcl script.

The Specify block has the form

```
Specify Oxs_TransformZeeman:name {
    field vector_field_spec
    type transform_type
    script Tcl_script
    script_args { args_request }
    multiplier multiplier
}
```

The **field** specified by *vector_field_spec* is evaluated during problem initialization and held throughout the life of the problem. On each iteration, the specified Tcl **script** is called once. Appended to the script are the arguments requested by **script_args**, as explained in the User Defined Support Procedures section (Sec. 17.1.2.6) of the MIF 2.1 file format documentation. The value for `script_args` should be a subset of {`stage stage_time total_time`}. The default value for `script_args` is the complete list in the aforementioned order. The time arguments are specified in seconds.

The script return value should define a 3x3 linear transform and its time derivative. The transform must be differentiable with respect to time throughout each

stage, but is allowed to be discontinuous between stages. The transform is applied pointwise to the fixed field obtained from *vector_field_spec*, which is additionally scaled by *multiplier*. The **multiplier** entry is optional, with default value 1.0.

The **type** *transform_type* value declares the format of the result returned from the Tcl script. Recognized formats are `identity`, `diagonal`, `symmetric` and `general`. The most flexible is `general`, which indicates that the return from the Tcl script is a list of 18 numbers, defining a general 3x3 matrix and its 3x3 matrix of time derivatives. The matrices are specified in row-major order, i.e., $M_{1,1}$, $M_{1,2}$, $M_{1,3}$, $M_{2,1}$, $M_{2,2}$, ... Of course, this is a long list to construct; if the desired transform is symmetric or diagonal, then the `type` may be set accordingly to reduce the size of the Tcl result string. Scripts of the `symmetric` type return 12 numbers, the 6 upper diagonal entries in row-major order, i.e., $M_{1,1}$, $M_{1,2}$, $M_{1,3}$, $M_{2,2}$, $M_{2,3}$, $M_{3,3}$, for both the transformation matrix and its time derivative. Use the `diagonal` type for diagonal matrices, in which case the Tcl script result should be a list of 6 numbers.

The simplest *transform_type* is `identity`, which is the default. This identifies the transform as the identity matrix, which means that effectively no transform is applied, aside from the `multiplier` option which is still active. For the `identity` transform type, `script` and `script_args` should not be specified, and `Oxs_TransformZeeman` becomes a clone of the `Oxs_FixedZeeman` class.

The following example produces a 1000 A/m field that rotates in the xy-plane at a frequency of 1 GHz:

```
Specify Oxs_TransformZeeman {
  type general
  script {Rotate 1e9}
  field {0 1000. 0}
}
proc Rotate { freq stage stagetime totaltime } {
   global PI
   set w [expr {$freq*2*$PI}]
   set ct [expr {cos($w*$totaltime)}]
   set mct [expr {-1*$ct}]      ;# "mct" is "minus cosine (w)t"
   set st [expr {sin($w*$totaltime)}]
   set mst [expr {-1*$st}]      ;# "mst" is "minus sine (w)t"
   return [list  $ct $mst  0 \
                 $st $ct   0 \
                   0   0   1 \
                 [expr {$w*$mst}] [expr {$w*$mct}] 0 \
                 [expr {$w*$ct}]  [expr {$w*$mst}] 0 \
                        0                 0         0]
}
```

This particular effect could be obtained using the `Oxs_ScriptUZeeman` class, because the `field` is uniform. But the field was taken uniform only to simplify the example. The *vector_field_spec* may be any Oxs vector field object (Sec. 7.3.6). For example, the base field could be large in the center of the sample, and decay towards the edges. In that case, the above example would generate an applied rotating field that is concentrated in the center of the sample.

Oxs_StageZeeman: The `Oxs_StageZeeman` class provides spatially varying applied fields that are updated once per stage. In its general form, the field at each stage is provided by an Oxs vector field object (Sec. 7.3.6) determined by a user supplied Tcl script. There is also a simplified interface that accepts a list of vector field files (Sec. 19), one per stage, that are used to specify the applied field.

The Specify block takes the form

```
Specify Oxs_StageZeeman:name {
    script Tcl_script
    files { list_of_files }
    stage_count number_of_stages
    multiplier multiplier
}
```

The initialization string should specify either `script` or `files`, but not both. If a **script** is specified, then each time a new stage is started in the simulation, a Tcl command is formed by appending to *Tcl_script* the 0-based integer stage number. This command should return a reference to an `Oxs_VectorField` object, as either the instance name of an object defined via a top-level Specify block elsewhere in the MIF file, or as a two item list consisting of the name of an `Oxs_VectorField` class and an appropriate initialization string. In the latter case the `Oxs_VectorField` object will be created as a temporary object via an inlined Specify call.

The following example should help clarify the use of the `script` parameter.

```
Specify Oxs_StageZeeman {
  script SlidingFieldSpec
  stage_count 11
}

proc SlidingFieldSpec { stage } {
  set xcutoff [expr {double($stage)/10.}]
  set spec Oxs_ScriptVectorField
  lappend spec [subst {
      atlas :atlas
      script {SlidingField $xcutoff}
   }]
   return $spec
```

```
}

proc SlidingField { xcutoff xrel yrel zrel } {
   if {$xrel>$xcutoff} { return [list 0. 0. 0.] }
   return [list 2e4 0. 0.]
}
```

The `SlidingFieldSpec` proc is used to generate the initialization string for an `Oxs_ScriptVectorField` vector field object, which in turn uses the `SlidingField` proc to specify the applied field on a position-by-position basis. The resulting field will be 2×10^4 A/m in the positive x-direction at all points with relative x-coordinate larger than `$stage/10.`, and 0 otherwise. `$stage` is the stage index, which here is one of 0, 1, ..., 10. For example, if `$stage` is 5, then the left half of the sample will see a 2×10^4 A/m field directed to the right, and the right half of the sample will see none. The return value from `SlidingFieldSpec` in this case will be

```
Oxs_ScriptVectorField {
   atlas :atlas
   script {SlidingField 0.5}
}
```

The `:atlas` reference is to an `Oxs_Atlas` object defined elsewhere in the MIF file.

The **stage_count** parameter lets the `Oxs_Driver` (Sec. 7.3.5) know how many stages the `Oxs_StageZeeman` object wants. A value of 0 indicates that the object is prepared for any range of stages. Zero is the default value for `stage_count` when using the *Tcl_script* interface.

The example above made use of two scripts, one to specify the `Oxs_VectorField` object, and one used internally by the `Oxs_ScriptVectorField` object. But any `Oxs_VectorField` class may be used, as in the next example.

```
Specify Oxs_StageZeeman {
  script FileField
  stage_count 3
}

proc FileField { stage } {
  set filelist { field-a.ohf field-b.ohf field-c.ohf }
  set spec Oxs_FileVectorField
  lappend spec [subst {
      atlas :atlas
      file [lindex $filelist $stage]
   }]
   return $spec
}
```

The `FileField` proc yields a specification for an `Oxs_FileVectorField` object that loads one of three files, `field-a.ohf`, `field-b.ohf`, or `field-c.ohf`, depending on the stage number.

Specifying applied fields from a sequence of files is common enough to warrant a simplified interface. This is the purpose of the **files** parameter:

```
Specify Oxs_StageZeeman {
  files { field-a.ohf field-b.ohf field-c.ohf }
}
```

This is essentially equivalent to the preceding example, with two differences. First, `stage_count` is not needed because `Oxs_StageZeeman` knows the length of the list of files. You may specify `stage_count`, but the default value is the length of the `files` list. This is in contrast to the default value of 0 when using the `script` interface. If `stage_count` is set larger than the file list, then the last file is repeated as necessary to reach the specified size.

The second difference is that no `Oxs_Atlas` is specified when using the `files` interface. The `Oxs_FileVectorField` object spatially scales the field read from the file to match a specified volume. Typically a volume is specified by explicit reference to an atlas, but with the `files` interface to `Oxs_StageZeeman` the file fields are implicitly scaled to match the whole of the meshed simulation volume. This is the most common case; to obtain a different spatial scaling use the `script` interface as illustrated above with a different atlas or an explicit x/y/z-range specification.

The *list_of_files* value is interpreted as a *grouped list*. See the notes in Sec. 17.1.2.3 for details on grouped lists.

The remaining `Oxs_StageZeeman` parameter is **multiplier**. The value of this parameter is applied as a scale factor to the field magnitude on a point-by-point basis. For example, if the field returned by the `Oxs_VectorField` object were in Oe, instead of the required A/m, then `multiplier` could be set to 79.5775 to perform the conversion. The direction of the applied field can be reversed by supplying a negative `multiplier` value.

7.3.4 Evolvers

Evolvers are responsible for updating the magnetization configuration from one step to the next. There are two types of evolvers, *time evolvers*, which track Landau-Lifshitz-Gilbert dynamics, and *minimization evolvers*, which locate local minima in the energy surface through direct minimization techniques. Evolvers are controlled by *drivers* (Sec. 7.3.5), and must be matched with the appropriate driver type, i.e., time evolvers must be paired with time drivers, and minimization evolvers must be paired with minimization drivers. The drivers hand a magnetization configuration to the evolvers with a request to advance the configuration by one *step* (also called an *iteration*). It is the role of the drivers, not the evolvers, to

determine when a simulation stage or run is complete. Specify blocks for evolvers contain parameters to control all aspects of individual stepwise evolution, but stopping criteria are communicated in the Specify block of the driver, not the evolver.

There is currently one representative of each type of evolver in the standard OOMMF distribution, time evolver `Oxs_EulerEvolve` and minimization evolver `Oxs_CGEvolve`.

Oxs_EulerEvolve: Time evolver implementing a simple first order forward Euler method with step size control on the Landau-Lifshitz ODE [7, 9]:

$$\frac{d\mathbf{M}}{dt} = -|\bar{\gamma}|\,\mathbf{M}\times\mathbf{H}_{\text{eff}} - \frac{|\bar{\gamma}|\alpha}{M_s}\,\mathbf{M}\times(\mathbf{M}\times\mathbf{H}_{\text{eff}})\,, \tag{2}$$

where $\mathbf{M}$ is the magnetization, $\mathbf{H}_{\text{eff}}$ is the effective field, $\bar{\gamma}$ is the Landau-Lifshitz gyromagnetic ratio, and α is the damping constant. The Gilbert form

$$\frac{d\mathbf{M}}{dt} = -|\gamma|\,\mathbf{M}\times\mathbf{H}_{\text{eff}} + \frac{\alpha}{M_s}\left(\mathbf{M}\times\frac{d\mathbf{M}}{dt}\right), \tag{3}$$

where γ is the Gilbert gyromagnetic ratio, is mathematically equivalent to the Landau-Lifshitz form under the relation $\gamma = (1+\alpha^2)\,\bar{\gamma}$.

The Specify block has the form

```
Specify Oxs_EulerEvolve:name {
    alpha α
    gamma_LL γ̄
    gamma_G γ
    do_precess precess
    min_timestep minimum_stepsize
    max_timestep maximum_stepsize
    fixed_spins {
        atlas_spec
        region1 region2 ...
    }
    start_dm Δm
    error_rate rate
    absolute_step_error abs_error
    relative_step_error rel_error
    step_headroom headroom
}
```

All the entries have default values, but the ones most commonly adjusted are listed first.

The options **alpha**, **gamma_LL** and **gamma_G** are as in the Landau-Lifshitz-Gilbert ODE (2,3), where the units on $\bar{\gamma}$ and γ are m/A·s and α is dimensionless. At most one

of $\bar{\gamma}$ and γ should be specified. If neither is specified, then the default is $\gamma = 2.211 \times 10^5$. (Because of the absolute value convention adopted on $\bar{\gamma}$ and γ in (2,3), the sign given to the value of `gamma_LL` or `gamma_G` in the Specify block is irrelevant.) The default value for α is 0.5, which is large compared to experimental values, but allows simulations to converge to equilibria in a reasonable time. However, for accurate dynamic studies it is important to assign an appropriate value to α.

The **do_precess** value should be either 1 or 0, and determines whether or not the precession term in the Landau-Lifshitz ODE (i.e., the first term on the righthand side in (2)) is used. If *precess* is 0, then precession is disabled and the simulation evolves towards equilibrium along a steepest descent path. The default value is 1.

The **min_timestep** and **max_timestep** parameters provide soft limits on the size of steps taken by the evolver. The minimum value may be overridden by the driver if a smaller step is needed to meet time based stopping criteria. The maximum value will be ignored if a step of that size would produce a magnetization state numerically indistinguishable from the preceding state. The units for `min_timestep` and `max_timestep` are seconds. Default values are 0 and 10^{-10} respectively.

The optional **fixed_spins** entry allows the magnetization in selected regions of the simulation to be frozen in its initial configuration. The value portion of the entry should be a list, with the first element of the list being either an inline atlas definition (grouped as a single item), or else the name of a previously defined atlas. The remainder of the list are names of regions in that atlas for which the magnetization is to be be fixed, i.e., $\mathbf{M}(t) = \mathbf{M}(0)$ for all time t for all points in the named regions. Fields and energies are computed and reported normally across these regions. Although any atlas may be used, it is frequently convenient to set up an atlas with special regions defined expressly for this purpose.

The stepsize for the first candidate iteration in the problem run is selected so that the maximum change in the normalized (i.e., unit) magnetization $\mathbf{m}$ is the value specified by **start_dm**. The units are degrees, with default value 0.01.

The four remaining entries, **error_rate**, **absolute_step_error**, **relative_step_error**, and **step_headroom**, control fine points of stepsize selection, and are intended for advance use only. Given normalized magnetization $\mathbf{m}_i(t)$ at time t and position i, and candidate magnetization $\mathbf{m}_i(t + \Delta t)$ at time $t + \Delta t$, the error at position i is estimated to be

$$\text{Error}_i = |\dot{\mathbf{m}}_i(t + \Delta t) - \dot{\mathbf{m}}_i(t)|\, \Delta t \,/\, 2,$$

where the derivative with respect to time, $\dot{\mathbf{m}}$, is computed using the Landau-Lifshitz ODE (2). First order methods essentially assume that $\dot{\mathbf{m}}$ is constant on the interval $[t, t + \Delta t]$; the above formula uses the difference in $\dot{\mathbf{m}}$ at the endpoints of the interval to estimate (guess) how untrue that assumption is.

A candidate step is accepted if the maximum error across all positions i is smaller than `absolute_step_error`, `error_rate` $\times \Delta t$, and `relative_step_error` $\times |\dot{\mathbf{m}}_{\max}| \Delta t$,

where $|\dot{\mathbf{m}}_{\max}|$ is the maximum value of $|\dot{\mathbf{m}}_i|$ across all i at time t. If the step is rejected, then a smaller stepsize is computed that appears to pass the above tests, and a new candidate step is proposed using that smaller stepsize times `step_headroom`. Alternatively, if the step is accepted, then the error information is used to determine the stepsize for the next step, modified in the same manner by `step_headroom`.

The error calculated above is in terms of unit magnetizations, so the natural units are radians or radians/second. Inside the Specify block, however, the `error_rate` and `absolute_step_error` are specified in degrees/nanosecond and degrees, respectively; they are converted appropriately inside the code before use. The `relative_step_error` is a dimensionless quantity, representing a proportion between 0 and 1. The error check controlled by each of these three quantities may be disabled by setting the quantity value to -1. They are all optional, with default values of -1 for `error_rate`, 0.2 for `absolute_step_error`, and 0.2 for `relative_step_error`.

The `headroom` quantity should lie in the range $(0, 1)$, and controls how conservative the code will be in stepsize selection. If `headroom` is too large, then much computation time will be lost computing candidate steps that fail the error control tests. If `headroom` is small, then most candidate steps will pass the error control tests, but computation time may be wasted calculating more steps than are necessary. The default value for `headroom` is 0.85.

In addition to the above error control tests, a candidate step will also be rejected if the total energy, after adjusting for effects due to any time varying external field, is found to increase. In this case the next candidate stepsize is set to one half the rejected stepsize.

The `Oxs_EulerEvolve` module provides five scalar outputs and three vector field outputs. The scalar outputs are

- **Max dm/dt:** maximum $|d\mathbf{m}/dt|$, in degrees per nanosecond; $\mathbf{m}$ is the unit magnetization direction.
- **Total energy:** in joules.
- **Delta E:** change in energy between last step and current step, in joules.
- **dE/dt:** derivative of energy with respect to time, in joules per second.
- **Energy calc count:** number of times total energy has been calculated.

The vector field outputs are

- **Total field:** total effective field $\mathbf{H}$ in A/m.
- **mxH:** torque in A/m; $\mathbf{m}$ is the unit magnetization direction, $\mathbf{H}$ is the total effective field.
- **dm/dt:** derivative of spin $\mathbf{m}$ with respect to time, in radians per second.

Oxs_CGEvolve: The minimization evolver is `Oxs_CGEvolve`, which is an in-development conjugate gradient minimizer with no preconditioning. The Specify block has the form

```
Specify Oxs_CGEvolve:name {
    gradient_reset_count count
    minimum_bracket_step minbrack
    maximum_bracket_step maxbrack
    line_minimum_relwidth relwidth
    fixed_spins {
        atlas_spec
        region1 region2 ...
    }
}
```

All entries have default values.

The evolution to an energy minimum precedes by a sequence of line minimizations. Each line represents a one dimensional affine subspace in the $3N$ dimensional space of possible magnetization configurations, where N is the number of spins in the simulation. Once a minimum has been found along a line, a new direction is chosen that is ideally orthogonal to all preceding directions, but related to the gradient of the energy taken with respect to the magnetization. In practice the line direction sequence cannot be extended indefinitely; the parameter **gradient_reset_count** controls the maximum number of line directions selected before resetting the process. The default value for *count* is 42. Because the first line in the sequence is selected along the gradient direction, setting *count* to 1 effectively turns the algorithm into a steepest descent minimization method.

Once a minimization direction has been selected, the first stage of the line minimization is to bracket the minimum energy on that line, i.e., given a start point on the line—the location of the minimum from the previous line minimization—find another point on the line such that the energy minimum lies between those two points. As one moves along the line, the spins in the simulation rotate, with one spin rotating faster than (or at least as fast as) all the others. If the start point was not the result of a successful line minimization from the previous stage, then the first bracket attempt step is sized so that the fastest moving spin rotates through the angle specified by **minimum_bracket_step**. In the more usual case that the start point is a minimum from the previous line minimization stage, the initial bracket attempt step size is set to the distance between the current start point and the start point of the previous line minimization stage.

The energy and gradient of the energy are examined at the candidate bracket point to test if an energy minimum lies in the interval. If not, the interval is extended, based on the size of the first bracket attempt interval and the derivatives of the energy at the

interval endpoints. This process is continued until either a minimum is bracketed or the fastest moving spin rotates through the angle specified by **maximum_bracket_step**.

If the bracketing process is successful, then a one dimensional minimization is carried out in the interval, using both energy and energy derivative information. Each step in this process reduces the width of the bracketing interval. This process is continued until the width of the interval relative to the distance of the interval from the start point (i.e., the stop point from the previous line minimization process) is less than **line_minimum_relwidth**. The stop point, i.e., the effective minimum, is then taken to be the endpoint of the final interval with smallest energy. If the bracketing process is unsuccessful, i.e., the check for bracketed energy minimum failed at the maximum bracket interval size allowed by `maximum_bracket_step`, then the maximum bracket endpoint is accepted as the next point in the minimization iteration.

Once the line minimum stop point has been selected, the next iteration begins with selection of a new line direction, as described above, except in the case where the stop point was not obtained as an actual minimum, but rather by virtue of satisfying the `maximum_bracket_step` constraint. In that case the orthogonal line sequence is reset, in the same manner as when the `gradient_reset_count` switch is triggered, and the next line direction is taken directly from the energy gradient.

There are several factors to bear in mind when selecting values for the parameters `minimum_bracket_step`, `maximum_bracket_step`, and `line_minimum_relwidth`. If `minimum_bracket_step` is too small, then it may take a great many steps to obtain an interval large enough to bracket the minimum. If `minimum_bracket_step` is too large, then the bracket interval will be unnecessarily generous, and many steps may be required to locate the minimum inside the bracketing interval. However, this value only comes into play when resetting the line minimization direction sequence, so the setting is seldom critical. It is specified in degrees, with default value 0.05.

If `maximum_bracket_step` is too small, then the minima will be mostly not bracketed, and the minimization will degenerate into a type of steepest descent method. On the other hand, if `maximum_bracket_step` is too large, then the line minimizations may draw the magnetization far away from a local energy minimum (i.e., one on the full $3N$ dimensional magnetization space), eventually ending up in a different, more distant minimum. The value for `maximum_bracket_step` is specified in degrees, with default value 10.

The `line_minimum_relwidth` value determines the precision of the individual line minimizations, not the total minimization procedure, which is governed by the stopping criteria specified in the driver's Specify block. However, the `line_minimum_relwidth` value is important because the precision of the line minimizations affects the the line direction sequence orthogonality. If `line_minimum_relwidth` is too coarse, then the selected line directions will quickly drift away from mutual orthogonality. Conversely, selecting `line_minimum_relwidth` too fine will produce additional line minimization steps that do nothing to improve convergence towards the energy minimum in the full

$3N$ dimensional magnetization space. The default value for `line_minimum_relwidth` is 10^{-6}.

Referring back to the `Oxs_CGEvolve` Specify block, the **fixed_spins** entry performs the same function as for the `Oxs_EulerEvolve` class.

The `Oxs_CGEvolve` module provides seven scalar outputs and two vector field outputs. The scalar outputs are

- **Max mxHxm:** maximum $|\mathbf{m} \times \mathbf{H} \times \mathbf{m}|$, in A/m; $\mathbf{m}$ is the unit magnetization direction.
- **Total energy:** in joules.
- **Delta E:** change in energy between last step and current step, in joules.
- **Energy calc count:** number of times total energy has been calculated.
- **Bracket count:** total number of attempts required to bracket energy minimum during first phase of line minimization procedures.
- **Line min count:** total number of minimization steps during second phase of line minimization procedures (i.e., steps after minimum has been bracketed).
- **Cycle count:** number of line direction selections.

The vector field outputs are

- **H:** total effective field in A/m.
- **mxHxm:** in A/m; $\mathbf{m}$ is the unit magnetization direction.

7.3.5 Drivers

While evolvers (Sec. 7.3.4) are responsible for moving the simulation forward in individual steps, *drivers* coordinate the action of the evolver on the simulation as a whole, by grouping steps into tasks, stages and runs.

Tasks are small groups of steps that can be completed without adversely affecting user interface responsiveness. Stages are larger units specified by the MIF problem description; in particular, problem parameters are not expected to change in a discontinuous manner inside a stage. The run is the complete sequence of stages, from problem start to finish. The driver detects when stages and runs are finished, using criteria specified in the MIF problem description, and can enforce constraints, such as making sure stage boundaries respect time stopping criteria.

There are two drivers in Oxs, `Oxs_TimeDriver` for controlling time evolvers such as `Oxs_EulerEvolve`, and `Oxs_MinDriver` for controlling minimization evolvers like `Oxs_CGEvolve`.

Oxs_TimeDriver: The Oxs time driver is **Oxs_TimeDriver**. The specify block has the form

```
Specify Oxs_TimeDriver:name {
    evolver evolver_spec
    mesh mesh_spec
    Ms scalar_field_spec
    m0 vector_field_spec
    stopping_dm_dt torque_criteria
    stopping_time time_criteria
    stage_iteration_limit stage_iteration_count
    total_iteration_limit total_iteration_count
    stage_count number_of_stages
    stage_count_check test
    basename base_file_name
    scalar_output_format format
    vector_field_output_format { style precision }
}
```

The first four parameters, **evolver**, **mesh**, **Ms** and **m0** provide references to a time evolver, a mesh, a scalar field and a vector field, respectively. Here `Ms` is the pointwise saturation magnetization in A/m, and `m0` is the initial configuration for the magnetization unit spins, i.e., $|\mathbf{m}| = 1$ at each point. These four parameters are required.

The next group of 3 parameters control stage stopping criteria. The **stopping_dm_dt** value, in degrees per nanosecond, specifies that a stage should be considered complete when the maximum $|d\mathbf{m}/dt|$ across all spins drops below this value. Similarly, the **stopping_time** value specifies the maximum "Simulation time," i.e., the Landau-Lifshitz-Gilbert ODE (2,3) time, allowed per stage. For example, if *time_criteria* is 10^{-9}, then no stage will evolve for more than 1 ns. If there were a total of 5 stages in the simulation, then the total simulation time would be not more than 5 ns. The third way to terminate a stage is with a **stage_iteration_limit**. This is a limit on the number of successful evolver steps allowed per stage. A stage is considered complete when any one of these three criteria are met. Each of the criteria may be either a single value, which is applied to every stage, or else a *grouped list* (Sec. 17.1.2.3) of values. If the simulation has more stages than a criteria list has entries, then the last criteria value is applied to all additional stages. These stopping criteria all provide a default value of 0, meaning no constraint, but usually at least one is specified since otherwise there is no automatic stage termination control. For quasi-static simulations, a `stopping_dm_dt` value in the range of 1.0 to 0.01 is reasonable; the numerical precision of the energy calculations usually makes in not possible to obtain $|d\mathbf{m}/dt|$ much below 0.001 degree per nanosecond.

The **total_iteration_limit**, **stage_count** and **stage_count_check** parameters involve simulation run completion conditions. The default value for the first is 0, interpreted as no limit, but one may limit the total number of steps performed in a simulation by specifying a positive integer value here. The more usual run completion condition

is based on the stage count. If a positive integer value is specified for `stage_count`, then the run will be considered complete when the stage count reaches that value. If `stage_count` is not specified, or is given the value 0, then the effective *number_of_stages* value is computed by examining the length of the stopping criteria lists, and also any other `Oxs_Ext` object that has stage length expectations, such as `Oxs_UZeeman`. The longest of these is taken to be the stage limit value. Typically these lengths, along with `stage_count` if specified, will all be the same, and any differences indicate an error in the MIF file. Oxs will automatically test this condition, provided `stage_count_check` is set to 1, which is the default value. Stage length requests of 0 or 1 are ignored in this test, since those lengths are commonly used to represent sequences of arbitrary length. At times a short sequence is intentionally specified that is meant to be implicitly extended to match the full simulation stage length. In this case, the stage count check can be disabled by setting *test* to 0.

The value associated with **basename** is used as a prefix for output filename construction by some of the data output routines. It may represent either an absolute path (i.e., one with a leading "`/`"), or else a relative path taken with respect to the directory containing the MIF file. This an optional field with default value `oxs`. The value assigned to **scalar_output_format** should be a C-style printf string specifying the output format for DataTable output (this includes output sent to **mmDataTable** (Sec. 11), **mmGraph** (Sec. 12), and **mmArchive** (Sec. 14)). This is optional, with default value "`%.17g`". The value associated with **vector_field_output_format** should be a two element list, specifying the style and precision for vector field output sent to **mmDisp** (Sec. 13) and **mmArchive** (Sec. 14). The first element in the list should be one of `binary` or `text`, specifying the output style. If binary output is selected, then the second element specifying precision should be either 4 or 8, denoting the individual field component binary output length in bytes. For text output, the second element should be a C-style printf string like that used by `scalar_output_format`. The default value for `vector_field_output_format` is "`binary 8`".

`Oxs_TimeDriver` provides seven scalar outputs and two vector field outputs. The scalar outputs are

- **Stage:** current stage number, counting from 0.
- **Stage iteration:** number of successful evolver steps in the current stage.
- **Iteration:** number of successful evolver steps in the current simulation.
- **Simulation time:** Landau-Lifshitz-Gilbert evolution time, in seconds.
- **Mx:** magnetization component in the x direction, averaged across the entire simulation, in A/m.
- **My:** magnetization component in the y direction, averaged across the entire simulation, in A/m.
- **Mz:** magnetization component in the z direction, averaged across the entire simulation, in A/m.

The vector field outputs are

- **Magnetization:** magnetization vector **M**, in A/m.
- **Spin:** unit magnetization **m**. This output ignores the `vector_field_output_format` *precision* setting, instead always exporting at full precision.

Oxs_MinDriver: The Oxs driver for controlling minimization evolvers is **Oxs_MinDriver**. The specify block has the form

```
Specify Oxs_MinDriver:name {
    evolver evolver_spec
    mesh mesh_spec
    Ms scalar_field_spec
    m0 vector_field_spec
    stopping_mxHxm torque_criteria
    stage_iteration_limit stage_iteration_count
    total_iteration_limit total_iteration_count
    stage_count number_of_stages
    stage_count_check test
    basename base_file_name
    scalar_output_format format
    vector_field_output_format { style precision }
}
```

These parameters are the same as those described for the `Oxs_TimeDriver`, except that **stopping_mxHxm** replaces `stopping_dm_dt`, and there is no analogue to `stopping_time`. The value for `stopping_mxHxm` is in A/m, and may be a *grouped list* (Sec. 17.1.2.3). This is a required value. Choice depends on the particulars of the simulation, but typical values are in the range 10 to 0.1. Limits in the numerical precision of the energy calculations usually makes it not possible to obtain $|\mathbf{m} \times \mathbf{H} \times \mathbf{m}|$ below about 0.01 A/m.

`Oxs_MinDriver` provides six scalar outputs and two vector field outputs. The scalar outputs are

- **Stage:** current stage number, counting from 0.
- **Stage iteration:** number of successful evolver steps in the current stage.
- **Iteration:** number of successful evolver steps in the current simulation.
- **Mx:** magnetization component in the x direction, averaged across the entire simulation, in A/m.
- **My:** magnetization component in the y direction, averaged across the entire simulation, in A/m.

- **Mz:** magnetization component in the z direction, averaged across the entire simulation, in A/m.

The vector field outputs are

- **Magnetization:** magnetization vector **M**, in A/m.
- **Spin:** unit magnetization **m**. This output ignores the `vector_field_output_format` *precision* setting, instead always exporting at full precision.

7.3.6 Field Objects

Field objects return values (either scalar or vector) as a function of position. These are frequently used as embedded objects inside Specify blocks of other `Oxs_Ext` objects to initialize spatially varying quantities, such as material parameters or initial magnetization spin configurations. Units on the returned values will be dependent upon the context in which they are used.

Scalar field objects are documented first. Vector field objects are considered farther below.

Oxs_UniformScalarField: Returns the same constant value regardless of the import position. The Specify block takes one parameter, **value**, which is the returned constant value.

Oxs_AtlasScalarField: Defines values that are constant across individual regions of an `Oxs_Atlas`. The Specify block looks like

```
Specify Oxs_AtlasScalarField:value {
    atlas atlas_spec
    default_value value
    values {
        region1_label value1
        region2_label value2
        ...
    }
}
```

The specified **atlas** is used to map cell locations to regions, and the corresponding value from the **values** sub-block is assigned to that cell. The **default_value** entry is optional; if specified, and if a cell's region is not included in the `values` sub-block, then the `default_value` value is used. If `default_value` is not specified, then missing regions will raise an error.

Oxs_LinearScalarField: Returns a value that varies linearly with position. The Specify block has the form:

```
Specify Oxs_LinearScalarField:name {
    vector { v_x v_y v_z }
    norm value
}
```

If optional value **norm** is specified, then the given **vector** is first scaled to the requested size. For any given point (x, y, z), the scalar function value returned by this object will be $xv_x + yv_y + zv_z$.

Oxs_RandomScalarField: Defines a scalar field that varies spatially in a random fashion. The value at each position is drawn uniformly from the range declared by the two Specify block required parameters, **range_min** and **range_max**.

Oxs_ScriptScalarField: Produces a field dependent on a Tcl script. The Specify block has the form

```
Specify Oxs_ScriptScalarField:name {
    script Tcl_script
    script_args { args_request }
    atlas atlas_spec
    xrange { xmin xmax }
    yrange { ymin ymax }
    zrange { zmin zmax }
}
```

For each position in the mesh, the specified **script** is called with the arguments requested by **script_args** appended to the command, as explained in the User Defined Support Procedures section (Sec. 17.1.2.6) of the MIF 2.1 file format documentation. The value for `script_args` should be a subset of {`relpt rawpt minpt maxpt span`}. If `script_args` is not specified, the default value `relpt` is used.

A bounding box must also be specified, by either referencing an **atlas** specification, or by explicitly stating the range via the three entries **xrange**, **yrange**, **zrange** (in meters). The following example uses the explicit range method. See the `Oxs_ScriptVectorField` documentation below for an example using an atlas specification.

```
proc Ellipsoid { xrel yrel zrel } {
    set xrad [expr {$xrel - 0.5}]
    set yrad [expr {$yrel - 0.5}]
    set zrad [expr {$zrel - 0.5}]
    set test [expr {$xrad*$xrad+$yrad*$yrad+$zrad*$zrad}]
    if {$test>0.25} {return 0}
    return 8.6e5
}
```

```
Specify Oxs_ScriptScalarField {
    script Ellipsoid
    xrange { 0   1e-6 }
    yrange { 0 250e-9 }
    zrange { 0  50e-9 }

}
```

This `Oxs_ScriptScalarField` object returns 8.6×10^5 if the import (x,y,z) lies within the ellipsoid inscribed inside the axes parallel parallelepiped defined by (xmin=0, ymin=0, zmin=0) and (xmax=1e-6, ymax=250e-9, zmax=50e-9), and 0 otherwise. See also the discussion of the `ReadFile` MIF extension command in Sec. 17.1.1 for an example using an imported image file for similar purposes.

The available vector field objects are:

Oxs_UniformVectorField: Returns the same constant value regardless of the import position. The Specify block takes one required parameter, **vector**, which is a 3-element list of the vector to return, and one optional parameter, **norm**, which if specified adjusts the size of export vector to the specified magnitude. For example,

```
Specify Oxs_UniformVectorField {
    norm 1
    vector { 1 1 1 }
}
```

This object returns the unit vector (a, a, a), where $a = 1/\sqrt{3}$, regardless of the import position.

Oxs_AtlasVectorField: Defines vector values that are constant across individual regions of an `Oxs_Atlas`. The Specify block has the form

```
Specify Oxs_AtlasVectorField:name {
    atlas atlas_spec
    default_value { v_x v_y v_z }
    values {
        region1_label { v1_x v1_y v1_z }
        region2_label { v2_x v2_y v2_z }
        ...
    }
}
```

Interpretation is analogous to the `Oxs_AtlasScalarField` specify block, except here the output values are 3 dimensional vectors rather than scalars.

Oxs_ScriptVectorField: Conceptually similar to the `Oxs_ScriptScalarField` scalar field object, except that the script should return a vector (as a 3 element list) rather than a scalar. In addition to the **script**, **script_args**, and **xrange/yrange/zrange** or **atlas** parameters, the Specify string for `Oxs_ScriptVectorField` also accepts an optional parameter **norm**. If specified, the return values from the script are size adjusted to the specified magnitude. The following example produces a vortex-like unit vector field, with an interior core region pointing parallel to the z-axis. Here the scaling region is specified using an `atlas` reference to an object named ":atlas", which is presumed to be defined earlier in the MIF file. See the `Oxs_ScriptScalarField` sample Specify block for an example using the explicit range option.

```
proc Vortex { xrel yrel zrel } {
    set xrad [expr {$xrel-0.5}]
    set yrad [expr {$yrel-0.5}]
    set normsq [expr {$xrad*$xrad+$yrad*$yrad}]
    if {$normsq <= 0.025} {return "0 0 1"}
    return [list [expr {-1*$yrad}] $xrad 0]
}

Specify Oxs_ScriptVectorField {
    script Vortex
    norm  1
    atlas :atlas
}
```

See also the discussion of the `ReadFile` MIF extension command in Sec. 17.1.1 for an example using an imported image file for similar purposes.

Oxs_FileVectorField: Provides a file-specified vector field. Required values in the Specify block are the name of the input vector field file and the desired scaling parameters. The filename is specified via the **file** entry, which names a file containing a vector field in one of the formats recognized by **avf2ovf** (Sec. 16.3). The file will be scaled and sub-sampled as necessary to fit the *scaling region*. The scaling region is specified in the same manner as for the `Oxs_ScriptScalarField` and `Oxs_ScriptVectorField` classes, by either an explicit range specification or an atlas reference.

The magnitude of the field can be modified by the optional **norm** and **multiplier** options. If the norm parameter is given, then each vector in the field will be renormalized to the specified magnitude. If the multiplier parameter is given, then each vector in the field will be multiplied by the given scalar value. If the multiplier value is negative, the field direction will be reversed. If both `norm` and `multiplier` are given, then the field vectors are renormalized before being scaled by the multiplier value.

Oxs_RandomVectorField: Defines a vector field that varies spatially in a random fashion. The Specify block takes two required parameters, **min_norm** and **max_norm**.

The vectors produced will have magnitude between these two specified values. If `min_norm` = `max_norm`, then the samples are uniformly distributed on the sphere of radius = min_norm. Otherwise, first a uniformly distributed sample is chosen on the unit sphere, and then the magnitude is adjusted to a size drawn uniformly from the interval [min_norm,max_norm].

Oxs_PlaneRandomVectorField: Similar to `Oxs_RandomVectorField`, except that all samples are drawn from a plane rather than 3-space. In addition to **min_norm** and **max_norm**, the Specify block for `Oxs_PlaneRandomVectorField` also requires the parameter **plane_normal**. This parameter takes as its value a list of 3 elements, representing a vector orthogonal to the plane from which the random vectors are to be drawn.

7.3.7 MIF Support Classes

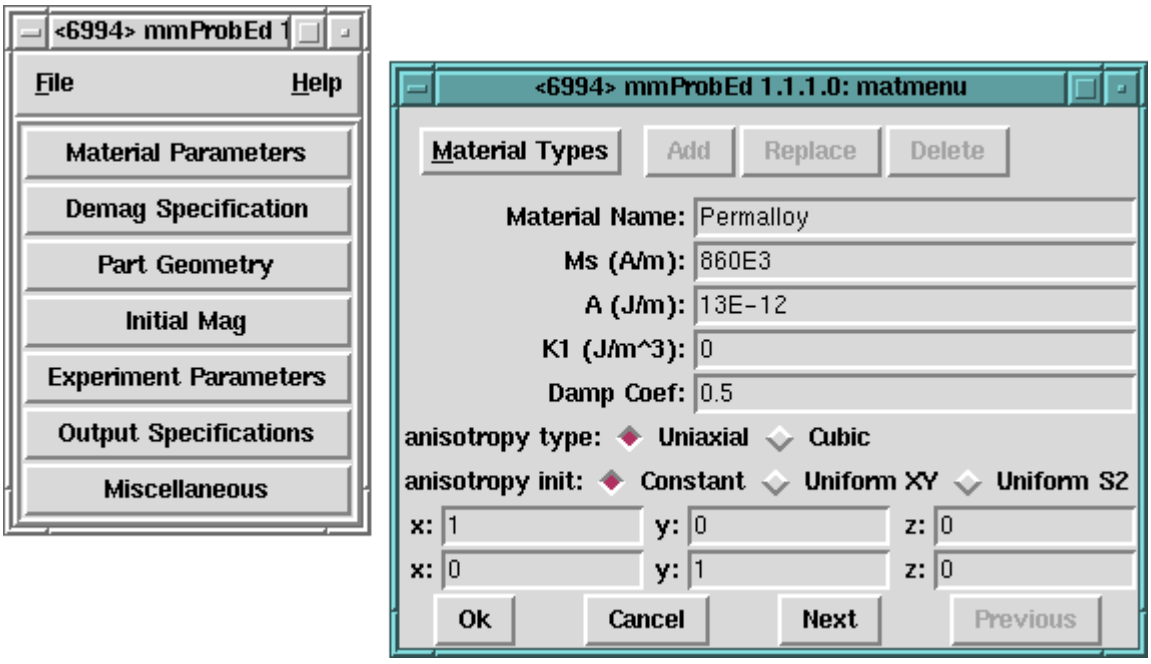
Oxs_LabelValue: A convenience object that holds label + value pairs. `Oxs_LabelValue` objects may be referenced via the standard `attributes` field in other Specify blocks, as in this example:

```
Specify Oxs_LabelValue:probdata {
  alpha 0.5
  start_dm 0.01
}
Specify Oxs_EulerEvolve {
  attributes :probdata
}
```

The Specify block string for `Oxs_LabelValue` objects is an arbitrary Tcl list with an even number of elements. The first element in each pair is interpreted as a label, the second as the value. The `attribute` option causes this list to be dropped verbatim into the surrounding object. This technique is most useful if the label + value pairs in the `Oxs_LabelValue` object are used in multiple Specify blocks, either inside the same MIF file, or across several MIF files into which the `Oxs_LabelValue` block is imported using the `ReadFile` MIF extension command.

Refer to Sec. 17.1 for details on the base MIF 2.1 format specification.

8 Micromagnetic Problem Editor: mmProbEd



Overview

The application **mmProbEd** provides a user interface for creating and editing micromagnetic problem descriptions in the *Micromagnetic Input Format* (MIF 1.1) (Sec. 17.2). **mmProbEd** also acts as a server, supplying problem descriptions to running **mmSolve2D** micromagnetic solvers.

Launching

mmProbEd may be started either by selecting the **mmProbEd** button on **mmLaunch**, or from the command line via

```
tclsh oommf.tcl mmProbEd [standard options] [-net <0|1>]
```

-net <0|1> Disable or enable a server which provides problem descriptions to other applications. By default, the server is enabled. When the server is disabled, **mmProbEd** is only useful for editing problem descriptions and saving them to files.

Inputs

The menu selection **File|Open...** displays a dialog box for selecting a file from which to load a MIF problem description. Several example files are included in the OOMMF release in the directory `oommf/app/mmpe/examples`. At startup, **mmProbEd** loads the problem

contained in `oommf/app/mmpe/init.mif` as an initial problem. Note: When loading a file, **mmProbEd** discards comments and moves records it does not understand to the bottom of its output file. Use the **FileSource** application (Sec.9) to serve unmodified problem descriptions.

Outputs

The menu selection **File**|**Save as...** displays a dialog box for selecting/entering a file in which the problem description currently held by **mmProbEd** is to be saved. Because the internal data format use by **mmProbEd** is an unordered array that does not include comments (or unrecognized records), the simple operation of reading in a MIF file and then writing it back out may alter the file.

Each instance of **mmProbEd** contains exactly one problem description at a time. When the option `-net 1` is active (the default), each also services requests from client applications (typically solvers) for the problem description it contains.

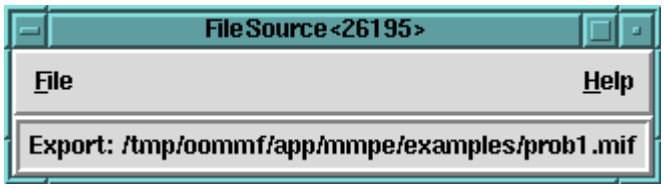
Controls

The main panel in the **mmProbEd** window contains buttons corresponding to the sections in a MIF 1.1 problem description. Selecting a button brings up another window through which the contents of that section of a problem description may be edited. The MIF sections and the elements they contain are described in detail in the MIF 1.1 (Sec. 17.2) documentation. Only one editing window is displayed at a time. The windows may be navigated in order using their **Next** or **Previous** buttons.

PLEASE NOTE: The material parameter values provided for the symbolic material types of **Iron**, **Nickel**, etc., should *not* be taken as standard reference values for these materials. These values are only approximate. They are included for convenience, and as examples for users who wish to supply their own material types with symbolic names. To introduce additional material types, edit the file `oommf/app/mmpe/materials`, appending your new entries in the same format as the example materials.

The menu selection **File**|**Exit** terminates the **mmProbEd** application. The menu **Help** provides the usual help facilities.

9 Micromagnetic Problem File Source: FileSource



Overview

The application **FileSource** provides the same service as **mmProbEd** (Sec. 8), supplying MIF 1.1 problem descriptions to running **mmSolve2D** micromagnetic solvers. As the MIF specification evolves, **mmProbEd** may lag behind. There may be new fields in the MIF specification that **mmProbEd** is not capable of editing, or which **mmProbEd** may not pass on to solvers after loading them in from a file. To make use of such fields, a MIF file may need to be edited "by hand" using a general purpose text editor. **FileSource** may then be used to supply the MIF problem description contained in a file to a solver without danger of corrupting its contents.

Launching

FileSource must be launched from the command line. You may specify on the command line the MIF problem description file it should serve to client applications. The command line is

```
tclsh oommf.tcl FileSource [standard options] [filename]
```

Although **FileSource** does not appear on the list of **Programs** that **mmLaunch** offers to launch, running copies do appear on the list of **Threads** since they do provide a service registered with the account service directory.

Inputs

FileSource takes its MIF problem description from the file named on the command line, or from a file selected through the **File|Open** dialog box. No checking of the file contents against the MIF specification is performed. The file contents are passed uncritically to any client application requesting a problem description. Those client applications should raise errors when presented with invalid problem descriptions.

Outputs

Each instance of **FileSource** provides the contents of exactly one file at a time. The file name is displayed in the **FileSource** window to help the user associate each instance of

FileSource with the data file it provides. Each instance of **FileSource** accepts and services requests from client applications (typically solvers) for the contents of the file it exports.

The contents of the file are read at the time of the client request, so if the contents of a file change between the time of the **FileSource** file selection and the arrival of a request from a client, the new contents will be served to the client application.

Controls

The menu selection **File|Exit** terminates the **FileSource** application. The **Help** menu provides the usual help facilities.

10 The 2D Micromagnetic Solver

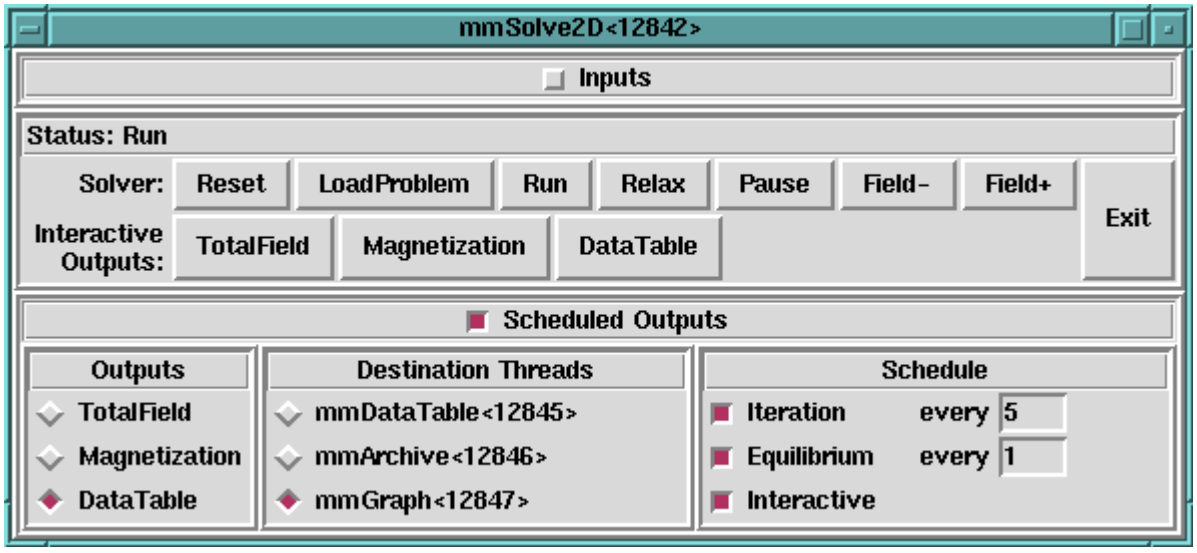
The OOMMF 2D micromagnetic computation engine, mmSolve, is capable of solving problems defined on a two-dimensional grid of square cells with three-dimensional spins. This solver is older, less flexible and less extensible than the Oxs (Sec. 7) solver. Users are encouraged to migrate to Oxs where possible.

There are two interfaces provided to mmSolve, the interactive **mmSolve2D** (Sec. 10.1) interface and the command line driven **batchsolve** (Sec. 10.2.1) interface which can be used in conjunction with the OOMMF Batch System (Sec. 10.2).

Problem definition for mmSolve is accomplished using input files in the MIF 1.1 format (Sec. 17.2). Please note that this format is incompatible with the newer MIF 2.x format used by the Oxs solver. However, the command line utility **mifconvert** (Sec. 16.7) can be used to aid conversion from the MIF 1.1 format to MIF 2.1.

Note on Tk dependence: If a problem is loaded that uses a bitmap mask file (Sec. 17.2.3), and if that mask file is not in the PPM P3 (text) format, then **mmSolve2D** will launch **any2ppm** (Sec. 16.1) to convert it into the PPM P3 format. Since **any2ppm** requires Tk, at the time the mask file is read a valid display must be available. See the **any2ppm** documentation for details.

10.1 The 2D Micromagnetic Interactive Solver: mmSolve2D



Overview

The application **mmSolve2D** is a micromagnetic computation engine capable of solving problems defined on two-dimensional square grids of three-dimensional spins. Within the OOMMF architecture (see Sec. 4), **mmSolve2D** is both a server and a client application. **mmSolve2D** is a client of problem description server applications, data table display and

storage applications, and vector field display and storage applications. **mmSolve2D** is the server of a solver control service for which the only client is **mmLaunch** (Sec. 6). It is through this service that **mmLaunch** provides a user interface window (shown above) on behalf of **mmSolve2D**.

Launching

mmSolve2D may be started either by selecting the **mmSolve2D** button on mmLaunch, or from the command line via

```
tclsh oommf.tcl mmSolve2D [standard options] [-restart <0|1>]
```

-restart <0|1> Affects the behavior of the solver when a new problem is loaded. Default value is 0. When launched with `-restart 1`, the solver will look for *`basename`*`.log` and *`basename*`*`.omf` files to restart a previous run from the last saved state (where *`basename`* is the "Base Output Filename" specified in the input MIF 1.1 problem specification file (Sec. 17.2)). If these files cannot be found, then a warning is issued and the solver falls back to the default behavior (`-restart 0`) of starting the problem from scratch. The specified `-restart` setting holds for **all** problems fed to the solver, not just the first. (There is currently no interactive way to change the value of this switch.)

Since **mmSolve2D** does not present any user interface window of its own, it depends on **mmLaunch** to provide an interface on its behalf. The entry for an instance of **mmSolve2D** in the **Threads** column of any running copy of **mmLaunch** has a checkbutton next to it. This button toggles the presence of a user interface window through which the user may control that instance of **mmSolve2D**. The user interface window is divided into panels, providing user interfaces to the **Inputs**, **Outputs**, and **Controls** of **mmSolve2D**.

Inputs

The top panel of the user interface window may be opened and closed by toggling the **Inputs** checkbutton. When open, the **Inputs** panel reveals two subpanels. The left subpanel contains a list of the inputs required by **mmSolve2D**. There is only one item in the list: **ProblemDescription**. When **ProblemDescription** is selected, the right subpanel (labeled **Source Threads**) displays a list of applications that can supply a problem description. The user selects from among the listed applications the one from which **mmSolve2D** should request a problem description.

Outputs

When **mmSolve2D** has outputs available to be controlled, a **Scheduled Outputs** checkbutton appears in the user interface window. Toggling the **Scheduled Outputs** checkbutton causes a bottom panel to open and close in the user interface window. When open, the

Scheduled Outputs panel contains three subpanels. The **Outputs** subpanel is filled with a list of the types of output **mmSolve2D** can generate while solving the loaded problem. The three elements in this list are **TotalField**, for the output of a vector field representing the total effective field, **Magnetization**, for the output of a vector field representing the current magnetization state of the grid of spins, and **DataTable**, for the output of a table of data values describing other quantities of interest calculated by **mmSolve2D**.

Upon selecting one of the output types from the **Outputs** subpanel, a list of applications appears in the **Destination Threads** subpanel which provide a display and/or storage service for the type of output selected. The user may select from this list those applications to which the selected type of output should be sent.

For each application selected, a final interface is displayed in the **Schedule** subpanel. Through this interface the user may set the schedule according to which the selected type of data is sent to the selected application for display or storage. The schedule is described relative to events in **mmSolve2D**. An **Iteration** event occurs at every step in the solution of the ODE. A **ControlPoint** event occurs whenever the solver determines that a control point specification is met. (Control point specs are discussed in the Experiment parameters paragraph in the MIF 1.1 documentation (Sec. 17.2), and are triggered by solver equilibrium, simulation time, and iteration count conditions.) An **Interactive** event occurs for a particular output type whenever the corresponding "Interactive Outputs" button is clicked in the **Runtime Control** panel. The **Interactive** schedule gives the user the ability to interactively force data to be delivered to selected display and storage applications. For the **Iteration** and **ControlPoint** events, the granularity of the output delivery schedule is under user control. For example, the user may elect to send vector field data describing the current magnetization state to an **mmDisp** instance for display every 25 iterations of the ODE, rather than every iteration.

The quantities included in **DataTable** output produced by **mmSolve2D** include:

- **Iteration:** The iteration count of the ODE solver.
- **Field Updates:** The number of times the ODE solver has calculated the effective field.
- **Sim Time (ns):** The elapsed simulated time.
- **Time Step (ns):** The interval of simulated time spanned by the last step taken in the ODE solver.
- **Step Size:** The magnitude of the last step taken by the ODE solver as a normalized value. (This is currently the time step in seconds, multiplied by the gyromagnetic ratio times the damping coefficient times M_s.)
- **Bx, By, Bz (mT):** The x, y, and z components of the nominal applied field (see Sec. 17.2.5).
- **B (mT):** The magnitude of the nominal applied field (always non-negative).

- **|m x h|:** The maximum of the point-wise quantity $\|\mathbf{M} \times \mathbf{H}_{\text{eff}}\|/M_s^2$ over all the spins. This "torque" value is used to test convergence to an equilibrium state (and raise control point –torque events).

- **Mx/Ms, My/Ms, Mz/Ms:** The x, y, and z components of the average magnetization of the magnetically active elements of the simulated part.

- **Total Energy ($\mathbf{J/m^3}$):** The total average energy density for the magnetically active elements of the simulated part.

- **Exchange Energy ($\mathbf{J/m^3}$):** The component of the average energy density for the magnetically active elements of the simulated part due to exchange interactions.

- **Anisotropy Energy ($\mathbf{J/m^3}$):** The component of the average energy density for the magnetically active elements of the simulated part due to crystalline and surface anisotropies.

- **Demag Energy ($\mathbf{J/m^3}$):** The component of the average energy density for the magnetically active elements of the simulated part due to self-demagnetizing fields.

- **Zeeman Energy ($\mathbf{J/m^3}$):** The component of average energy density for the magnetically active elements of the simulated part due to interaction with the applied field.

- **Max Angle:** The maximum angle (in degrees) between the magnetization orientation of any pair of neighboring spins in the grid. (The neighborhood of a spin is the same as that defined by the exchange energy calculation.)

In addition, the solver automatically keeps a log file that records the input problem specification and miscellaneous runtime information. The name of this log file is *`basename`*`.log`, where *`basename`* is the "Base Output Filename" specified in the input problem specification. If this file already exists, then new entries are appended to the end of the file.

Controls

The middle section of the user interface window contains a series of buttons providing user control over the solver. After a problem description server application has been selected, the **LoadProblem** button triggers a fetch of a problem description from the selected server. The **LoadProblem** button may be selected at any time to (re-)load a problem description from the currently selected server. After loading a new problem the solver goes automatically into a paused state. (If no problem description server is selected when the **LoadProblem** button is invoked, nothing will happen.) The **Reset** button operates similarly, except that the current problem specifications are used.

Once a problem is loaded, the solver can be put into any of three states: run, relax and pause. Selecting **Relax** puts the solver into the "relax" state, where it runs until a

control point is reached, after which the solver pauses. If the **Relax** button is reselected after reaching a control point, then the solver will simply re-pause immediately. The **Field+** or **Field–** button must be invoked to change the applied field state. (Field state schedules are discussed below.) The **Run** selection differs in that when a control point is reached, the solver automatically steps the nominal applied field to the next value, and continues. In "run" mode the solver will continue to process until there are no more applied field states in the problem description. At any time the **Pause** button may be selected to pause the solver. The solver will stay in this state until the user reselects either **Run** or **Relax**. The current state of the solver is indicated in the **Status** line in the center panel of the user interface window.

The problem description (in MIF 1.1 format) specifies a fixed applied field schedule (see Sec. 17.2.5). This schedule defines an ordered list of applied fields, which the solver in "run" mode steps through in sequence. The **Field–** and **Field+** buttons allow the user to interactively adjust the applied field sequence. Each click on the **Field+** button advances forward one step through the specified schedule, while **Field–** reverses that process. In general, the step direction is *not* related to the magnitude of the applied field. Also note that hitting these buttons does not generate a "ControlPoint" event. In particular, if you are manually accelerating the progress of the solver through a hysteresis loop, and want to send non-ControlPoint data to a display or archive widget before advancing the field, then you must use the appropriate "Interactive Output" button.

The second row of buttons in the interaction control panel, **TotalField**, **Magnetization** and **DataTable**, allow the user to view the current state of the solver at any time. These buttons cause the solver to send out data of the corresponding type to all applications for which the "Interactive" schedule button for that data type has been selected, as discussed in the Outputs section above.

At the far right of the solver controls is the **Exit** button, which terminates **mmSolve2D**. Simply closing the user interface window does not terminate **mmSolve2D**, but only closes the user interface window. To kill the solver the **Exit** button must be pressed.

Details

Given a problem description, **mmSolve2D** integrates the Landau-Lifshitz equation [7, 9]

$$\frac{d\mathbf{M}}{dt} = -|\bar{\gamma}|\,\mathbf{M}\times\mathbf{H}_{\text{eff}} - \frac{|\bar{\gamma}|\alpha}{M_s}\,\mathbf{M}\times(\mathbf{M}\times\mathbf{H}_{\text{eff}})\,, \tag{4}$$

where

$\mathbf{M}$	is the pointwise magnetization (A/m),
$\mathbf{H}_{\text{eff}}$	is the pointwise effective field (A/m),
$\bar{\gamma}$	is the Landau-Lifshitz gyromagnetic ratio (m/(A·s)),
α	is the damping coefficient (dimensionless).

(Compare to (2), page 54.)

The effective field is defined as

$$\mathbf{H}_{\text{eff}} = -\mu_0^{-1} \frac{\partial E}{\partial \mathbf{M}}.$$

The average energy density E is a function of $\mathbf{M}$ specified by Brown's equations [4], including anisotropy, exchange, self-magnetostatic (demagnetization) and applied field (Zeeman) terms.

The micromagnetic problem is impressed upon a regular 2D grid of squares, with 3D magnetization spins positioned at the centers of the cells. Note that the constraint that the grid be composed of square elements takes priority over the requested size of the grid. The actual size of the grid used in the computation will be the nearest integral multiple of the grid's cell size to the requested size. It is important when comparing the results from grids with different cell sizes to account for the possible change in size of the overall grid.

The anisotropy and applied field energy terms are calculated assuming constant magnetization in each cell. The exchange energy is calculated using the eight-neighbor bilinear interpolation described in [5], with Neumann boundary conditions. The more common four-neighbor scheme is available as a compile-time option. Details can be found in the source-code file `oommf/app/mmsolve/magelt.cc`.

The self-magnetostatic field is calculated as the convolution of the magnetization against a kernel that describes the cell to cell magnetostatic interaction. The convolution is evaluated using fast Fourier transform (FFT) techniques. Several kernels are supported; these are selected as part of the problem description in MIF format; for details see Sec. 17.2.2: Demag specification. Each kernel represents a different interpretation of the discrete magnetization. The recommended model is `ConstMag`, which assumes the magnetization is constant in each cell, and computes the average demagnetization field through the cell using formulae from [12] and [2].

The Landau-Lifshitz ODE (4) is integrated using a second order predictor-corrector technique of the Adams type. The right side of (4) at the current and previous step is extrapolated forward in a linear fashion, and is integrated across the new time interval to obtain a quadratic prediction for $\mathbf{M}$ at the next time step. At each stage the spins are renormalized to M_s before evaluating the energy and effective fields. The right side of (4) is evaluated at the predicted $\mathbf{M}$, which is then combined with the value at the current step to produce a linear interpolation of $d\mathbf{M}/dt$ across the new interval. This is then integrated to obtain the final estimate of $\mathbf{M}$ at the new step. The local (one step) error of this procedure should be $O(\Delta t^3)$.

The step is accepted if the total energy of the system decreases, and the maximum error between the predicted and final $\mathbf{M}$ is smaller than a nominal value. If the step is rejected, then the step size is reduced and the integration procedure is repeated. If the step is accepted, then the error between the predicted and final $\mathbf{M}$ is used to adjust the size of the next step. No fixed ratio between the previous and current time step is assumed.

A fourth order Runge-Kutta solver is used to prime the predictor-corrector solver, and is used as a backup in case the predictor-corrector fails to find a valid step. The Runge-Kutta

solver is not selectable as the primary solver at runtime, but may be so selected at compile time by defining the `RUNGE_KUTTA_ODE` macro. See the file `oommf/app/mmsolve/grid.cc` for all details of the integration procedure.

For a given applied field, the integration continues until a control point (cf. Experiment parameters, Sec. 17.2.5) is reached. A control point event may be raised by the ODE iteration count, elapsed simulation time, or by the maximum value of $\|\mathbf{M}\times\mathbf{H}_{\text{eff}}\|/M_s^2$ dropping below a specified control point –torque value (implying an equilibrium state has been reached).

Depending on the problem size, **mmSolve2D** can require a good deal of working memory. The exact amount depends on a number of factors, but a reasonable estimate is 5 MB + 1500 bytes per cell. For example, a 1 μm $\times$ 1 μm part discretized with 5 nm cells will require approximately 62 MB.

Known Bugs

mmSolve2D requires the damping coefficient to be non-zero. See the MIF 1.1 documentation (Sec. 17.2) for details on specifying the damping coefficient.

When multiple copies of **mmLaunch** are used, each can have its own interface to a running copy of **mmSolve2D**. When the interface presented by one copy of **mmLaunch** is used to set the output schedule in **mmSolve2D**, those settings are not reflected in the interfaces presented by other copies of **mmLaunch**. For example, although the first interface sets a schedule that DataTable data is to be sent to an instance of **mmGraph** every third Iteration, there is no indication of that schedule presented to the user in the second interface window. It is unusual to have more than one copy of **mmLaunch** running simultaneously. However, this bug also appears when one copy of **mmLaunch** is used to load a problem and start a solver, and later a second copy of **mmLaunch** is used to monitor the status of that running solver.

A bug in the network traffic handling code of Tcl on Windows 9X systems can sometimes interfere with communications between the control interface of **mmSolve2D** and the actual computation engine. If **mmSolve2D** is sending out data to two or more data display services every iteration, the resulting network traffic can "crowd out" the receipt of control messages from the control interface. You may observe this as a long delay between the time you click the **Pause** button and the time the solver stops iterating. This bug first appeared in Tcl release 8.0.3, and remained through Tcl release 8.1.1. It is fixed in Tcl releases 8.2 and later, which we recommend for OOMMF users on Windows 9X systems. Other platforms do not have this problem.

10.2 OOMMF 2D Micromagnetic Solver Batch System

The OOMMF Batch System (OBS) provides a scriptable interface to the same micromagnetic solver engine used by **mmSolve2D** (Sec. 10.1), in the form of three Tcl applicatons (**batchmaster**, **batchslave**, and **batchsolve**) that provide support for complex job scheduling. All OBS script files are in the OOMMF distribution directory `app/mmsolve/scripts`.

Unlike much of the OOMMF package, the OBS is meant to be driven primarily from the command line or shell (batch) script. OBS applications are launched from the command line using the bootstrap application (Sec. 5).

10.2.1 2D Micromagnetic Solver Batch Interface: batchsolve

Overview

The application **batchsolve** provides a simple command line interface to the OOMMF 2D micromagnetic solver engine.

Launching

The application **batchsolve** is launched by the command line:

```
tclsh oommf.tcl batchsolve [standard options]
   [-end_exit <0|1>] [-end_paused] [-interface <0|1>] \
   [-restart <0|1>] [-start_paused] [file]
```

where

-end_exit <0|1> Whether or not to explicitly call exit at bottom of `batchsolve.tcl`. When launched from the command line, the default is to exit after solving the problem in `file`. When sourced into another script, like `batchslave.tcl`, the default is to wait for the caller script to provide further instructions.

-interface <0|1> Whether to register with the account service directory application, so that **mmLaunch** (Sec. 6), can provide an interactive interface. Default = 1 (do register), which will automatically start account service directory and host service directory applications as necessary.

-start_paused Pause solver after loading problem.

-end_paused Pause solver and enter event loop at bottom of `batchsolve.tcl` rather than just falling off the end (the effect of which will depend on whether or not Tk is loaded).

-restart <0|1> Determines solver behavior when a new problem is loaded. If 1, then the solver will look for *basename*`.log` and *basename*`*.omf` files to restart a previous run from the last saved state (where *basename* is the "Base Output Filename" specified in the input problem specification). If these files cannot be found, then a warning is issued and the solver falls back to the default behavior (equivalent to `-restart 0`) of starting the problem from scratch. The specified `-restart` setting holds for **all** problems fed to the solver, not just the first.

file Immediately load and run the specified MIF 1.1 file.

The input file `file` should contain a Micromagnetic Input Format 1.1 (Sec. 17.2) problem description, such as produced by **mmProbEd** (Sec. 8). The batch solver searches several directories for this file, including the current working directory, the `data` and `scripts` subdirectories, and parallel directories relative to the directories `app/mmsolve` and `app/mmpe` in the OOMMF distribution. Refer to the `mif_path` variable in `batchsolve.tcl` for the complete list.

If `-interface` is set to 1 (enabled), **batchsolve** registers with the account service directory application, and **mmLaunch** will be able to provide an interactive interface. Using this interface, **batchsolve** may be controlled in a manner similar to **mmSolve2D** (Sec. 10.1). The interface allows you to pause, un-pause, and terminate the current simulation, as well as to attach data display applications to monitor the solver's progress. If more interactive control is needed, **mmSolve2D** should be used.

If `-interface` is 0 (disabled), **batchsolve** does not register, leaving it without an interface, unless it is sourced into another script (e.g., `batchslave.tcl`) that arranges for an interface on the behalf of **batchsolve**.

Use the `-start_paused` switch to monitor the progress of **batchsolve** from the very start of a simulation. With this switch the solver will be paused immediately after loading the specified MIF file, so you can bring up the interactive interface and connect display applications before the simulation begins. Start the simulation by selecting the **Run** command from the interactive interface. This option cannot be used if `-interface` is disabled.

The `-end_paused` switch insures that the solver does not automatically terminate after completing the specified simulation. This is not generally useful, but may find application when **batchsolve** is called from inside a Tcl-only wrapper script.

Note on Tk dependence: If a problem is loaded that uses a bitmap mask file (Sec. 17.2.3), and if that mask file is not in the PPM P3 (text) format, then **batchsolve** will launch **any2ppm** (Sec. 16.1) to convert it into the PPM P3 format. Since **any2ppm** requires Tk, at the time the mask file is read a valid display must be available. See the **any2ppm** documentation for details.

Output

The output may be changed by a Tcl wrapper script (see Sec. 10.2.1), but the default output behavior of **batchsolve** is to write tabular text data and the magnetization state at the control point for each applied field step. The tabular data are appended to the file *basename*`.odt`, where *basename* is the "Base Output Filename" specified in the input MIF 1.1 file. See the routine `GetTextData` in `batchsolve.tcl` for details, but at present the output consists of the solver iteration count, nominal applied field **B**, reduced average magnetization **m**, and total energy. This output is in the ODT file format.

The magnetization data are written to a series of OVF (OOMMF Vector Field) files, *basename*`.field`*nnnn*`.omf`, where *nnnn* starts at `0000` and is incremented at each applied field step. (The ASCII text header inside each file records the nominal applied field at that step.) These files are viewable using **mmDisp** (Sec. 13).

The solver also automatically appends the input problem specification and miscellaneous runtime information to the log file *basename*`.log`.

Programmer's interface

In addition to directly launching **batchsolve** from the command line, `batchsolve.tcl` may also be sourced into another Tcl script that provides additional control structures. Within the scheduling system of OBS, `batchsolve.tcl` is sourced into **batchslave**, which provides additional control structures that support scheduling control by **batchmaster**. There are several variables and routines inside `batchsolve.tcl` that may be accessed and redefined from such a wrapper script to provide enhanced functionality.

Global variables

mif A Tcl handle to a global `mms_mif` object holding the problem description defined by the input MIF 1.1 file.

solver A Tcl handle to the `mms_solver` object.

search_path Directory search path used by the `FindFile` proc (see below).

Refer to the source code and sample scripts for details on manipulation of these variables.

Batchsolve procs

The following Tcl procedures are designed for external use and/or redefinition:

SolverTaskInit Called at the start of each task.

BatchTaskIterationCallback Called after each iteration in the simulation.

BatchTaskRelaxCallback Called at each control point reached in the simulation.

SolverTaskCleanup Called at the conclusion of each task.

FindFile Searches the directories specified by the global variable `search_path` for a specified file. The default `SolverTaskInit` proc uses this routine to locate the requested input MIF file.

`SolverTaskInit` and `SolverTaskCleanup` accept an arbitrary argument list (`args`), which is copied over from the `args` argument to the `BatchTaskRun` and `BatchTaskLaunch` procs in `batchsolve.tcl`. Typically one copies the default procs (as needed) into a task script, and makes appropriate modifications. You may (re-)define these procs either before or after sourcing `batchsolve.tcl`. See Sec. 10.2.2.4 for example scripts.

10.2.2 2D Micromagnetic Solver Batch Scheduling System

Overview

The OBS supports complex scheduling of multiple batch jobs with two applications, **batchmaster** and **batchslave**. The user launches **batchmaster** and provides it with a task script. The task script is a Tcl script that describes the set of tasks for **batchmaster** to accomplish. The work is actually done by instances of **batchslave** that are launched by **batchmaster**. The task script may be modeled after the included `simpletask.tcl` or `multitask.tcl` sample scripts (Sec. 10.2.2.4).

The OBS has been designed to control multiple sequential and concurrent micromagnetic simulations, but **batchmaster** and **batchslave** are completely general and may be used to schedule other types of jobs as well.

10.2.2.1 Master Scheduling Control: batchmaster The application **batchmaster** is launched by the command line:

```
tclsh oommf.tcl batchmaster [standard options] task_script \
      [host [port]]
```

task_script is the user defined task (job) definition Tcl script,

host specifies the network address for the master to use (default is *localhost*),

port is the port address for the master (default is *0*, which selects an arbitrary open port).

When **batchmaster** is run, it sources the task script. Tcl commands in the task script should modify the global object `$TaskInfo` to inform the master what tasks to perform and optionally how to launch slaves to perform those tasks. The easiest way to create a task script is to modify one of the example scripts in Sec. 10.2.2.4. More detailed instructions are in Sec. 10.2.2.3.

After sourcing the task script, **batchmaster** launches all the specified slaves, initializes each with a slave initialization script, and then feeds tasks sequentially from the task list to the slaves. When a slave completes a task it reports back to the master and is given the next unclaimed task. If there are no more tasks, the slave is shut down. When all the tasks are complete, the master prints a summary of the tasks and exits.

When the task script requests the launching and controlling of jobs off the local machine, with slaves running on remote machines, then the command line argument `host` **must** be set to the local machine's network name, and the `$TaskInfo` methods `AppendSlave` and `ModifyHostList` will need to be called from inside the task script. Furthermore, OOMMF does not currently supply any methods for launching jobs on remote machines, so a task script which requests the launching of jobs on remote machines requires a working `rsh` command or equivalent. See Sec. 10.2.2.3 for details.

10.2.2.2 Task Control: batchslave The application **batchslave** may be launched by the command line:

```
tclsh oommf.tcl batchslave [standard options] \
   host port id password [auxscript [arg ...]]
```

host, port Host and port at which to contact the master to serve.

id, password ID and password to send to the master for identification.

auxscript arg ... The name of an optional script to source (which actually performs the task the slave is assigned), and any arguments it needs.

In normal operation, the user does not launch **batchslave**. Instead, instances of **batchslave** are launched by **batchmaster** as instructed by a task script. Although **batchmaster** may launch any slaves requested by its task script, by default it launches instances of **batchslave**.

The function of **batchslave** is to make a connection to a master program, source the `auxscript` and pass it the list of arguments `aux_arg ...`. Then it receives commands from the master, and evaluates them, making use of the facilities provided by `auxscript`. Each command is typically a long-running one, such as solving a complete micromagnetic problem. When each command is complete, the **batchslave** reports back to its master program, asking for the next command. When the master program has no more commands **batchslave** terminates.

Inside **batchmaster**, each instance of **batchslave** is launched by evaluating a Tcl command. This command is called the spawn command, and it may be redefined by the task script in order to completely control which slave applications are launched and how they are launched. When **batchslave** is to be launched, the spawn command might be:

```
exec tclsh oommf.tcl batchslave -tk 0 -- $server(host) $server(port) \
   $slaveid $passwd batchsolve.tcl -restart 1 &
```

The Tcl command `exec` is used to launch subprocesses. When the last argument to `exec` is `&`, the subprocess runs in the background. The rest of the spawn command should look familiar as the command line syntax for launching **batchslave**.

The example spawn command above cannot be completely provided by the task script, however, because parts of it are only known by **batchmaster**. Because of this, the task script should define the spawn command using "percent variables" which are substituted by **batchmaster**. Continuing the example, the task script provides the spawn command:

```
exec %tclsh %oommf batchslave -tk 0 %connect_info \
   batchsolve.tcl -restart 1
```

batchmaster replaces `%tclsh` with the path to `tclsh`, and `%oommf` with the path to the OOMMF bootstrap application. It also replaces `%connect_info` with the five arguments from `--` through `$password` that provide **batchslave** the hostname and port where **batchmaster** is waiting for it to report to, and the ID and password it should pass back. In this example, the task script instructs **batchslave** to source the file `batchsolve.tcl` and pass it the arguments `-restart 1`. Finally, **batchmaster** always appends the argument `&` to the spawn command so that all slave applications are launched in the background.

The communication protocol between **batchmaster** and **batchslave** is evolving and is not described here. Check the source code for the latest details.

10.2.2.3 Batch Task Scripts The application **batchmaster** creates an instance of a `BatchTaskObj` object with the name `$TaskInfo`. The task script uses method calls to this object to set up tasks to be performed. The only required call is to the `AppendTask` method, e.g.,

```
$TaskInfo AppendTask A "BatchTaskRun taskA.mif"
```

This method expects two arguments, a label for the task (here "A") and a script to accomplish the task. The script will be passed across a network socket from **batchmaster** to a slave application, and then the script will be interpreted by the slave. In particular, keep in mind that the file system seen by the script will be that of the machine on which the slave process is running.

This example uses the default `batchsolve.tcl` procs to run the simulation defined by the `taskA.mif` MIF 1.1 file. If you want to make changes to the MIF problem specifications on the fly, you will need to modify the default procs. This is done by creating a slave initialization script, via the call

```
$TaskInfo SetSlaveInitScript { <insert script here> }
```

The slave initialization script does global initializations, and also usually redefines the `SolverTaskInit` proc; optionally the `BatchTaskIterationCallback`, `BatchTaskRelaxCallback` and `SolverTaskCleanup` procs may be redefined as well. At the start of each task `SolverTaskInit` is called by `BatchTaskRun` (in `batchsolve.tcl`), after each iteration `BatchTaskIterationCallback` is executed, at each control point `BatchTaskRelaxCallback` is run, and at the end of each task `SolverTaskCleanup` is called. `SolverTaskInit` and `SolverTaskCleanup` are passed the arguments that were passed to `BatchTaskRun`. A simple `SolverTaskInit` proc could be

```
proc SolverTaskInit { args } {
   global mif basename outtextfile
   set A [lindex $args 0]
   set outbasename "$basename-A$A"
   $mif SetA $A
   $mif SetOutBaseName $outbasename
   set outtextfile [open "$outbasename.odt" "a+"]
```

```
    puts $outtextfile [GetTextData header \
          "Run on $basename.mif, with A=[$mif GetA]"]
}
```

This proc receives the exchange constant `A` for this task on the argument list, and makes use of the global variables `mif` and `basename`. (Both should be initialized in the slave initialization script outside the `SolverTaskInit` proc.) It then stores the requested value of `A` in the `mif` object, sets up the base filename to use for output, and opens a text file to which tabular data will be appended. The handle to this text file is stored in the global `outtextfile`, which is closed by the default `SolverTaskCleanup` proc. A corresponding task script could be

```
$TaskInfo AppendTask "A=13e-12 J/m" "BatchTaskRun 13e-12"
```

which runs a simulation with `A` set to 13×10^{-12} J/m. This example is taken from the `multitask.tcl` script in Sec. 10.2.2.4. (For commands accepted by `mif` objects, see the file `mmsinit.cc`. Another object than can be gainfully manipulated is `solver`, which is defined in `solver.tcl`.)

If you want to run more than one task at a time, then the `$TaskInfo` method `AppendSlave` will have to be invoked. This takes the form

```
$TaskInfo AppendSlave <spawn count> <spawn command>
```

where `<spawn command>` is the command to launch the slave process, and `<spawn count>` is the number of slaves to launch with this command. (Typically `<spawn count>` should not be larger than the number of processors on the target system.) The default value for this item (which gets overwritten with the first call to `$TaskInfo AppendSlave`) is

```
1 {Oc_Application Exec batchslave -tk 0 %connect_info batchsolve.tcl}
```

The Tcl command `Oc_Application Exec` is supplied by OOMMF and provides access to the same application-launching capability that is used by the OOMMF bootstrap application (Sec. 5). Using a `<spawn command>` of `Oc_Application Exec` instead of `exec %tclsh %oommf` saves the spawning of an additional process. The default `<spawn command>` launches the **batchslave** application, with connection information provided by **batchmaster**, and using the `auxscript batchsolve.tcl`.

Before evaluating the `<spawn command>`, **batchmaster** applies several percent-style substitutions useful in slave launch scripts: %tclsh, %oommf, %connect_info, %oommf_root, and %%. The first is the Tcl shell to use, the second is an absolute path to the OOMMF bootstrap program on the master machine, the third is connection information needed by the `batchslave` application, the fourth is the path to the OOMMF root directory on the master machine, and the last is interpreted as a single percent. **batchmaster** automatically appends the argument `&` to the `<spawn command>` so that the slave applications are launched in the background.

To launch **batchslave** on a remote host, use `rsh` in the spawn command, e.g.,

```
# FILE: simpletask.tcl
#
# This is a sample batch task file.  Usage example:
#
#   tclsh oommf.tcl batchmaster simpletask.tcl
#
# Form task list
$TaskInfo AppendTask A "BatchTaskRun taskA.mif"
$TaskInfo AppendTask B "BatchTaskRun taskB.mif"
$TaskInfo AppendTask C "BatchTaskRun taskC.mif"
```

Figure 1: Sample task script `simpletask.tcl`.

```
$TaskInfo AppendSlave 1 {exec rsh foo tclsh oommf/oommf.tcl \
      batchslave -tk 0 %connect_info batchsolve.tcl}
```

This example assumes `tclsh` is in the execution path on the remote machine `foo`, and OOMMF is installed off of your home directory. In addition, you will have to add the machine `foo` to the host connect list with

```
$TaskInfo ModifyHostList +foo
```

and `batchmaster` must be run with the network interface specified as the server host (instead of the default `localhost`), e.g.,

```
tclsh oommf.tcl batchmaster multitask.tcl bar
```

where `bar` is the name of the local machine.

This may seem a bit complicated, but the examples in the next section should make things clearer.

10.2.2.4 Sample task scripts The first sample task script (Fig. 1) is a simple example that runs the 3 micromagnetic simulations described by the MIF 1.1 files `taskA.mif`, `taskB.mif` and `taskC.mif`. It is launched with the command

```
tclsh oommf.tcl batchmaster simpletask.tcl
```

This example uses the default slave launch script, so a single slave is launched on the current machine, and the 3 simulations will be run sequentially. Also, no slave initialization script is given, so the default procs in `batchsolve.tcl` are used. Output will be magnetization states and tabular data at each control point, stored in files on the local machine with base names as specified in the MIF files.

The second sample task script (Fig. 2) builds on the previous example by defining `BatchTaskIterationCallback` and `BatchTaskRelaxCallback` procedures in the slave init

script. The first set up to write tabular data every 10 iterations, while the second writes tabular data on each control point event. The data is written to the output file specified by the `Base Output Filename` entry in the input MIF files. Note that there is no magnetization vector field output in this example. This task script is launched the same way as the previous example:

```
tclsh oommf.tcl batchmaster octrltask.tcl
```

The third task script (Fig. 3) is a more complicated example running concurrent processes on two machines. This script should be run with the command

```
tclsh oommf.tcl batchmaster multitask.tcl bar
```

where `bar` is the name of the local machine.

Near the top of the `multitask.tcl` script several Tcl variables (`RMT_MACHINE` through `A_list`) are defined; these are used farther down in the script. The remote machine is specified as `foo`, which is used in the `$TaskInfo AppendSlave` and `$TaskInfo ModifyHostList` commands.

There are two `AppendSlave` commands, one to run two slaves on the local machine, and one to run a single slave on the remote machine (`foo`). The latter changes to a specified working directory before launching the `batchslave` application on the remote machine. (For this to work you must have `rsh` configured properly. In the future it may be possible to launch remote commands using the OOMMF account server application, thereby lessening the reliance on system commands like `rsh`.)

Below this the slave initialization script is defined. The Tcl `regsub` command is used to place the task script defined value of `BASEMIF` into the init script template. The init script is run on the slave when the slave is first brought up. It first reads the base MIF file into a newly created `mms_mif` instance. (The MIF file needs to be accessible by the slave process, irrespective of which machine it is running on.) Then replacement `SolverTaskInit` and `SolverTaskCleanup` procs are defined. The new `SolverTaskInit` interprets its first argument as a value for the exchange constant `A`. Note that this is different from the default `SolverTaskInit` proc, which interprets its first argument as the name of a MIF 1.1 file to load. With this task script, a MIF file is read once when the slave is brought up, and then each task redefines only the value of `A` for the simulation (and corresponding changes to the output filenames and data table header).

Finally, the Tcl loop structure

```
foreach A $A_list {
    $TaskInfo AppendTask "A=$A" "BatchTaskRun $A"
}
```

is used to build up a task list consisting of one task for each value of `A` in `A_list` (defined at the top of the task script). For example, the first value of `A` is 10e-13, so the first task will have the label `A=10e-13` and the corresponding script is `BatchTaskRun 10e-13`. The

```
# FILE: octrltask.tcl
#
# This is a sample batch task file, with expanded output control.
# Usage example:
#
#        tclsh oommf.tcl batchmaster octrltask.tcl
#
# "Every" output selection count
set SKIP_COUNT 10

# Initialize solver. This is run at global scope
set init_script {
    # Text output routine
    proc MyTextOutput {} {
        global outtextfile
        puts $outtextfile [GetTextData data]
        flush $outtextfile
    }
    # Change control point output
    proc BatchTaskRelaxCallback {} {
        MyTextOutput
    }
    # Add output on iteration events
    proc BatchTaskIterationCallback {} {
        global solver
        set count [$solver GetODEStepCount]
        if { ($count % __SKIP_COUNT__) == 0 } { MyTextOutput }
    }
}

# Substitute $SKIP_COUNT in for __SKIP_COUNT__ in above "init_script"
regsub -all -- __SKIP_COUNT__ $init_script $SKIP_COUNT init_script
$TaskInfo SetSlaveInitScript $init_script

# Form task list
$TaskInfo AppendTask A "BatchTaskRun taskA.mif"
$TaskInfo AppendTask B "BatchTaskRun taskB.mif"
$TaskInfo AppendTask C "BatchTaskRun taskC.mif"
```

Figure 2: Task script with iteration output `octrltask.tcl`.

value 10e-13 is passed on by `BatchTaskRun` to the `SolverTaskInit` proc, which has been redefined to process this argument as the value for `A`, as described above.

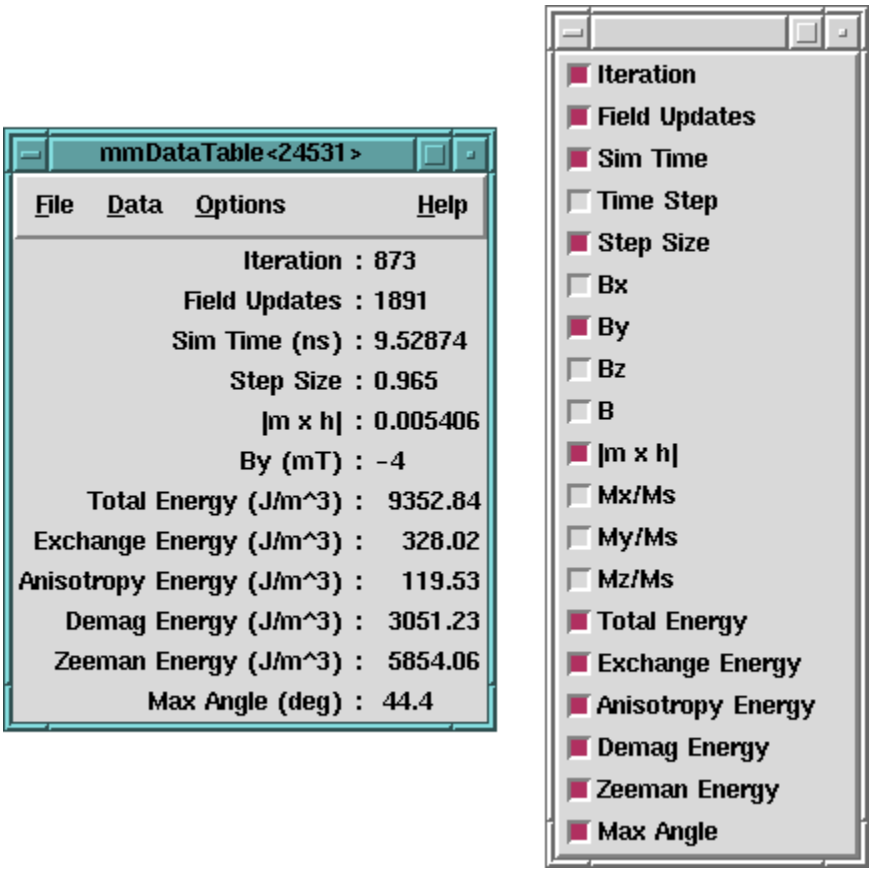
There are 6 tasks in all, and 3 slave processes, so the first three tasks will run concurrently in the 3 slaves. As each slave finishes it will be given the next task, until all the tasks are complete.

```
# FILE: multitask.tcl
#
# This is a sample batch task file.  Usage example:
#
#   tclsh oommf.tcl batchmaster multitask.tcl hostname [port]
#
# Task script configuration
set RMT_MACHINE   foo
set RMT_TCLSH      tclsh
set RMT_OOMMF      "/path/to/oommf/oommf.tcl"
set RMT_WORK_DIR   "/path/to/oommf/app/mmsolve/data"
set BASEMIF taskA
set A_list { 10e-13 10e-14 10e-15 10e-16 10e-17 10e-18 }

# Slave launch commands
$TaskInfo ModifyHostList +$RMT_MACHINE
$TaskInfo AppendSlave 2 "exec %tclsh %oommf batchslave -tk 0 \
        %connect_info batchsolve.tcl"
$TaskInfo AppendSlave 1 "exec rsh $RMT_MACHINE \
        cd $RMT_WORK_DIR \\\;\
        $RMT_TCLSH $RMT_OOMMF batchslave -tk 0 %connect_info batchsolve.tcl"

# Slave initialization script (with batchsolve.tcl proc
# redefinitions)
set init_script {
    # Initialize solver. This is run at global scope
    set basename __BASEMIF__      ;# Base mif filename (global)
    mms_mif New mif
    $mif Read [FindFile ${basename}.mif]
    # Redefine TaskInit and TaskCleanup proc's
    proc SolverTaskInit { args } {
        global mif outtextfile basename
        set A [lindex $args 0]
        set outbasename "$basename-A$A"
        $mif SetA $A
        $mif SetOutBaseName $outbasename
        set outtextfile [open "$outbasename.odt" "a+"]
```

```
        puts $outtextfile [GetTextData header \
                "Run on $basename.mif, with A=[$mif GetA]"]
        flush $outtextfile
    }
    proc SolverTaskCleanup { args } {
        global outtextfile
        close $outtextfile
    }
}
# Substitute $BASEMIF in for __BASEMIF__ in above script
regsub -all -- __BASEMIF__ $init_script $BASEMIF init_script
$TaskInfo SetSlaveInitScript $init_script

# Create task list
foreach A $A_list {
    $TaskInfo AppendTask "A=$A" "BatchTaskRun $A"
}
```

Figure 3: Advanced sample task script `multitask.tcl`.

11 Data Table Display: mmDataTable



Overview

The application **mmDataTable** provides a data display service to its client applications. It accepts data from clients which are displayed in a tabular format in a top-level window. Its typical use is to display the evolving values of quantities computed by micromagnetic solver programs.

Launching

mmDataTable may be started either by selecting the **mmDataTable** button on **mmLaunch**, or from the command line via

```
tclsh oommf.tcl mmDataTable [standard options] [-net <0|1>]
```

-net <0|1> Disable or enable a server which allows the data displayed by **mmDataTable** to be updated by another application. By default, the server is enabled. When the server is disabled, **mmProbEd** is only useful if it is embedded in another application.

Inputs

The client application(s) that send data to **mmDataTable** for display control the flow of data. The user, interacting with the **mmDataTable** window, controls how the data is displayed. Upon launch, **mmDataTable** displays only a menubar. Upon user request, a display window below the menubar displays data values.

Each message from a client contains a list of (name, value, units) triples containing data for display. For example, one element in the list might be {`Magnetization 800000 A/m`}. **mmDataTable** stores the latest value it receives for each name. Earlier values are discarded when new data arrives from a client.

Outputs

mmDataTable does not support any data output or storage facilities. To save tabular data, use the **mmGraph** (Sec. 12) or **mmArchive** (Sec. 14) applications.

Controls

The **Data** menu holds a list of all the data names for which **mmDataTable** has received data. Initially, **mmDataTable** has received no data from any clients, so this menu is empty. As data arrives from clients, the menu fills with the list of data names. Each data name on the list lies next to a checkbutton. When the checkbutton is toggled from off to on, the corresponding data name and its value and units are displayed at the bottom of the display window. When the checkbutton is toggled from on to off, the corresponding data name is removed from the display window. In this way, the user selects from all the data received what is to be displayed. Selecting the dashed rule at the top of the **Data** menu detaches it so the user may easily click multiple checkbuttons.

Displayed data values can be individually selected (or deselected) with a left mouse button click on the display entry. Highlighting is used to indicated which data values are currently selected. The **Options** menu also contains commands to select or deselect all displayed values. The selected values can be copied into the cut-and-paste (clipboard) buffer with the `CTRL-c` key combination, or the **Options|Copy** menu command.

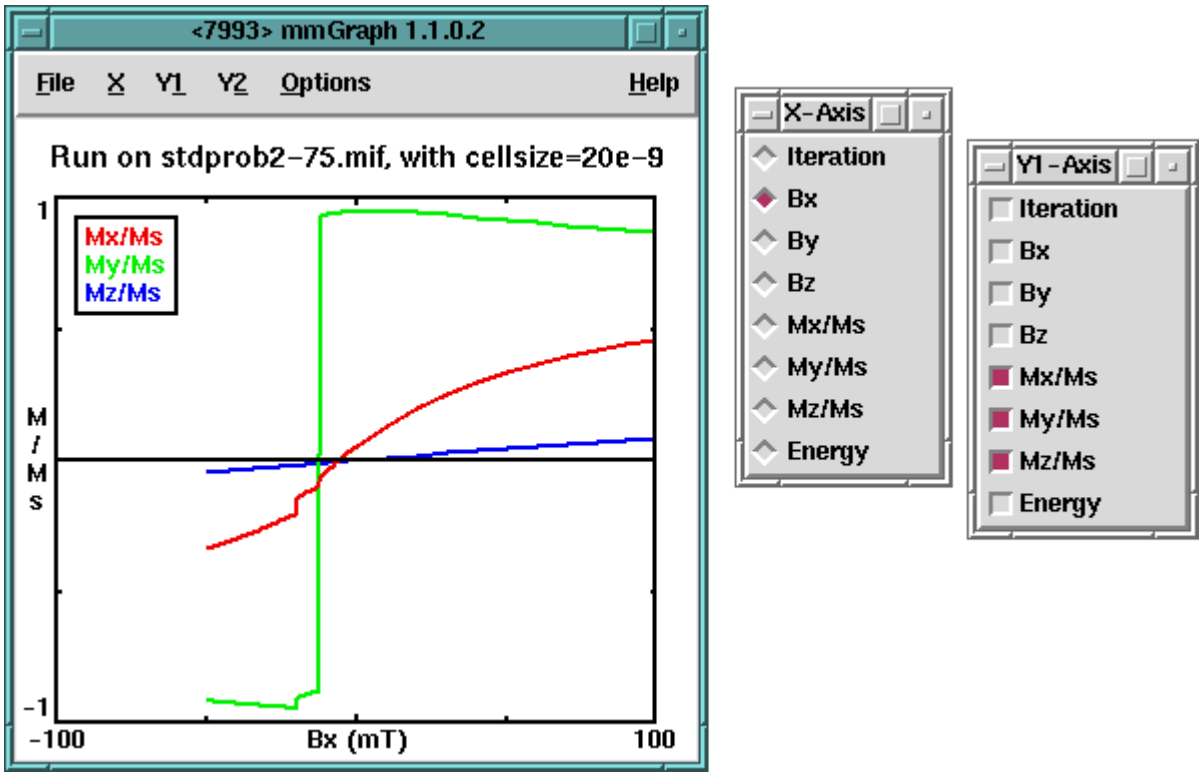
The data value selection mechanism is also used for data value formatting control. The **Options|Format** menu command brings up a **Format** dialog box to change the justification and format specification string. The latter is the conversion string passed to the Tcl `format` command, which uses the C `printf` format codes. If the **Adjust:Selected** radiobutton is active, then the specification will be applied to only the currently selected (highlighted) data values. Alternately, if **Adjust:All** is active, then the specification will be applied to all data values, and will additionally become the default specification.

A right mouse button click on a display entry will select that entry, and bring up the **Format** dialog box with the justification and format specifications of the selected entry. These specifications, with any revisions, may then be applied to all of the selected entries.

If a value cannot be displayed with the selected format specification string, e.g., if a "%d" integer format were applied to a string containing a decimal point, then the value will be printed in red in the form as received by **mmDataTable**, without any additional formatting.

The menu selection **File | Reset** reinitializes the **mmDataTable** application to its original state, clearing the display and the **Data** menu. The reset operation is also automatically invoked upon receipt of new data following a data set close message from a solver application. The menu selection **File | Exit** terminates the application. The menu **Help** provides the usual help facilities.

12 Data Graph Display: mmGraph



Overview

The application **mmGraph** provides a data display service similar to that of **mmDataTable** (Sec. 11). The usual data source is a running solver, but rather than the textual output provided by **mmDataTable**, **mmGraph** produces 2D line plots. **mmGraph** also stores the data it receives, so it can produce multiple views of the data and can save the data to disk. Postscript output is also supported.

Launching

mmGraph may be started either by selecting the **mmGraph** button on mmLaunch or from the command line via

```
tclsh oommf.tcl mmGraph [standard options] [-net <0|1>]
```

-net <0|1> Disable or enable a server which allows the data displayed by **mmGraph** to be updated by another application. By default, the server is enabled. When the server is disabled, **mmGraph** may only input data from a file.

Inputs

Input to **mmGraph** may come from either a file in the ODT format (Sec. 18), or when `-net 1` (the default) is active, from a client application (typically a running solver). The **File|Open...** dialog box is used to select an input file. Receipt of data from client applications is the same as for **mmDataTable** (Sec. 11). In either case, input data are appended to any previously held data.

Curve breaks (i.e., separation of a curve into disjoint segments) are recorded in the data storage buffer via *curve break records.* These records are generated whenever a `Table Start` or a `Table End` record is read from an ODT file, when a data set close message is received from a client application, or when requested by the user using the **mmGraph Options|Break** menu option.

Outputs

Unlike **mmDataTable**, **mmGraph** internally stores the data sent to it. These data may be written to disk via the **File|Save As...** dialog box. If the file specified already exists, then **mmGraph** output is appended to that file. The output is in the tabular ODT format described in Sec. 18. The data are segmented into separate `Table Start/Table End` blocks across each curve break record.

By default, all data currently held by **mmGraph** is written, but the **Save: Selected Data** option presented in the **File|Save As...** dialog box causes the output to be restricted to those curves currently selected for display. In either case, the graph display limits do *not* affect the output.

The save operation writes records that are held by **mmGraph** at the time the **File|Save As...** dialog box **OK** button is invoked. Additionally, the **Auto Save** option in this dialog box may be used to automatically append to the specified file each new data record as it is received by **mmGraph**. The appended fields will be those chosen at the time of the save operation, i.e., subsequent changing of the curves selected for display does not affect the automatic save operation. The automatic save operation continues until either a new output file is specified, the **Options|Stop autosave** control is invoked, or **mmGraph** is terminated.

The **File|Print...** dialog is used to produce a Postscript file of the current graph. On Unix systems, the output may be sent directly to a printer by filling the **Print to:** entry with the appropriate pipe command, e.g., `|lpr`. (The exact form is system dependent.)

Controls

Graphs are constructed by selecting any one item off the **X**-axis menu, and any number of items off the **Y1**-axis and **Y2**-axis menus. The y1-axis is marked on the left side of the graph; the y2-axis on the right. These menus may be detached by selecting the dashed rule at the top of the list. Sample results are shown in the figure at the start of this section.

When **mmGraph** is first launched, all the axis menus are empty. They are dynamically built based on the data received by **mmGraph**. By default, the graph limits and labels are

automatically set based on the data. The x-axis label is set using the selected item data label and measurement unit (if any). The y-axes labels are the measurement unit of the first corresponding y-axis item selected.

The **Options|Configure...** dialog box allows the user to override default settings. To change the graph title, simply enter the desired title into the **Title** field. To set the axis labels, deselect the **Auto Label** option in this dialog box, and fill in the **X Label**, **Y1 Label** and **Y2 Label** fields as desired. The axis limits can be set in a similar fashion. In addition, if an axis limit is left empty, a default value (based on all selected data) will be used.

The size of the margin surrounding the plot region is computed automatically. Larger margins may be specified by filling in the appropriate fields in the **Margin Requests** section. Units are pixels. Requested values smaller than the computed (default) values are ignored.

As mentioned earlier, **mmGraph** stores in memory all data it receives. Over the course of a long run, the amount of data stored can grow to many megabytes. This storage can be limited by specifying a positive (> 0) value for the **Point buffer size** entry in the **Options|Configure...** dialog box. The oldest records are removed as necessary to keep the total number of records stored under the specified limit. A zero value for **Point buffer size** is interpreted as no limit. (The storage size of an individual record depends upon several factors, including the number of items in the record and the version of Tcl being used.) Data erasures may not be immediately reflected in the graph display.

At any time, the point buffer storage may be completely emptied with the **Options|clear Data** command. The **Options|stop Autosave** selection will turn off the auto save feature, if currently active. Also on this menu is **Options|Rescale**, which autoscales the graph axis limits from the selected data. This command ignores but does not reset the "Auto Scale" settings in the **Options|Configure...** dialog box. The **Options|Break** item inserts a curve break record into the point buffer, causing a break in each curve after the current point. This option may be useful if **mmGraph** is being fed data from multiple sources.

The **Options|Key** selection toggles the key (legend) display on and off. The key may also be repositioned by dragging with the left mouse button. If curves are selected off both the y1 and y2 menus, then a horizontal line in the key separates the two sets of curves, with the labels for the y1 curves on top.

The last command on the options menu is **Options|Smooth**. If smoothing is disabled, then the data points are connected by straight line segments. If enabled, then each curve is rendered as a set of parabolic splines, which do not in general pass through the data points. This is implemented using the `--smooth 1` option to the Tcl `canvas create line` command; see that documentation for details.

The width of the lines may be changed by the usual method of local customization (Sec. 2.3.2). The default setting in `config/options.tcl` is

```
Oc_Option Add mmGraph Ow_GraphWin curve_width 2
```

which makes the line segments 2 pixels wide. This can be set to any positive integer value. Curves will be rendered more quickly, especially on Windows, if `curve_width` is set to 1.

A few other controls are also available only through the mouse. If the mouse pointer is positioned over a drawn item in the graph, holding down the left mouse button will bring up the coordinates of that point, with respect to the y1-axis. Similarly, depressing the right mouse button, or alternatively holding down the shift key while pressing the left mouse button will bring up the coordinates of the point with respect to the y2-axis. The coordinates displayed are the coordinates of a point on a drawn line, which are not necessarily the coordinates of a plotted data point. (The data points are plotted at the endpoints of each line segment.) The coordinate display is cleared when the mouse button is released.

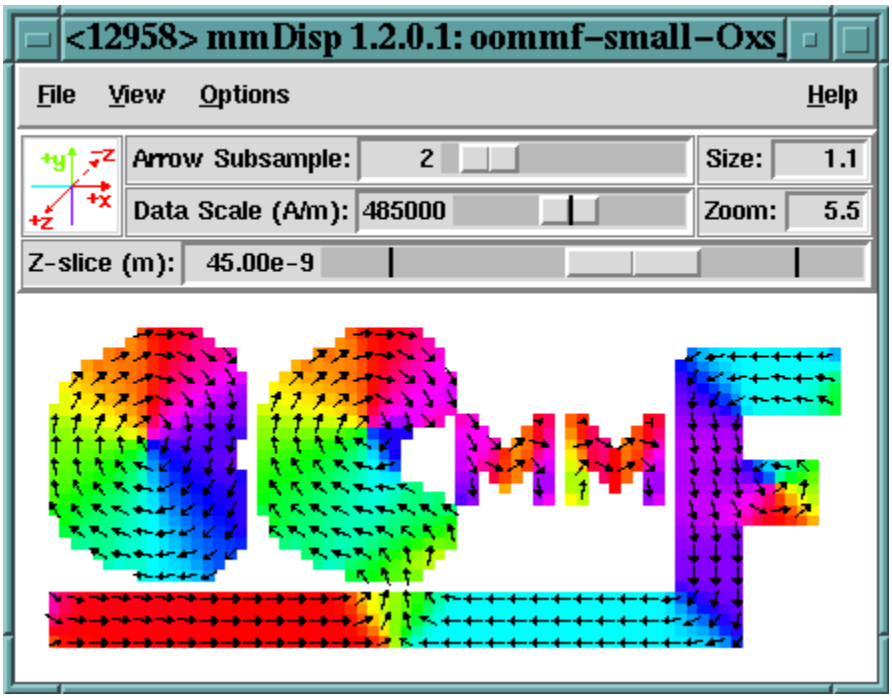
One vertical and one horizontal rule (line) are also available. Initially, these rules are tucked and hidden against the left and bottom graph axes, respectively. Either may be repositioned by dragging with the left or right mouse button.

The menu selection **File|Exit** terminates the **mmGraph** application. The menu **Help** provides the usual help facilities.

Details

The axes menus are configured based on incoming data. As a result, these menus are initially empty. If a graph widget is scheduled to receive data only upon control point or stage done events in the solver, it may be a long time after starting a problem in the solver before the graph widget can be configured. Because **mmGraph** keeps all data up to the limit imposed by the `Point buffer size`, data loss is usually not a problem. Of more importance is the fact that automatic data saving can not be set up until the first data point is received. As a workaround, the solver initial state may be sent interactively as a dummy point to initialize the graph widget axes menus. Select the desired quantities off the axes menus, and use the **Options|clear Data** command to remove the dummy point from **mmGraph**'s memory. The **File|Save As...** dialog box may then be used—with the **Auto Save** option enabled—to write out an empty table with proper column header information. Subsequent data will be written to this file as they arrive.

13 Vector Field Display: mmDisp



Overview

The application **mmDisp** displays two-dimensional spatial distributions of three-dimensional vectors (i.e., vector fields). It can load vector fields from files in a variety of formats, or it can accept vector field data from a client application, typically a running solver. **mmDisp** offers a rich interface for controlling the display of vector field data, and can also save the data to a file and produce Postscript print output.

Launching

mmDisp may be started either by selecting the **mmDisp** button on mmLaunch, or from the command line via

```
tclsh oommf.tcl mmDisp [standard options] [-net <0|1>] [filename]
```

-net <0|1> Disable or enable a server which allows the data displayed by **mmDisp** to be updated by another application. By default, the server is enabled. When the server is disabled, **mmDisp** may only input data from a file.

If a filename is supplied on the command line, **mmDisp** takes it to be the name of a file containing vector field data for display. That file will be opened on startup.

Inputs

Input to **mmDisp** may come from either a file or from a client application (typically a running solver), in any of the vector field formats described in Sec. 19. Other file formats can also be supported if a translation filter program is available.

Client applications that send data to **mmDisp** control the flow of data. The user, interacting with the **mmDisp** window, determines how the vector field data are displayed.

File input is initiated through the **File|Open...** dialog box. Several example files are included in the OOMMF release in the directory `app/mmdisp/examples`. When the **Browse** button is enabled, the "Open File" dialog box will remain open after loading a file, so that multiple files may be displayed in sequence. The **Auto** configuration box determines whether the vector subsampling, data scale, zoom and slice settings should be determined automatically (based on the data in the file and the current display window size), or whether their values should be held constant while loading the file.

mmDisp permits local customization allowing for automatic translation from other file formats into one of the vector field formats (Sec. 19) that **mmDisp** recognizes. When loading a file, **mmDisp** compares the file name to a list of glob-style patterns. These patterns typically match on the filename extension. An example pattern is `*.gz`. The assumption is that the pattern identifies files containing data in a particular format. For each pattern in the list, there is a corresponding translation program. **mmDisp** calls on that program as a filter which takes data in one format from standard input and writes to standard output the same data in one of the formats supported by **mmDisp**. In its default configuration, **mmDisp** recognizes the pattern `*.gz` and invokes the translation program `gzip -dc` to perform the "translation." In this way, support for reading gzip compressed files is "built in" to **mmDisp** on any platform where the **gzip** program is installed.

New patterns and translation programs may be added to **mmDisp** by the usual method of local customization (Sec. 2.3.2). The command to add to the customization file is of the form

```
Oc_Option Add mmDisp Input filters {{*.gz {gzip -dc}}}
```

The final argument in this command is a list of pairs. The first element in each pair is the filename pattern. The second element in each pair is the command line for launching the corresponding translation program. If a program `foo` were known to translate a file format identified by the extension `.bar` into the OVF file format, that program could be made known to **mmDisp** by changing the above customization command to:

```
Oc_Option Add mmDisp Input filters {{*.gz {gzip -dc}} {*.bar foo}}
```

Outputs

The vector field displayed by **mmDisp** may be saved to disk via the **File|Save As...** dialog box. The output is in the OVF format (Sec. 19.1). The OVF file options may be set by

selecting the appropriate radio buttons in the OVF File Options panel. The **Title** and **Desc** fields may be edited before saving. Enabling the **Browse** button allows for saving multiple files without closing the "Save File" dialog box.

The **File|Print...** dialog is used to produce a Postscript file of the current display. On Unix systems, the output may be sent directly to a printer by filling the **Print to:** entry with the appropriate pipe command, e.g., `|lpr`. (The exact format is system dependent.)

To produce bitmap output, save the file to disk in the OVF format, and use the **avf2ppm** (Sec. 16.4) utility to do the conversion.

Controls

The menu selection **File|Clear** clears the display window. The menu selection **File|Exit** terminates the **mmDisp** application. The menu **Help** provides the usual help facilities.

The **View** menu provides high-level control over how the vector field is placed in the display window. The menu selection **View|Wrap Display** resizes the display window so that it just contains the entire vector field surrounded by a margin. **View|Fill Display** resizes the vector field until it fills the current size of the display window. If the aspect ratio of the display window does not match the aspect ratio of the vector field, a larger than requested margin appears along one edge to make up the difference. **View|Rotate ccw** and **View|Rotate cw** rotate the display one quarter turn counter-clockwise and clockwise respectively. The display window also rotates, so that the portion of the vector field seen and any margins are preserved (unless the display of the control bar forces the display window to be wider). **View|reDraw** allows the user to invoke a redrawing of the display window. The **View|Viewpoint** tearable submenu supports rotation of the vector field out of the plane of the display, so that it may be viewed from along a different axis.

The menu selection **Options|Configure...** brings up a dialog box through which the user may control many features of the vector field display. Vectors in the vector field may be displayed as arrows, pixels, or both. The **Arrow** and **Pixel** buttons in the **Plot type** column on the left of the dialog box enable each type of display.

Columns 2–4 in the Configure dialog box control the use of color. Both arrows and pixels may be independently colored to indicate some quantity. The **Color Quantity** column controls which scalar quantity the color of the arrow or pixel represents. Available color quantities include vector x, y, and z components, total vector magnitude, slice depth, and angles as measured in-plane from a fixed axis. On regularly gridded data the vector field divergence is also available for display.

The assignment of a color to a quantity value is determined by the **Colormap** selected. Colormaps are labeled by a sequence of colors that are mapped across the range of the selected quantity. For example, if the "Red-Black-Blue" colormap is applied to the **Color Quantity** "z", then vectors pointing into the xy-plane ($z < 0$) are colored red, those lying in the plane ($z = 0$) are colored black, and those pointing out of the plane ($z > 0$) are colored blue. Values between the extremes are colored with intermediate colors, selected using a discretization determined by the **# of Colors** value. This value governs the use of

potentially limited color resources, and can be used to achieve some special coloring effects. (Note: The in-plane angle quantities are generally best viewed with a colormap that begins and ends with the same color, e.g., "Red-Green-Blue-Red.") The ordering of the colormap can be reversed by selecting the **Reverse** checkbox. For example, this would change the "Red-Black-Blue" colormap to effectively "Blue-Black-Red."

Below the **Reverse** checkbutton in the pixel plot type row is a **Opaque** checkbutton. If this is selected then arrows below the top row in the pixel slice range (see slice discussion below) will be hidden by the pixel object. If disabled, then the pixel object is translucent, so objects further below are partially visible.

When there are many vectors in a vector field, a display of all of them may be more confusing than helpful. The **Subsample** column allows the user to request that only a sampling of vectors from the vector field be displayed. The **Subsample** value is roughly the number of vectors along one spatial dimension of the vector field which map to a single displayed vector (arrow or pixel). Each vector displayed is an actual vector in the vector field—the selection of vectors for display is a sampling process, not an averaging or interpolation process. The subsample rates for arrows and pixels may be set independently. A subsample rate of 0 is interpreted specially to display all data. (This is typically much quicker than subsampling at a small rate, e.g., 0.1.)

The length of an arrow represents the magnitude of the vector field. All arrows are drawn with a length between zero and "full-scale." By default, the full-scale arrow length is computed so that it covers the region of the screen that one displayed vector is intended to represent, given the current subsample rate. Following this default, arrows do not significantly overlap each other, yet all non-zero portions of the vector field have a representation in the display. Similarly, pixels are drawn with a default size that fills an area equal to the region of the screen one pixel is intended to represent, given the pixel subsample rate. The **Size** column allows the user to (independently) override the default size of pixels and full-scale arrows. A value of 1 represents the default size. By changing to a larger or smaller **Size** value, the user may request arrows or pixels larger or smaller than the default size.

Below the Arrow and Pixel Controls are several additional controls. The **Data Scale** entry affects the data value scaling. As described above, all arrows are displayed with length between zero and full-scale. The full-scale arrow length corresponds to some scalar value of the magnitude of the vector field. The **Data Scale** entry allows the user to set the value at which the drawn arrow length goes full-scale. Any vectors in the vector field with magnitude equal to or greater than the data scale value will be represented by arrows drawn at full scale. Other vectors will be represented by shorter arrows with length determined by a linear scale between zero and the data scale value. Similarly, the data scale value controls the range of values spanned by the colormap used to color pixels. The usual use of the **Data Scale** entry is to reduce the data scale value so that more detail can be seen in those portions of the vector field which have magnitude less than other parts of the vector field. If the data scale value is increased, then the length of the arrows in the plot is reduced accordingly. If the data scale value is decreased, then the length of the arrows is increased, until they reach full-scale. An arrow representing a vector with magnitude larger than the data scale

value may be thought of as being truncated to the data scale value. The initial (default) data scale value is usually the maximum vector magnitude in the field, so at this setting no arrows are truncated. Entering `0` into the data scale box will cause the data scale to be reset to the default value. (For OVF files (Sec. 19.1), the default data scale value is set from the `ValueRangeMaxMag` header line. This is typically set to the maximum vector magnitude, but this is not guaranteed.) The data scale control is intended primarily for use with vector fields of varying magnitude (e.g., **H**-fields), but may also be used to adjust the pixel display contrast for any field type.

The **Zoom** entry controls the spatial scaling of the display. The value roughly corresponds to the number of pixels per vector in the fully-sampled vector field. (This value is not affected by the subsampling rate.)

The **Margin** entry specifies the margin size, in pixels, to be maintained around the vector field.

The next row of entry boxes control slice display. Slice selection allows display of that subset of the data that is within a specified distance of a plane running perpendicular to the view axis. The location of that plane with respect to the view axis is specified in the **X-slice center**, **Y-slice center** or **Z-slice center** entry, depending on the current view axis. The thickness of the slice may be varied separately for arrow and pixel displays, as specified in the next two entry boxes. The slice span boxes interpret specially the following values: 0 resets the slice thickness to the default value, which is usually the thickness of a single cell. Any negative value sets the slice thickness to be the full thickness of the mesh. Values for all of the slice control entries are specified in the fundamental mesh spatial unit, for example, meters. (Refer to the vector field file format (Sec. 19) documentation for more on mesh spatial units.)

Below the slice contols are controls to specify whether or not a bounding polygon is displayed, and the background color for the display window.

No changes made by the user in the **Options|Configure...** dialog box affect the display window until either the **Apply** or **OK** button is selected. If the **OK** button is selected, the dialog box is also dismissed. The **Close** button dismisses the dialog without changing the display window.

The next item under the **Options** menu is a checkbutton that toggles the display of a control bar. The control bar offers alternative interfaces to some of the operations available from the **Options|Configure...** dialog box and the **View** menu. On the left end of the control bar is a display of the coordinate axes. These axes rotate along with the vector field in the display window to identify the coordinate system of the display, and are color coded to agree with the pixel (if active) or arrow coloring. A click of the left mouse button on the coordinate axes causes a counter-clockwise rotation. A click of the right mouse button on the coordinate axes causes a clockwise rotation.

To the right of the coordinate axes are two rows of controls. The top row allows the user to control the subsample rate and size of displayed arrows. The subsample rate may be modified either by direct entry of a new rate, or by manipulation of the slider. The second row controls the current data scale value and zoom (spatial magnification). A vertical bar

in the slider area marks the default data scale value. Specifying `0` for the data scale value will reset the data scale to the default value.

The spatial magnification may be changed either by typing a value in the Zoom box of the control bar, or by using the mouse inside the display window. A click and drag with the left mouse button displays a red rectangle that changes size as the mouse is dragged. When the left mouse button is released, the vector field is rescaled so that the portion of the display window within the red rectangle expands until it reaches the edges of the display window. Both dimensions are scaled by the same amount so there is no aspect distortion of the vector field. Small red arrows on the sides of the red rectangle indicate which dimension will expand to meet the display window boundaries upon release of the left mouse button. After the rescaling, the red rectangle remains in the display window briefly, surrounding the same region of the vector field, but at the new scale.

A click and drag with the right mouse button displays a blue rectangle that changes size as the mouse is dragged. When the right mouse button is released, the vector field is rescaled so that all of the vector field currently visible in the display window fits the size of the blue rectangle. Both dimensions are scaled by the same amount so there is no aspect distortion of the vector field. Small blue arrows on the sides of the blue rectangle indicate the dimension in which the vector field will shrink to exactly transform the display window size to the blue rectangle size. After the rescaling, the blue rectangle remains in the display window briefly, surrounding the same region of the vector field, now centered in the display window, and at the new scale.

When the zoom value is large enough that a portion of the vector field lies outside the display window, scrollbars appear that may be used to translate the vector field so that different portions are visible in the display window. On systems that have a middle mouse button, clicking the middle button on a point in the display window translates the vector field so that the selected point is centered within the display window.

mmDisp remembers the previous zoom value and data scale values. To revert to the previous settings, the user may hit the `ESC` key. This is a limited "Undo" feature.

Below the data scale and zoom controls in the control bar is the slice center selection control. This will be labeled **Z-slice**, **X-slice**, or **Y-slice**, depending on which view axis is selected. The thickness of the slice can be set from the **Options|Configure...** dialog box.

The final item under the **Options** menu is the **Options|Lock size** checkbutton. By default, when the display is rotated in-plane, the width and height of the viewport are interchanged so that the same portion of the vector field remains displayed. Selecting the **Options|Lock size** checkbutton disables this behavior, and also other viewport changing operations (e.g., display wrap).

Several keyboard shortcuts are available as alternatives to menu- or mouse-based operations. (These are in addition to the usual keyboard access to the menu.) The effect of a key combination depends on which subwindow of **mmDisp** is active. The `TAB` key may be used to change the active subwindow. The `SHIFT-TAB` key combination also changes the active subwindow, in reverse order.

When the active subwindow is the display window, the following key combinations are

active:

- `CTRL-o` – same as menu selection **File|Open...**
- `CTRL-s` – same as menu selection **File|Save as...**
- `CTRL-p` – same as menu selection **File|Print...**
- `CTRL-c` – same as menu selection **Options|Configure...**
- `CTRL-v` – launches viewpoint selection menu, **View|Viewpoint**
- `CTRL-w` – same as menu selection **View|Wrap Display**
- `CTRL-f` – same as menu selection **View|Fill Display**
- `HOME` – First fill, then wrap the display.
- `CTRL-space` – Center view in display.
- `CTRL-r` – same as menu selection **View|Rotate ccw**
- `SHIFT-CTRL-r` – same as menu selection **View|Rotate cw**
- `INSERT` – decrease arrow subsample by 1
- `DEL` – increase arrow subsample by 1
- `SHIFT-INSERT` – decrease arrow subsample by factor of 2
- `SHIFT-DEL` – increase arrow subsample by factor of 2
- `PAGEUP` – increase the zoom value by a factor of 1.149
- `PAGEDOWN` – decrease the zoom value by a factor of 1.149
- `SHIFT-PAGEUP` – increase the zoom value by factor of 2
- `SHIFT-PAGEDOWN` – decrease the zoom value by factor of 2
- `ESC` – revert to previous data scale and zoom values

When the active subwindow is the control bar's coordinate axes display, the following key combinations are active:

- `LEFT` – same as menu selection **View|Rotate ccw**
- `RIGHT` – same as menu selection **View|Rotate cw**

When the active subwindow is any of the control bar's value entry windows – arrow subsample, size, data scale or zoom, the following key combinations are active:

- `ESC` – undo uncommitted value (displayed in red)
- `RETURN` – commit entered value

When the active subwindow is in any of the control bar's sliders—arrow subsample, data scale or slice—the following key combinations are active:

- `LEFT` – slide left (decrease value)
- `RIGHT` – slide right (increase value)
- `ESC` – undo uncommitted value (displayed in red)
- `RETURN` – commit current value

When any of the separate dialog windows are displayed (e.g., the **File|Open...** or **Options|Configure...** dialogs), the shortcut `CTRL-.` (control-period) will raise and transfer keyboard focus back to the root **mmDisp** window.

Details

The selection of vectors for display according to the **Subsample** differs depending on whether or not the data lie on a regular grid. If so, **Subsample** takes integer values and determines the ratio of data points to displayed points. For example, a value of 5 means that every fifth vector on the grid is displayed. This means that the number of vectors displayed is 25 times fewer than the number of vectors on the grid.

For an irregular grid of vectors, an average cell size is computed, and the **Subsample** takes values in units of 0.1 times the average cell size. A square grid of that size is overlaid on the irregular grid. For each cell in the square grid, the data vector from the irregular grid closest to the center of the square grid cell is selected for display. The vector is displayed at its true location in the irregular grid, not at the center of the square grid cell. As the subsample rate changes, the set of displayed vectors also changes, which can in some circumstances substantially change the appearance of the displayed vector field.

Using mmDisp as a WWW browser helper application

You may configure your web browser to automatically launch **mmDisp** when downloading an OVF file. The exact means to do this depends on your browser, but a couple of examples are presented below.

In Netscape Navigator 4.X, bring up the **Edit|Preferences...** dialog box, and select the `Category` **Navigator|Applications** subwindow. Create a **New Type**, with the following fields:

Description of type: OOMMF Vector Field

MIME Type: application/x-oommf-vf

Suffixes: ovf,omf,ohf,obf

Application: *tclsh oommfroot*/oommf.tcl +fg mmDisp -net 0 "*arg*"

On Windows platforms, the `Suffixes` field is labeled `File Extension`, and only one file extension may be entered. Files downloaded from a web server are handled according to their MIME Type, rather than their file extension, so that restriction isn't important when web browsing. If you wish to have files on the local disk with all the above file extensions recognized as OOMMF Vector Field files, you must repeat the **New Type** entry for each file extension. In the `Application` field, the values of *tclsh*, *oommfroot*, and *arg* vary with your platform configuration. The value of *tclsh* is the full path to the **tclsh** application on your platform (see Section 5). On Unix systems, *tclsh* may be omitted, assuming that the `oommf.tcl` script is executable. If *tclsh* is not omitted on Unix systems, Netscape may issue a security warning each time it opens an OOMMF Vector Field file. The value of *oommfroot* should be the full path to the root directory of your OOMMF installation. The value of *arg* should be "%1" on Windows and "%s" on Unix. The MIME type "application/x-oommf-vf" must be configured on any HTTP server which provides OOMMF Vector Field files as well.

For Microsoft Internet Explorer 3.X, bring up the **View|Options...** dialog box, and select the **Program** tab. Hit the **File Types...** button, followed by the **New Type...** button. Fill the resulting dialog box with

Description of type: OOMMF Vector Field

Associated extension: ovf

Content type (MIME): application/x-oommf-vf

You may also disable the **Confirm open after download** checkbutton if you want. Then hit the **New...** button below the `Actions:` window, and in the pop-up fill in

Action: open

Application used to perform action:
tclsh oommfroot/oommf.tcl +fg mmDisp -net 0 "%1"

Hit **OK**, **Close**, **Close** and **OK**. Replace *tclsh* and *oommfroot* with the appropriate paths on your system (cf. Section 5). This will set up an association on files with the .ovf extension. Internet Explorer 3.X apparently ignores the HTML Content Type field, so you must repeat this process for each file extension (.ovf, .omf, .ohf, .obf and .svf) that you want to recognize. This means, however, that Internet Explorer will make the appropriate association even if the HTML server does not properly set the HTML Content Type field.

Microsoft Internet Explorer 4.X is integrated with the Windows operating system. Internet Explorer 4.X doesn't offer any means to set up associations between particular file types and the applications which should be used to open them. Instead, this association is configured within the Windows operating system. To set up associations for the OOMMF

Vector Field file type on Windows 95 or Windows NT, select **Settings|Control Panel** from the **Start** menu. The Control Panel window appears. Select **View|Options...** to display a dialog box. A Windows 98 shortcut to the same dialog box is to select **Settings|Folder Options...** from the **Start** menu. Select the **File Types** tab and proceed as described above for Internet Explorer 3.X. Depending on the exact release/service patch of your Windows operating system, the exact instructions may vary.

Once you have your browser configured, you can test with the following URL:

http://math.nist.gov/~MDonahue/cubevortex.ovf

Known Bugs

The slice selection feature does not work properly with irregular meshes.

14 Data Archive: mmArchive

Overview

The application **mmArchive** provides automated vector field and data table storage services. Although **mmDisp** (Sec. 13) and **mmGraph** (Sec. 12) are able to save such data under the direction of the user, there are situations where it is more convenient to write data to disk without interactive control.

mmArchive does not present a user interface window of its own, but like **mmSolve2D** (Sec. 10.1) relies on **mmLaunch** (Sec. 6) to provide an interface on its behalf. Because **mmArchive** does not require a window, it is possible on Unix systems to bring down the X (window) server and still keep **mmArchive** running in the background.

Launching

mmArchive may be started either by selecting the **mmArchive** button on **mmLaunch**, or from the command line via

```
tclsh oommf.tcl mmArchive [standard options]
```

When the **mmArchive** button of **mmLaunch** is invoked, **mmArchive** is launched with the `-tk 0` option. This allows **mmArchive** to continue running if the X window server is killed. The `-tk 1` option is useful only for enabling the `-console` option for debugging.

As noted above, **mmArchive** depends upon **mmLaunch** to provide an interface. The entry for an instance of **mmArchive** in the **Threads** column of any running copy of **mmLaunch** has a checkbutton next to it. This button toggles the presence of a user interface window through which the user may control that instance of **mmArchive**.

Inputs

mmArchive accepts vector field and data table style input from client applications (typically running solvers) on its network (socket) interface.

Outputs

The client applications that send data to **mmArchive** control the flow of data. **mmArchive** copies the data it receives into files specified by the client. There is no interactive control to select the names of these output files. A simple status line shows the most recent vector file save, or data table file open/close event.

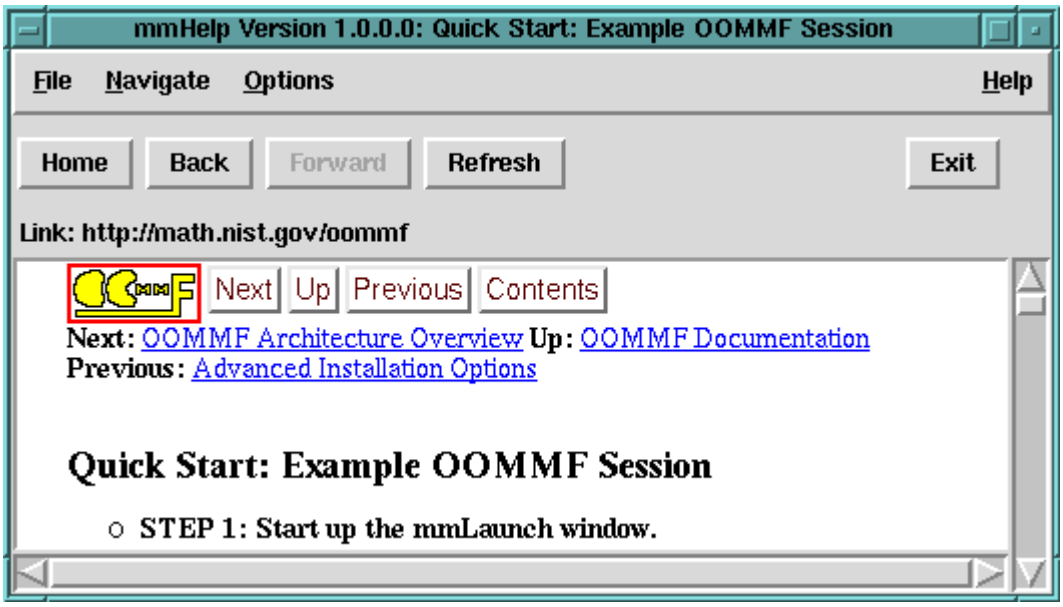
Controls

The **Exit** in the **mmLaunch**-supplied user interface terminates **mmArchive**. Simply closing the user interface window does not terminate **mmArchive**, but only hides the control window. To kill **mmArchive** the **Exit** button must be pressed.

Known Bugs

mmArchive appends data table output to the file specified by the source client application (e.g., a running solver). If, at the same time, more than one source specifies the same file, or if the the same source sends data table output to more than one instance of **mmArchive**, then concurrent writes to the same file may corrupt the data in that file. It is the responsibility of the user to ensure this does not happen; there is at present no file locking mechanism in OOMMF to protect against this situation.

15 Documentation Viewer: mmHelp



Overview

The application **mmHelp** manages the display and navigation of hypertext (HTML) help files. It presents an interface similar to that of World Wide Web browsers.

Although **mmHelp** is patterned after World Wide Web browsers, it does not have all of their capabilities. **mmHelp** displays only a simplified form of hypertext required to display the OOMMF help pages. It is not able to display many of the advanced hypertext features provided by modern World Wide Web browsers. In the current release, **mmHelp** is not able to follow `http:` URLs. It only follows `file:` URLs, such as

file:/path/to/oommf/doc/userguide/userguide/Documentation_Viewer_mmHelp.html

OOMMF software can be customized (See Sec. 2.3.2) to use another program to display the HTML help files.

Launching

mmHelp may be launched from the command line via

```
tclsh oommf.tcl mmHelp [standard options] [URL]
```

The command line argument `URL` is the URL of the first page (home page) to be displayed. If no URL is specified, **mmHelp** displays the Table of Contents of the *OOMMF User's Guide* by default.

Controls

Each page of hypertext is displayed in the main **mmHelp** window. Words which are underlined and colored blue are hyperlinks which **mmHelp** knows how to follow. Words which are underlined and colored red are hyperlinks which **mmHelp** does not know how to follow. Moving the mouse over a hyperlink displays the target URL of the hyperlink in the **Link:** line above the display window. Clicking on a blue hyperlink will follow the hyperlink and display a new page of hypertext.

mmHelp keeps a list of the viewed pages in order of view. Using the **Back** and **Forward** buttons, the user may move backward and forward through this list of pages. The **Home** button causes the first page to be displayed, allowing the user to start again from the beginning. These three buttons have corresponding entries in the **Navigate** menu.

Use the menu selection **File|Open** to directly select a file from the file system to be displayed by **mmHelp**.

The menu selection **File|Refresh**, or the **Refresh** button causes **mmHelp** to reload and redisplay the current page. This may be useful if the display becomes corrupted, or for repeatedly loading a hypertext file which is being edited.

When **mmHelp** encounters hypertext elements it does not recognize, it will attempt to work around the problem. However, in some cases it will not be able to make sense of the hypertext, and will display an error message. Documentation authors should take care to use only the hypertext elements supported by **mmHelp** in their documentation files. Users should never see such an error message.

mmHelp displays error messages in one of two ways: within the display window, or in a separate window. Errors reported in the display window replace the display of the page of hypertext. They usually indicate that the hypertext page could not be retrieved, or that its contents are not hypertext. File permission errors are also reported in this way.

Errors reported in a separate window are usually due to a formatting error within the page of hypertext. Selecting the **Continue** button of the error window instructs **mmHelp** to attempt to resume display of the hypertext page beyond the error. Selecting **Abort** abandons further display.

The menu selection **Options|Font scale...** brings up a dialog box through which the user may select the scale of the fonts to use in the display window, relative to their initial size.

The menu selection **File|Exit** or the **Exit** button terminates the **mmHelp** application. The menu **Help** provides the usual help facilities.

Known Bugs

mmHelp is pretty slow. You may be happier using local customization (Sec. 2.3.2) methods to replace it with another more powerful HTML browser. Also, we have noticed that the underscore character in the italic font is not displayed (or is displayed as a space) at some font sizes on some platforms.

16 Command Line Utilities

This section documents a few utilities distributed with OOMMF that are run from the command line (Unix shell or Windows DOS prompt). They are typically used in pre- or post-processing of data associated with a micromagnetic simulation.

16.1 Bitmap File Format Conversion: any2ppm

The **any2ppm** program converts bitmap files of various formats into the Portable Pixmap (PPM) P3 (text) format. Supported input formats are PPM, BMP, and GIF. Additional formats may be available if the Tcl/Tk *Img*[12] package is installed on your system. (Note: OOMMF support for BMP requires Tk 8.0 or later.)

Launching

The **any2ppm** launch command is:

```
tclsh oommf.tcl any2ppm [standard options] [-noinfo] \
   [-o outfile] [infile ...]
```

where

-noinfo Suppress writing of progress information to stderr.

-o outfile Write output to `outname`; use "-" to pipe output to stdout. The default is to create a new file by stripping the extension, if any, off of each input filename, and appending `.ppm`. If the generated filename already exists, a "-000" or "-001" ... suffix is appended. If the input is coming from stdin, i.e., there is no input filename, then the default output is to stdout. Note that if `outname` is specified, then all output will go to this one file; in this case it is unlikely that one wants to specify more than one input file.

infile ... List of input files to process. If none, or if an infile is the empty string, then read from stdin.

Tk Requirement: any2ppm uses the Tk `image` command in its processing. This requires that Tk be properly initialized, which in particular means that a valid display must be available. This is not a problem on Windows, where a desktop is always present, but on Unix this means that an X server must be running. The *Xvfb*[13] virtual framebuffer can be used if desired. (Xvfb is an X server distributed with X11R6 that requires no display hardware or physical input devices.)

[12] http://purl.oclc.org/net/nijtmans/img.html
[13] http://www.itworld.com/AppDev/1461/UIR000330xvfb/

16.2 Making Data Tables from Vector Fields: avf2odt

The **avf2odt** program converts rectangularly meshed vector field files in any of the recognized formats (OVF, VIO; see Sec. 19) into the ODT 1.0 (Sec. 18) data table format.

Launching

The **avf2odt** launch command is:

```
tclsh oommf.tcl avf2odt [standard options] \
   [-average <space|plane|line|point>] [-axis <x|y|z>] \
   [-region <xmin> <ymin> <zmin> <xmax> <ymax> <zmax>] \
   infile >outfile
```

where

-average <space|plane|line|point> Specify type of averaging. `Space` outputs 1 data line consisting of the average v_x, v_y and v_z field values in the selected region (see `-region` option below). For example, in magnetization files, v_x, v_y and v_z correspond to M_x, M_y and M_z. If `plane` or `line` is selected, then the output data table consists of multiple lines with 4 or 5 columns respectively. The last 3 columns in both cases are the v_x, v_y and v_z averaged over the specified axes-parallel affine subspace (i.e., plane or line). In the `plane` case, the first column specifies the averaging plane offset along the coordinate axis normal to the plane (see `-axis` option below). In the `line` case, the first 2 columns specify the offset of the averaging line in the coordinate plane perpendicular to the line. If `-type` is set to `point`, then no averaging is done, and the output consists of 6 column data lines, one line for each point in the selected region, where the first 3 columns are the point coordinates, and the last 3 are the v_x, v_y and v_z values at the point.

This parameter is optional. The default value is `space`.

-axis <x|y|z> For the `-type plane` and `-type line` averaging types, selects which subset of affine subspaces the averaging will be performed over. In the `plane` case, the `-axis` represents the normal direction to the planes, while for `line` it is the direction parallel to the lines. This parameter is ignored if `-type` is `space` or `point`. Optional; default is `x`.

-region <xmin> <ymin> <zmin> <xmax> <ymax> <zmax> Axes-parallel rectangular box denoting region in the vector field file over which data is to be collected. The locations are in problem units (typically meters). A single hyphen, "-", may be specified for any of the box corner coordinates, in which case the corresponding extremal value from the input file is used. Optional; the default, `-region - - - - - -`, selects the entire input file.

infile Name of input file to process. Must be one of the recognized formats, OVF 1.0 or VIO, in a rectangular mesh subformat. Required.

>outfile Avf2odt writes its output to stdout. Use the redirection operator ">" to send the output to a file. For output format details, see the ODT file description (Sec. 18).

Note: The m_x, m_y and m_z average magnetization values reported by mmSolve2D (Sec. 10.1) exclude points with 0 saturation magnetization. Such points are *included* by **avf2odt**, so the data table output from this program will probably not agree with that directly output by **mmSolve2D** if there are any such regions.

16.3 Vector Field File Format Conversion: avf2ovf

The **avf2ovf** program converts vector field files from any of the recognized formats (OVF, VIO; see Sec. 19) into the OVF 1.0 format.

Launching

The **avf2ovf** launch command is:

```
tclsh oommf.tcl avf2ovf [standard options] \
   [-clip xmin ymin zmin xmax ymax zmax] [-flip flipstr] \
   [-format <text|b4|b8>] [-grid <reg|irreg>] [-info] \
   infile >outfile
```

where

-clip xmin ymin zmin xmax ymax zmax The arguments specify the vertices of a bounding clip box. Only mesh points inside the clip box are brought over into the output file.

-flip flipstr Provides an axis coordinate transformation. Flipstr has the form A:B:C, where A, B, C is a permutation of x, y, z, with an optional minus sign on each entry. The first component A denotes the axis to which x is mapped, B where y is mapped, and C where z is mapped. The default is the identity map, `x:y:z`. To rotate 90° about the z-axis, use "-flip y:-x:z", which sends x to the $+y$ axis, y to the -x axis, and leaves z unchanged.

-format <text|b4|b8> Specify output data format. The default is ASCII `text`; `b4` selects 4-byte binary, `b8` selects 8-byte binary. (The OVF format has an ASCII text header in all cases.)

-grid <reg|irreg> Specify output grid structure. The default is `reg`, which will output a regular (rectangular) grid if the input is recognized as a regular grid. The option "-grid irreg" forces irregular mesh style output.

-info Instead of converting the file, print to stderr information about the file, such as size, range, and descriptive text from the file header.

infile Name of input file to process. Must be one of the recognized formats, OVF 0.0, OVF 1.0, or VIO.

>outfile Avf2ovf writes its output to stdout. Use the redirection operator ">" to send the output to a file.

The `-clip` option is useful when one needs to do analysis on a small piece of a large simulation. The `-info` option is helpful here to discover the extents of the original mesh.

The `-flip` option can change be used to align different simulations to the same orientation. It can also be used to change a file into its mirror image; for example, "-flip -x:y:z" reflects the mesh through the yz-plane.

The `-format text` and `-grid irreg` options are handy for preparing files for import into non-OOMMF applications, since all non-data lines are readily identified by a leading "`#`," and each data line is a 6-tuple consisting of the node location and vector value. Pay attention, however, to the scaling of the vector value as specified by "`# valueunit`" and "`# valuemultiplier`" header lines.

For output format details, see the OVF file description (Sec. 19.1).

16.4 Making Bitmaps from Vector Fields: avf2ppm

The **avf2ppm** utility converts a collection of vector field files (e.g., `.omf`, `.ovf`) into color bitmaps suitable for inclusion into documents or collating into movies. The command line arguments control filename and format selection, while plain-text configuration files, modeled after the **mmDisp** (Sec. 13) configuration dialog box, specify conversion parameters.

Launching

The **avf2ppm** launch command is:

```
tclsh oommf.tcl avf2ppm [standard options] [-config file] [-f] \
   [-filter program] [-format <P3|P6|B24>] [-ipat pattern] \
   [-opatexp regexp] [-opatsub sub] [-v level] [infile ...]
```

where

-config file User configuration file that specifies the image conversion parameters. This file is discussed in detail below.

-f Force overwriting of existing (output) files. By default, if **avf2ppm** tries to create a file, say `foo.ppm`, that already exists, it generates instead a new name of the form `foo.ppm-000`, or `foo.ppm-001`, ..., or `foo.ppm-999`, that doesn't exist and writes to that instead. The `-f` flag disallows alternate filename generation, and overwrites `foo.ppm` instead.

-filter program Post-processing application to run on each **app2ppm** output file. May be a pipeline of many programs.

-format <P3|P6|B24> Specify the output image file format. Currently supported formats are the true color *Portable Pixmap* (PPM) formats P3 (ASCII text) and P6 (binary), and the uncompressed Windows BMP 24 bits-per-pixel format. The default is P6.

-ipat pattern Specify input files using a pattern including "glob-style" wildcards. Mostly useful in DOS.

-opatexp regexp Specify the "regular expression" applied to input filenames to determine portion to be replaced in generation of output filenames. Default: `(\.[^.]?[^.]?[^.]?$|$)`

-opatsub sub The string with which to replace the portion of input filenames matched by the `-opatexp regexp` during output filename generation. The default is `.ppm` for type P3 and P6 file output, `.bmp` for B24 file output.

-v level Verbosity (informational message) level, with 0 generating only error messages, and larger numbers generating additional information. The `level` value is an integer, defaulting to 1.

infile ... List of input files to process.

The file specification options require some explanation. Input files may be specified either by an explicit list (`infile ...`), or by giving a wildcard pattern, e.g., `-ipat *.omf`, which is expanded in the usual way by **avf2ppm** (using the Tcl command `glob`). Unix shells (sh, csh, etc.) automatically expand wildcards before handing control over to the invoked application, so the `-ipat` option is not needed (although it is useful in case of a "command-line too long" error). DOS does not do this expansion, so you must use `-ipat` to get wildcard expansion in Windows.

As each input file is processed, a name for the output file is produced from the input filename by rules determined by handing the `-opatexp` and `-opatsub` expressions to the Tcl `regsub` command. Refer to the Tcl `regsub` documentation for details, but essentially whatever portion of the input filename is matched by the `-opatexp` expression is removed and replaced by the `-opatsub` string. The default `-opatexp` expression matches against any filename extension of up to 3 characters, and the default `-opatsub` string replaces this with the extension `.ppm` or `.bmp`.

If you have command line image processing "filter" programs, e.g., **ppmtogif** (part of the NetPBM package), then you can use the `-filter` option to pipe the output of **avf2ppm** through that filter before it is written to the output file specified by the `-opat*` expressions. If the processing changes the format of the file, (e.g., **ppmtogif** converts from PPM to GIF), then you will likely want to specify a `-opatsub` different from the default.

Here is an example that processes all files with the `.omf` extension, sending the output through **ppmtogif** before saving the results in a files with the extension `.gif`:

```
tclsh oommf.tcl avf2ppm -ipat *.omf -opatsub .gif -filter ppmtogif
```

(On Unix, either drop the `-ipat` flag, or use quotes to protect the input file specification string from expansion by the shell, as in `-ipat '*.omf'`.) You may also pipe together multiple filters, e.g., `-filter "ppmquant 256 | ppmtogif"`.

Configuration files

The details of the conversion process are specified by plain-text configuration files, with fields analogous to the entries in the **mmDisp** (Sec. 13) configuration dialog box. Each of the parameters is an element in an array named `plot_config`. The default values for this array are taken from the default configuration file `oommf/app/mmdisp/scripts/avf2ppm.def`, which is a Tcl script read during **avf2ppm** initialization.

There are several places in the configuration file where colors are specified. Colors may be represented using the symbolic names in `oommf/config/colors.def`, in any of the Tk hexadecimal formats, e.g., `#RRGGBB`, or as a shade of gray using the format "grayD" (or "greyD"), where D is a decimal integer from 0-100, inclusive. Examples in the latter two formats are `#FFFF00` for yellow, `gray0` for black and `gray100` or `#FFFFFF` for white.

The sample default configuration script shown in Fig. 4 (page 121) can be used as a starting point for user (per-run) configuration files. Refer to this sample file and the mmDisp documentation (Sec. 13) as we discuss each element of the array `plot_config`. (See the Tcl documentation for details of the `array set` command.)

colormaps A list of valid colormaps known to the program. This entry is *not* user-configurable, and should not appear in user configuration files.

arrow,status Set to 1 to display arrows, 0 to not draw arrows.

arrow,colormap Select the colormap to use when drawing arrows. Should be one of the strings specified in the `colormaps` array element.

arrow,colorcount Number of discretization levels to use from the colormap. A value of zero will color all arrows with the first color in the colormap.

arrow,quantity Scalar quantity the arrow color is to represent. Supported values include `x`, `y`, `z`, `xy-angle`, `xz-angle`, `yz-angle`, and `slice`. The **mmDisp** configuration dialog box will present the complete list of allowed quantities (which may be image dependent).

arrow,colorphase The phase is a real number between -1 and 1 that provides a translation in the selected `colormap`. For the `xy-angle`, `xz-angle` and `yz-angle` color quantities, this displays as a rotation of the colormap, for example, setting colorphase to 0.333

would effectively change the `Red-Green-Blue-Red` colormap into `Green-Blue-Red-Green`. For the other color quantities, it simply shifts the display band, saturating at one end. For example, setting colorphase to 0.5 changes the `Blue-White-Red` colormap into `White-Red-Red`.

arrow,colorinvert The `colorinvert` value should be 1 or 0, signifying to invert or not invert, respectively. If invert is selected, the the colormap ordering is reversed, changing for example `Blue-White-Red` into `Red-White-Blue`. If both inversion and phase adjustment are requested, then inversion is applied first.

arrow,autosample If 1, then ignore the value of `arrow,subsample` and automatically determine a "reasonable" subsampling rate for the arrows. Set to 0 to turn off this feature.

arrow,subsample If `arrow,autosample` is 0, then subsample the input vectors at this rate when drawing arrows. A value of 0 for `arrow,subsample` is interpreted specially to display all data.

arrow,size Size of the arrows relative to the default size (represented as 1.0).

arrow,antialias If 1, then each pixel along the edge of an arrow is drawn not with the color of the arrow, but with a mixture of the arrow color and the background color. This makes arrow boundaries appear less jagged, but increases computation time. Also, the colors used in the anti-aliased pixels are not drawn from the arrow or pixel colormap discretizations, so color allocation in the output bitmap may increase dramatically.

pixel,... Most of the pixel configuration elements have analogous arrow configuration elements, and are interpreted in the same manner. The exceptions are the `pixel,opaque` element which is discussed next, the `arrow,antialias` element which has no corresponding pixel element, and the qualitative difference in that the auto subsampling rate for pixels is considerably denser than for arrows.

pixel,opaque If the opaque value is 1, then the pixel is drawn in a solid manner, concealing any arrows which are drawn under it. If opaque is 0, then the pixel is drawn only partially filled-in, so objects beneath it can still be discerned.

misc,background Specify the background color.

misc,drawboundary If 1, then draw the bounding polygon, if any, as specified in the input vector field file.

misc,boundarywidth Width of the bounding polygon, in pixels. A value of 0 is treated the same as 1. To disable drawing of the boundary, use the `misc,drawboundary` option.

misc,boundarycolor Color of the bounding polygon.

misc,boundarypos Placement of the bounding polygon, either `back` or `front`, i.e., either behind or in front of the rendered arrows and pixel elements.

misc,matwidth Specifies the width, in pixels, of a mat (frame) around the outer edge of the image. The mat is drawn in front of all other objects. To disable, set matwidth to 0.

misc,matcolor Color of the mat.

misc,margin The size of the border margin, in pixels.

misc,width, misc,height Maximum width and height of the output bitmap, in pixels. If `misc,crop` is enabled, then one or both of these dimensions may be shortened.

misc,crop If disabled (0), then any leftover space in the bitmap (of dimensions `misc,width` by `misc,height`) after packing the image are filled with the background color. If enabled (1), then the bitmap is cropped to just include the image (with the margin specified by `misc,margin`). **NOTE:** Some movie formats require that bitmap dimensions be multiples of 8 or 16. For such purposes, you should disable `misc,crop` and specify appropriate dimensions directly with `misc,width` and `misc,height`.

misc,zoom Scaling factor for the display. This is the same value as shown in the "zoom" box in the **mmDisp** control bar, and corresponds roughly to the number of pixels per vector in the (original, fully-sampled) vector field. If set to zero, then **avf2ppm** will automatically set the scaling so the image (with margins) just fits inside the area specified by `misc,width` and `misc,height`.

misc,rotation Counterclockwise rotation in degrees; either 0, 90, 180 or 270.

misc,datascale Scale for arrow size and colormap range; this element is exactly equivalent to the **mmDisp** `Data Scale` control, Sec. 13. In general, this should be a positive real value, but a zero or empty value will set the scaling to its default value.

misc,centerpt This element should either be empty, or else be a three item list of real numbers specifying the center of the display, in input file mesh units. The OOMMF command line program **avf2ovf** (Sec. 16.3) may be invoked with the `-info` option to determine the range of the file mesh. This information is also contained in the "Bounding box" message returned when **avf2ppm** is run, which should in any event be consulted since any portion of the mesh that is shifted outside the bounding box will not be rendered. This element provides the functionality of the scroll bars on the **mmDisp** main display window. If an empty value is given, then by default the center of the mesh is used.

viewaxis Select the view axis, which should be one of `+z`, `-z`, `+y`, `-y`, `+x`, or `-x`. This option is equivalent to the **mmDisp** **View|Viewpoint** menu control.

viewaxis,center The `viewaxis,center` element corresponds to the **mmDisp** "slice" selection control. It should be a single real number in the range of that portion of the input file mesh bounding box corresponding to the selected view axis. For example, if the view axis is `+z` or `-z`, then the view axis center value should lie between the minimum and maximum z-values of the file mesh. If an empty value is given, then the view axis center is set from the corresponding coordinate of the `misc,centerpt` value.

The portion of the mesh that is rendered depends upon the interaction between the bounding box, the center point, the view axis center, and the axis-selected arrow and pixel spans.

viewaxis,xarrowspan, viewaxis,yarrowspan, viewaxis,zarrowspan Specifies the thickness of the arrow display slice, for the corresponding view, e.g., if the view axis is `+z` or `-z`, then only **viewaxis,zarrowspan** is active. These values for each of these elements should be either a single real value in units relating to the corresponding coordinate range of the input file mesh, or else an empty string, which is interpreted to be the entire thickness of the mesh in the selected view direction.

viewaxis,xpixelspan, viewaxis,ypixelspan, viewaxis,zpixelspan Identical interpretation and behavior as the corresponding arrow span elements, but with regards to pixel display.

User (per-run) configuration files are specified on the command line with the `-config` option. To create a user configuration file, make a copy of the default `avf2ppm.def` configuration file, and edit it as desired in a plain text editor. You may omit any entries that you do not want to change from the default. (Each entry consists of a name + value pair, e.g., `misc,width 640`.) You may "layer" configuration files by specifying multiple user configuration files on the command line. These are processed from left to right, with the last value set for each entry taking precedence.

16.5 Vector Field File Difference: avfdiff

The **avfdiff** program computes differences between vector field files in any of the recognized formats (OVF, VIO; see Sec. 19). The input data must lie on rectangular meshes with identical dimensions.

Launching

The **avfdiff** launch command is:

```
tclsh oommf.tcl avfdiff [standard options] file-0 file-1 [... file-n]
```

where

file-0 Name of input file to subtract from other files. Must be either an OVF 1.0 file in the rectangular mesh subformat, or an VIO file. Required.

```
array set plot_config {
    colormaps  {
        Red-Black-Blue     Blue-White-Red      Teal-White-Red
        Black-Gray-White   Black-White-Black   White-Black-White
        White-Green-Black  Red-Green-Blue-Red
    }
    arrow,status       1               misc,background     #FFFFFF
    arrow,colormap  Black-Gray-White   misc,drawboundary   1
    arrow,colorcount   0               misc,boundarywidth  1
    arrow,quantity     z               misc,boundarycolor  #000000
    arrow,colorphase   0.              misc,boundarypos    front
    arrow,colorinvert  0               misc,matwidth       0
    arrow,autosample   1               misc,matcolor       #FFFFFF
    arrow,subsample    10              misc,margin         10
    arrow,size         1               misc,width          640
    arrow,antialias    1               misc,height         480
                                       misc,crop           1
    pixel,status       1               misc,zoom           0
    pixel,colormap  Teal-White-Red     misc,rotation       0
    pixel,colorcount   225             misc,datascale      0
    pixel,opaque       1               misc,centerpt       {}
    pixel,quantity     x
    pixel,colorphase   0.              viewaxis            {+z}
    pixel,colorinvert  0               viewaxis,center     {}
    pixel,autosample   1               viewaxis,xarrowspan {}
    pixel,subsample    0               viewaxis,xpixelspan {}
    pixel,size         1               viewaxis,yarrowspan {}
                                       viewaxis,ypixelspan {}
                                       viewaxis,zarrowspan {}
                                       viewaxis,zpixelspan {}
}
```

Figure 4: Sample default configuration script `avf2ppm.def`.

file-1 Name of first input file from which `file-0` is to be subtracted. Must also be either an OVF 1.0 file in the rectangular mesh subformat, or an VIO file, and must have the same dimensions as `file-0`. Required.

...file-n Optional additional files from which `file-0` is to be subtracted, with the same requirements as `file-1`.

For each input file `file-1` through `file-n`, a separate output file is generated, in the OVF 1.0 format. Each output file has a name based on the name of corresponding input file, with a `-diff` suffix. If a file with the same name already exists, it will be overwritten.

For output file format details, see the OVF file description (Sec. 19.1).

16.6 Calculating H Fields from Magnetization: mag2hfield

The **mag2hfield** utility takes a MIF 1.1 micromagnetic problem specification file (`.mif`, see Sec. 17.2) and a magnetization file (`.omf`, see Sec. 19) and uses the mmSolve2D (Sec. 10.1) computation engine to calculate the resulting component (magnetostatic, exchange, crystalline anisotropy, Zeeman) and total energy and/or **H** fields. The main use of this utility to study the fields in a simulation using magnetization files generated by an earlier **mmSolve2D** run.

Launching

The **mag2hfield** launch command is:

```
tclsh oommf.tcl mag2hfield [standard options]
   [-component [all,][anisotropy,][demag,][exchange,][total,][zeeman] \
   [-data [energy,][field]] [-energyfmt fmt] [-fieldstep #] \
   mif_file omf_file
```

where

-component [all,][anisotropy,][demag,][exchange,][total,][zeeman] Specify all energy/field components that are desired. Optional; default is `total`, which is the sum of the crystalline anisotropy, demagnetization (self-magnetostatic), exchange, and Zeeman (applied field) terms.

-data [energy,][field] Calculate energies, **H** fields, or both. Energy values are printed to stdout, **H** fields are written to files as described below. Optional; the default is `energy,field`.

-energyfmt fmt Output C printf-style format string for energy data. Optional. The default format string is `"%s"`.

-fieldstep # Applied field step index, following the schedule specified in the input MIF file (0 denotes the initial field). Optional; default is 0.

mif_file MIF micromagnetic problem specification file (.mif). Required.

omf_file Magnetization state file. This can be in any of the formats accepted by the `avfFile` record of the input MIF file. Required.

The **H** field output file format is determined by the `Total Field Output Format` record of the input MIF 1.1 file (Sec. 17.2). The output file names are constructed using the form *basename*`-hanisotropy.ohf`, *basename*`-hzeeman.ohf`, etc., where *basename* is the input `.omf` magnetization file name, stripped of any trailing `.omf` or `.ovf` extension.

16.7 MIF Format Conversion: mifconvert

The **mifconvert** utility converts a MIF 1.1 (Sec. 17.2) micromagnetic problem specification file into the MIF 2.1 (Sec. 17.1) format. Eventually, it should be possible to express any problem in the MIF 1.1 format using the MIF 2.1 format, but currently that is not the case. It is recommended that the user carefully inspect the MIF 2.1 files generated by this application for correctness.

As a migration aid, **mifconvert** will also convert most files from the obsolete MIF 2.0 format used with OOMMF 1.2a2 into the newer MIF 2.1 format.

Launching

The **mifconvert** launch command is:

```
tclsh oommf.tcl mifconvert [-f|--force] [-h|--help] [-q|--quiet] \
   [--unsafe] [-v|--version] input_file output_file
```

where

-f or –force Force overwrite of output file. Optional.

-h or –help Print help information and stop. Optional.

-q or –quiet Suppress normal informational output messages. Optional.

–unsafe Runs embedded Tcl scripts, if any, in unsafe interpreter. Optional.

-v or –version Print version string and stop. Optional.

input_file Import MIF 1.1 (or MIF 2.0) micromagnetic problem specification file. Required.

output_file Export MIF 2.1 micromagnetic problem specification file. Required.

16.8 ODT Column Extraction: odtcols

The **odtcols** utility extracts column subsets from ODT (Sec. 18) data table files.

Launching

The **odtcols** launch command is:

```
tclsh oommf.tcl odtcols [standard options] [-s] [-w colwidth] \
   [-f format] [col ...] <infile >outfile
```

where

-s Produces a file summary instead of column extraction. Output includes table titles, column and row counts, and the header for each specified column. If no columns are specified, then the headers for all the columns are listed.

-w colwidth Minimum horizontal spacing to provide for each column on output. Optional. Default value is 15.

-f format C printf-style format string for each output item. Optional. The default format string is `"%-${colwidth}s"`.

col ... Output column selections. These may either be integers representing the position of the column in the input data (with the first column numbered as 0), or else arbitrary strings used for case-insensitive glob-style matching against the column headers. The columns are output in match order, obtained by processing the column selections from left to right. If no columns are specified then by default all columns are selected.

<infile Odtcols reads its input from stdin. Use the redirection operator "<" to read input from a file.

>outfile Odtcols writes its output to stdout. Use the redirection operator ">" to send the output to a file.

Commonly the `-s` switch is used in a first pass, to reveal the column headers; specific column selections may then be made in a second, separate invocation.

16.9 Platform-Independent Make: pimake

The application **pimake** is similar in operation to the Unix utility program `make`, but it is written entirely in Tcl so that it will run anywhere Tcl is installed. Like `make`, `pimake` controls the building of one file, the *target*, from other files. Just as `make` is controlled by rules in files named `Makefile` or `makefile`, `pimake` is controlled by rules in files named `makerules.tcl`.

Launching

The **pimake** launch command is:

```
tclsh oommf.tcl pimake [standard options] \
   [-d] [-i] [-k] [-out file] [target]
```

where

-d Print verbose information about dependencies.

-i Normally an error halts operation. When `-i` is specified, ignore errors and try to continue updating all dependencies of target.

-k Normally an error halts operation. When `-k` is specified, and an error is encountered, stop processing dependencies which depend on the error, but continue updating other dependencies of target.

-out file Write output to named file instead of to the standard output.

target The file to build. May also be (and usually is) a symbolic target name. See below for standard symbolic targets. By default, the first target in `makerules.tcl` is built.

There are several targets which may be used as arguments to **pimake** to achieve different tasks. Each target builds in the current directory and all subdirectories. The standard targets are:

upgrade Used immediately after unpacking a distribution, it removes any files which were part of a previous release, but are not part of the unpacked distribution.

all Creates all files created by the `configure` target (see below). Compiles and links all the executables and libraries. Constructs all index files.

configure Creates subdirectories with the same name as the platform type. Constructs a `ocport.h` file which includes C++ header information specific to the platform.

objclean Removes the intermediate object files created by the compile and link steps. Leaves working executables in place. Leaves OOMMF in the state of its distribution with pre-compiled executables.

clean Removes the files removed by the `objclean` target. Also removes the executables and libraries created by the `all` target. Leaves the files generated by the `configure` target.

distclean Removes the files removed by the `clean` target. Also removes all files and directories generated by `configure` target. Leaves only the files which are part of the source code distribution.

maintainer-clean Remove all files which can possibly be generated from other files. The generation might require specialized developer tools. This target is not recommended for end-users, but may be helpful for developers.

help Print a summary of the standard targets.

17 Problem Specification File Formats (MIF)

Micromagnetic simulations are specified to the OOMMF solvers using the OOMMF *Micromagnetic Input Format* (MIF). There are two distinct, incompatible versions of this format. The older, version 1.1, is the format used by the 2D solver (**mmSolve2D**, Sec. 10.1 and **batchsolve**, Sec. 10.2.1) and the **mmProbEd** (Sec. 8) problem editor. The new MIF format, version 2.1, is used by the Oxs 3D solver (Sec. 7). In both cases all values are in SI units. A command line utility **mifconvert** (Sec. 16.7) is provided to aid in converting MIF 1.1 files to the MIF 2.1 format. For both versions it is recommended that MIF files be given names ending with the `.mif` file extension.

17.1 MIF 2.1

The MIF 2.x format was introduced with the Oxs 3D solver (Sec. 7). It is *not* backwards compatible with the MIF 1.1 format, however a conversion utility, **mifconvert** (Sec. 16.7), is available to aid in converting MIF 1.1 files to the MIF 2.1 format.

A sample MIF 2.1 file is presented in Fig. 5 (Sec. 17.1.4, pages 141–144). The first line of a MIF file must be of the form "`#` MIF x.y", where x.y represents the format revision number, here 2.1. Unlike MIF 1.1 files, the structure of MIF 2.1 files are governed by the requirement that they be valid Tcl scripts.

During processing MIF 2.1 files are evaluated inside a Tcl interpreter. That interpreter may be a "safe" interpreter, i.e., disk and other system access is disabled. Refer to documentation of the Tcl `interp` command for details. The security level is controlled by the `MIFinterp` option in the `options.tcl` customization file (Sec. 2.3.2). The default setting is

```
Oc_Option Add Oxs* MIFinterp safety custom
```

which enables all the Tcl interpreter extensions described below, but is otherwise a standard Tcl safe interpreter. The keyword `custom` above may be replaced with either `safe` or `unsafe`. The `safe` selection is similar to `custom`, except that the `ReadFile` extension is not provided, thereby eliminating all disk access. At the other extreme, choosing `unsafe` provides an unrestricted Tcl interpreter. This option should be used with caution, especially if you are reading MIF files from an unknown or untrusted source.

17.1.1 MIF 2.1 Extension Commands

In addition to the standard Tcl commands (modulo the safe Tcl restrictions outlined above), several additional commands are available in MIF 2.1 files: `Specify`, `ClearSpec`, `Destination`, `Ignore`, `OOMMFRootDir`, `Parameter`, `Random`, `RandomSeed`, `Report`, `ReadFile`, and `Schedule`.

Specify An Oxs simulation is built as a collection of `Oxs_Ext` (Oxs Extension) objects. In general, `Oxs_Ext` objects are specified and initialized in the input MIF 2.1 file using the `Specify` command, making Specify blocks the primary component of the problem definition. The `Specify` command takes two arguments: the name of the `Oxs_Ext`

object to create, and an *initialization string* that is passed to the `Oxs_Ext` object during its construction. The objects are created in the order in which they appear in the MIF file. Order is important in some cases, for example, if one `Oxs_Ext` object refers to another in its initialization string, then the referred to object must precede the referrer in the MIF file.

Here is a simple Specify block:

```
Specify Oxs_EulerEvolve:foo {
  alpha 0.5
  start_dm 0.01
}
```

The name of the new `Oxs_Ext` object is "Oxs_EulerEvolve:foo." The first part of this name, up to the colon, is the the C++ class name of the object. This must be a child of the `Oxs_Ext` class. Here, `Oxs_EulerEvolve` is a class that integrates the Landau-Lifshitz ODE using a simple forward Euler method. The second part of the name, i.e., the part following the colon, is the name for this particular instance of the object. In general, it is possible to have multiple instances of an `Oxs_Ext` child class in a simulation, but each instance must have a unique name. These names are used for identification by output routines, and to allow one Specify block to refer to another Specify block appearing earlier in the MIF file. If the second part of the name is not given, then as a default the empty string is appended. For example, if instead of "Oxs_EulerEvolve:foo" above the name was specified as just "Oxs_EulerEvolve", then the effective full name of the created object would be "Oxs_EulerEvolve:".

The second argument to the `Specify` command, here everything between the curly braces, is a string that is interpreted by the new `Oxs_Ext` (child) object in its constructor. The format of this string is up to the designer of the child class, but there are a number of conventions that designers are encouraged to follow. These conventions are described in Specify Conventions, Sec. 17.1.2, below.

ClearSpec This command is used to disable one or all preceding `Specify` commands. In particular, one could use `ClearSpec` to nullify a Specify block from a base MIF file that was imported using the `ReadFile` command. Sample usage is

```
ClearSpec Oxs_EulerEvolve:foo
```

where the parameter is the full name (here `Oxs_EulerEvolve:foo`) of the Specify block to remove. If no parameter is given, then all preceding Specify blocks are removed.

Destination The format for the `Destination` command is

```
Destination <desttag> <appname> [new]
```

This command associates a symbolic *desttag* with an application. The tags are used by the `Schedule` command (see below) to refer to specific application instances. The *appname* may either be an OOMMF application name, e.g., mmDisp, or else a specific application instance in the form application:nickname, such as mmDisp:Spock. In the first case, the tag is associated with the running instance of the requested application (here `mmDisp`) with the lowest OOMMF ID (OID) that has not yet been associated with another tag. If no running application can be found that meets these criteria, then a new instance of the application is launched.

If the *appname* refers to a specific application instance, then the tag is associated with the running instance of the application (say **mmDisp**) that has been assigned the specified nickname. Name matching is case insensitive. If there is no running copy of the application meeting this condition, then a new instance of the application is launched and it is assigned the specified nickname. The OOMMF account service directory guarantees that there is never more than one instance of an application with a given nickname. However, as with the object name in the `Specify` command, instances of two different applications, e.g., **mmDisp** and **mmGraph**, are allowed to share nicknames, because their full instance names, say mmDisp:Spock and mmGraph:Spock, are unique.

The `Destination` commands are processed in the order in which they appear in the the MIF file. No *desttag* may appear in more than one `Destination` command, and no two destination tags may refer to the same application instance. To insure the latter, the user is advised to place all `Destination` commands referring to specific instances (e.g., mmDisp:Spock) before any `Destination` commands using generic application references (e.g., mmDisp). Otherwise a generic reference might be associated to a running application holding a nickname that is referenced by a later `Destination` command.

The tag association by the `Destination` command is only known to the solver reading the MIF file. In contrast, assigned instance nicknames are recognized across applications. In particular, multiple solvers may reference the same running application by nickname. For example, several sequential solver runs could send stage output to the same **mmGraph** widget, to build up overlapping hysteresis loops.

The last parameter to the Destination command is the optional `new` keyword. If present, then a fresh copy of the requested application is always launched for association with the given tag. The `new` option can be safely used with any generic application reference, but caution must be taken when using this option with specific instance references, because an error is raised if the requested nickname is already in use.

Ignore The `Ignore` command takes an arbitrary number of arguments, which are thrown away without being interpreted. The primary use of `Ignore` is to temporarily "comment out" (i.e., disable) Specify blocks.

OOMMFRootDir This command takes no arguments, and returns the full directory path of the OOMMF root directory. This is useful in conjunction with the `ReadFile` command for locating files within the OOMMF hierarchy, and can also be used to place output files. File paths must be created directly since the Tcl `file` command is not accessible inside safe interpreters. For example

```
set outfile [OOMMFRootDir]/data/myoutput
```

In this context one should always use Tcl path conventions. In particular, use forward slashes, "/", to separate directories.

Parameter The Oxs batch mode interface (Sec. 7.2) allows specified variables in the MIF file to be set from the command line. This behavior is enabled inside the MIF file via the `Parameter` command:

`Parameter` *varname optional_default_value*

Here *varname* is the name of a variable that may be set from the command line. If it is not set on the command line then the variable is set to the optional default value, if any, or otherwise an error is raised. An error is also raised if a variable set on the command line does not have a corresponding `Parameter` command in the MIF file. See also the notes on variable substitution (Sec. 17.1.3) below.

Random Returns a pseudo-random number in the interval $[0, 1]$, using a C-library random number generator. This random number generator is specified by the `OMF_RANDOM` macro in the `ocport.h` file found in the system-specific subdirectory of `oommf/pkg/oc/`. The standard Tcl `expr rand()` command is also available.

RandomSeed Initializes both the Tcl and the C-library random number generators. If no parameter is given, then a seed is drawn from the system clock. Otherwise, one integer parameter may be specified to be used as the seed.

Report Intended primarily as a MIF debugging aid, `Report` takes one string argument that is printed to the solver interface console and the Oxs log file. It is essentially a replacement for the standard Tcl `puts` command, which is not available in safe interpreters.

ReadFile The Tcl `read` command is absent from safe interpreters. The `ReadFile` command is introduced as a replacement available in "custom" and "unsafe" interpreters. `ReadFile` takes two arguments, the file to be read and an optional translation specification. The file may either be specified with an absolute path, i.e., one including all its directory components, or with a relative path interpreted with respect to the directory containing the MIF file. The `OOMMFRootDir` command can be used to advantage to locate files in other parts of the OOMMF directory tree.

The translation specification should be one of `binary`, `auto` (the default), or `image`. The first two translation modes provide the functionality of the `-translation` option of the Tcl `fconfigure` command. Refer to the Tcl documentation for details. Specifying `image` causes the input file to be filtered through the `any2ppm` program and converted into a PPM P3 text file, minus the leading "P3". In particular, after conversion the first 3 whitespace separated values are image width, height and maxvalue, followed by an array of 3 × width × height values, where each triplet corresponds to the red, green and blue components of an image pixel, sequenced in normal English reading order. This output contains no comments, and may be treated directly as a Tcl list.

The return value from the `ReadFile` command is a string corresponding to the contents of the (possibly translated) file. For example,

```
eval [ReadFile extra_mif_commands.tcl]
```

can be used to embed a separate Tcl file into a MIF 2.1 file.

Here's a more complicated example that uses a color image file to create a vector field:

```
set image [ReadFile sample.gif image]
set imagewidth [lindex $image 0]
set imageheight [lindex $image 1]
set maxval [expr {double([lindex $image 2])}]

set colorimage {}
foreach {r g b} [lrange $image 3 end] {
   set r [expr {$r/$maxval}]
   set g [expr {$g/$maxval}]
   set b [expr {$b/$maxval}]
   lappend colorimage $r $g $b
   # colorimage is a list of normalized red, green,
   # and blue values.
}
unset image

proc ColorField { rel_x rel_y rel_z } {
   # Returns the normalized (r,g,b) triple corresponding
   # to position (rel_x,rel_y) in the image; Import rel_z
   # is not used.
   global colorimage imagewidth imageheight
   set i [expr {int(floor(double($rel_x)*$imagewidth))}]
   if {$i>=$imagewidth} {set i [expr {$imagewidth-1}]}
   set j [expr {int(floor(double(1-$rel_y)*$imageheight))}]
   if {$j>=$imageheight} {set j [expr {$imageheight-1}]}
   set red_index [expr {3*($j*$imagewidth+$i)}]
```

```
   set blue_index [expr {$red_index+2}]
   return [lrange $colorimage $red_index $blue_index]
}

Specify Oxs_ScriptVectorField:sample {
   atlas :atlas
   norm 1.0
   script ColorField
}
```

The `ReadFile` command is not available if the `MIFinterp safety` option is set to `safe` in the `options.tcl` customization file (Sec. 2.3.2).

Schedule The `Schedule` command is used to setup outputs from the MIF file. This functionality is critical for solvers running in batch mode, but is also useful for setting up default connections in interactive mode.

The syntax for the `Schedule` command is

```
Schedule <outname> <desttag> <event> <frequency>
```

The `Schedule` command mirrors the interactive output scheduling provided by the **Oxsii** and **Boxsi** graphical interfaces (Sec. 7). The first parameter to the `Schedule` command is the name of the output being scheduled. These names are the same as those appearing in the "Outputs" list in the Oxs graphical interfaces, for example "DataTable" or "Oxs_CubicAnisotropy:Nickel:Field." Aside from the DataTable output which is always present, the *outname*'s are MIF file dependent.

The second parameter to the `Schedule` command is a destination tag. This is a tag associated to a running application by a previous `Destination` command (see above). The symbolic destination tag replaces the application:OID nomenclature used in the graphical interface, because in general it is not possible to know the OOMMF ID for application instances at the time the MIF file is composed. In fact, some of the applications may be launched by `Destination` commands, and so don't even have OID's at the time the `Destination` command is processed.

The *event* parameter should be one of the keywords `Step` or `Stage`, and the *frequency* parameter should be a positive integer, representing with what frequency of the specified event should output be dispatched. For example, if `Step 5` is given, then on every fifth step of the solver output of the indicated type will be sent to the selected destination. Set *frequency* to 1 to send output each time the event occurs.

All of the parameters to the `Schedule` command are required.

There are examples of scheduling with the `Destination` and `Schedule` commands in the sample MIF 2.1 file presented in Fig. 5 (Sec. 17.1.4, pages 141–144). There, three destinations are tagged. The first refers to a possibly already running instance of

mmGraph, having nickname Hysteresis. The associated `Schedule` command sends DataTable output to this application at the end of each Stage, so hysteresis graphs can be produced. The second destination tag references a different copy of **mmGraph** that will be used for monitoring the run. To make sure that this output is rendered onto a blank slate, the `new` keyword is used to launch a fresh copy of **mmGraph**. The `Schedule` command for the monitor destination delivers output to the monitoring **mmGraph** every 5 iterations of the solver. The last `Destination` command tags an arbitrary **mmArchive** application, which is used for file storage of both DataTable results at the end of each stage, and snapshots of the magnetization at the end of every third stage.

17.1.2 Specify Conventions

The Specify blocks in the input MIF file determine the collection of `Oxs_Ext` objects defining the Oxs simulation. As explained above, the `Specify` command takes two arguments, the name of the `Oxs_Ext` object to create, and an initialization string. The format of the initialization string can be arbitrary, as determined by the author of the `Oxs_Ext` class. This section presents a number of recommended conventions which `Oxs_Ext` class authors are encouraged to follow. Any `Oxs_Ext` classes that don't follow these conventions should make that fact explicitly clear in their documentation. Details on the standard `Oxs_Ext` classes included with OOMMF can be found in the Oxs documentation (Sec. 7).

17.1.2.1 Initialization string format Consider again the simple Specify block presented above:

```
Specify Oxs_EulerEvolve:foo {
  alpha 0.5
  start_dm 0.01
}
```

The first convention is that the initialization string be structured as a Tcl list with an even number of elements, with consecutive elements consisting of a label + value pairs. In the above example, the initialization string consists of two label + value pairs, "alpha 0.5" and "start_dm 0.01". The first specifies that the damping parameter α in the Landau-Lifshitz ODE is 0.5. The second specifies the initial step size for the integration routine. Interested parties should refer to a Tcl programming reference (e.g., [16]) for details on forming a proper Tcl list, but in this example the list as a whole is set off with curly braces ("`{`" and "`}`"), and individual elements are white space delimited. Generally, the ordering of the label + value pairs in the initialization string is irrelevant, i.e., `start_dm 0.01` could equivalently precede `alpha 0.5`.

Sometimes the value portion of a label + value pair will itself be a list, as in this next example:

```
Specify Oxs_BoxAtlas:myatlas {
        ...
}

Specify Oxs_RectangularMesh:mymesh {
    cellsize { 5e-9 5e-9 5e-9 }
    atlas Oxs_BoxAtlas:myatlas
}
```

Here the value associated with "cellsize" is a list of 3 elements, which declare the sampling rate along each of the coordinate axes, in meters. (`Oxs_BoxAtlas` is a particular type of `Oxs_Atlas`, and "..." mark the location of the `Oxs_BoxAtlas` initialization string, which is omitted because it is not pertinent to the present discussion.)

17.1.2.2 `Oxs_Ext` referencing The "atlas" value in the mesh Specify block of the preceding example refers to an earlier `Oxs_Ext` object, "Oxs_BoxAtlas:myatlas". It frequently occurs that one `Oxs_Ext` object needs access to another `Oxs_Ext` object. In this example the mesh object `:mymesh` needs to query the atlas object `:myatlas` in order to know the extent of the space that is to be gridded. The atlas object is defined earlier in the MIF input file by its own, separate, top-level Specify block, and the mesh object refers to it by simply specifying its name. Here the full name is used, but the short form `:myatlas` would suffice, provided no other `Oxs_Ext` object has the same short name.

Alternatively, the `Oxs_RectangularMesh` object could define an `Oxs_BoxAtlas` object inline:

```
Specify Oxs_RectangularMesh:mymesh {
    atlas {
        Oxs_BoxAtlas {
            ...
        }
    }
    cellsize { 5e-9 5e-9 5e-9 }
}
```

In place of the name of an external atlas object, a two item list is provided consisting of the type of object (here `Oxs_BoxAtlas`) and the corresponding initialization string. The initialization string is provided as a sublist, with the same format that would be used if that object were initialized via a separate Specify block.

More commonly, embedded `Oxs_Ext` objects are used to initialize spatially varying quantities. For example,

```
Specify Oxs_UniaxialAnisotropy {
  axis { Oxs_RandomVectorField {
           min_norm 1
```

```
          max_norm 1
        }
  }
  K1  { Oxs_UniformScalarField { value 530e3 } }
}
```

The magneto-crystalline anisotropy class `Oxs_UniaxialAnisotropy` supports cellwise varying K1 and anisotropy axis directions. In this example, the anisotropy axis directions are randomly distributed. To initialize its internal data structure, `Oxs_UniaxialAnisotropy` creates a local `Oxs_RandomVectorField` object. This object is also a child of the `Oxs_Ext` hierarchy, which allows it to be constructed using the same machinery invoked by the `Specify` command. However, it is known only to the enclosing `Oxs_UniaxialAnisotropy` object, and no references to it are possible, either from other Specify blocks or even elsewhere inside the same initialization string. Because it cannot be referenced, the object does not need an instance name. It does need an initialization string, however, which is given here as the 4-tuple "min_norm 1 max_norm 1". Notice how the curly braces are nested so that this 4-tuple is presented to the `Oxs_RandomVectorField` initializer as a single item, while "Oxs_RandomVectorField" and the associated initialization string are wrapped up in another Tcl list, so that the value associated with "axis" is parsed at that level as a single item.

The value associated with "K1" is another embedded `Oxs_Ext` object. In this particular example, K1 is desired uniform (homogeneous) throughout the simulation region, so the trivial `Oxs_UniformScalarField` class is used for initialization (to the value 530×10^3 J/m^3). In the case of uniform fields, scalar or vector, a shorthand notation is available that implicitly supplies a uniform `Oxs_Ext` field class:

```
Specify Oxs_UniaxialAnisotropy {
  axis { 1 0 0 }
  K1  530e3
}
```

which is equivalent to

```
Specify Oxs_UniaxialAnisotropy {
  axis { Oxs_UniformVectorField {
           vector { 1 0 0 }
         }
  }
  K1  { Oxs_UniformScalarField { value 530e3 } }
}
```

While embedding `Oxs_Ext` objects inside Specify blocks can be convenient, it is important to remember that such objects are not available to any other `Oxs_Ext` object—only objects declared via top-level Specify blocks may be referenced from inside other Specify blocks.

Also, embedded `Oxs_Ext` objects cannot directly provide user output. Furthermore, the only `Oxs_Energy` energy objects included in energy and field calculations are those declared via top-level Specify blocks. For this reason `Oxs_Energy` terms are invariably created via top-level Specify blocks, and not as embedded objects.

17.1.2.3 Grouped lists As noted earlier, sometimes the value portion of a label + value pair will be a list. Some Oxs objects support *grouped lists*, which provide a type of run-length encoding for lists. Consider the sample list

```
{ 1.1  1.2  1.2  1.2  1.2  1.3 }
```

In a grouped list the middle run of 1.2's may be represented as a sublist with a repeat count of 4, like so

```
{ 1.1  { 1.2  4 } 1.3 :expand: }
```

Here the `:expand:` keyword, when appearing as the last element of the top level list, enables the group expansion mechanism. Any preceding element, such as `{ 1.2 4 }`, that 1) is a proper sublist, and 2) has a positive integer as the last element, is treated as a grouped sublist with repeat count given by the last element. No element of the top-level list is ever interpreted as a repeat count. For example, the short form of the list

```
{ 1e-9 1e-9 1e-9 1e-9 1e-9 1e-9 }
```

is

```
{ { 1e-9 6 } :expand: }
```

Note the additional level of brace grouping. Grouped lists may also be nested, as in this example

```
{ 5.0 { 5.1 { 5.2 3 } 5.3 2 } :expand: }
```

which is equivalent to

```
{ 5.0  5.1  5.2  5.2  5.2  5.3  5.1  5.2  5.2  5.2  5.3 }
```

There are some difficulties with this mechanism when the list components are strings, such as filenames, that may contain embedded spaces. For example, consider the list

```
{ "file 3" "file 3" "5 file" }
```

If we tried to write this as

```
{ { "file 3" 2 } "5 file" :expand: }
```

we would find that, because of the nested grouping rules, this grouped list gets expanded into

```
{ file file file file file file "5 file" }
```

Here the trailing "3" in "file 3" is interpreted as a repeat count. Following normal Tcl rules, the double quotes are treated as equivalents to braces for grouping purposes. However, the keyword `:noexpand:` may be used to disable further expansion, like so

```
{ { {"file 3" :noexpand:} 2 } "5 file" :expand: }
```

The `:noexpand:` keyword placed at the end of a list disables all group expansion in that list. Although it is an unlikely example, if one had a flat, i.e., non-grouped list with last element ":expand:", then one would have to disable the grouping mechanism that would otherwise be invoked by appending `:noexpand:` to the list. In flat lists generated by program code, it is recommended to append `:noexpand:` just to be certain that the list is not expanded.

As a matter of nomenclature, standard (i.e., flat) lists and single values are also considered grouped lists, albeit trivial ones. Any Oxs object that accepts grouped lists in its Specify block should explicitly state so in its documentation.

17.1.2.4 Comments The standard Tcl commenting mechanism treats all text running from an initial `#` symbol through to the end of a line as a comment. You may note in the above examples that newlines are treated the same as other whitespace inside the curly braces delimiting the `Specify` initialization string. Because of this and additional reasons, Tcl comments cannot be used inside Specify blocks. Instead, by convention any label + value pair where label is "comment" is treated as a comment and thrown away. For example:

```
Specify Oxs_UniaxialAnisotropy {
  axis { 1 0 0 }
  comment {K1  4500e3}
  K1 530e3
  comment { 530e3 J/m^3 is nominal for Co }
}
```

Pay attention to the difference between "comment" used here as the label portion of a label + value pair, and the MIF extension command "Ignore" used outside Specify blocks. In particular, `Ignore` takes an arbitrary number of arguments, but the value element associated with a comment label must be grouped as a single element, just as any other value element.

17.1.2.5 Attributes Sometimes it is convenient to define label + value pairs outside a particular Specify block, and then import them using the "attributes" label. For example:

```
Specify Oxs_LabelValue:probdata {
  alpha 0.5
  start_dm 0.01
}
Specify Oxs_EulerEvolve {
  attributes :probdata
}
```

The Oxs_LabelValue object is an Oxs_Ext class that does nothing except hold label + value pairs. The "attributes" label acts as an include statement, causing the label + value pairs contained in the specified Oxs_LabelValue object to be embedded into the enclosing Specify initialization string. This technique is most useful if the label + value pairs in the Oxs_LabelValue object are used in multiple Specify blocks, either inside the same MIF file, or across several MIF files into which the Oxs_LabelValue block is imported using the ReadFile MIF extension command.

17.1.2.6 User defined support procedures A number of Oxs_Ext classes utilize user-defined Tcl procedures (procs) to provide extended runtime functionality. The most common examples are the various field initialization script classes, which call a user specified Tcl proc for each point in the simulation discretization mesh. The proc returns a value, either scalar or vector, which is interpreted as some property of the simulation at that point in space, such as saturation magnetization, anisotropy properties, or an external applied field.

Here is an example proc that may be used to set the initial magnetization configuration into an approximate vortex state, with a central core in the positive z direction:

```
proc Vortex { x_rel y_rel z_rel } {
   set xrad [expr {$x_rel-0.5}]
   set yrad [expr {$y_rel-0.5}]
   set normsq [expr {$xrad*$xrad+$yrad*$yrad}]
   if {$normsq <= 0.0125} {return "0 0 1"}
   return [list [expr {-1*$yrad}] $xrad 0]
}
```

The return value in this case is a 3D vector representing the spin direction at the point (x_rel,y_rel,z_rel). Procs that are used to set scalar properties, such as saturation magnetization M_s, return a scalar value instead. But in both cases, the import argument list specifies a point in the simulation mesh.

In the above example, the import point is specified relative to the extents of the simulation mesh. For example, if x_rel were 0.1, then the x-coordinate of the point is one tenth of the way between the minimum x value in the simulation and the maximum x value. In all cases x_rel will have a value between 0 and 1.

In most support proc examples, relative coordinates are the most flexible and easiest representation to work with. However, by convention, scripting Oxs_Ext classes also support absolute coordinate representations. The representation used is selected in the Oxs_Ext object Specify block by the optional script_args entry. The Tcl proc itself is specified by the script entry, as seen in this example:

```
Specify ScriptScalarField:Ms {
   atlas :atlas
   script_args { rawpt }
   script SatMag
```

```
}
proc SatMag { x y z } {
   if {$z < 20e-9} {return 8e5}
   return 5e5
}
```

The value associated with the label `script_args` should in this case be a subset of {`relpt rawpt minpt maxpt span`}. Each selection represents a triple of values that are to be included in the proc argument list, in the order specified. The `relpt` selection provides `x_rel, y_rel, z_rel` as discussed above. Conversely, `rawpt` provides the point representation in problem coordinates, i.e., in meters. The `minpt` and `maxpt` options list the minimum and maximum values in the simulation mesh in problem coordinates, i.e., `minpt` $\leq$ `rawpt` $\leq$ `maxpt` in each coordinate. The `span` option provides the 3-vector resulting from (`maxpt` $-$ `minpt`). All `script_args` arguments except `relpt` are in meter units. If `script_args` is not specified then the effective default is `relpt`. Some `Oxs_Ext` objects may support a different list of allowed `script_args` values. Check the documentation of the `Oxs_Ext` object in question for details. Please note that the names used in the proc argument lists above are for exposition purposes only. You may use other names as you wish. It is the order of the arguments that is important, not their names. Also, because the MIF file is parsed first in toto before the Specify blocks are evaluated, the support procs may be placed anywhere in the MIF file, regardless of the location of the referencing Specify blocks.

The command call to the Tcl support proc is actually built up by appending to the `script` value the arguments as specified by the `script_args` value. This allows additional arguments to the Tcl proc to be specified in the `script` value, in which case they will appear in the argument list in front of the `script_args` values. The following is equivalent to the preceding example:

```
Specify ScriptScalarField:Ms {
   atlas :atlas
   script_args { rawpt }
   script {SatMag 20e-9 8e5 5e5}
}
proc SatMag { zheight Ms1 Ms2 x y z } {
   if {$z < $zheight} {return $Ms1}
   return $Ms2
}
```

Notice in this case that the `script` value is wrapped in curly braces so that the string `SatMag 20e-9 8e5 5e5` will be treated as the single value associated with the label `script`.

As seen in the earlier example using the `Vortex` Tcl proc, support procedures in MIF 2.1 files will frequently make use of the Tcl `expr` command. If you are using Tcl version 8.0 or later, then the cpu time required by the potentially large number of calls to such procedures can be greatly reduced by grouping the arguments to `expr` commands in curly braces, as

illustrated in the `Vortex` example. The braces aid the operation of the Tcl bytecode compiler, although there are a few rare situations involving multiple substitution where such bracing cannot be applied. See the Tcl documentation for the `expr` command for details.

Sometimes externally defined data can be put to good use inside a Tcl support proc, as in this example:

```
# Lay out a 6 x 16 mask, at global scope.
set mask {
   1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
   1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
   1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
   1 1 1 1 1 0 0 0 0 0 0 1 1 1 1 1
   1 1 1 1 1 0 0 0 0 0 0 1 1 1 1 1
   1 1 1 1 1 0 0 0 0 0 0 1 1 1 1 1
}
proc MyShape { xrel yrel znotused } {
   global mask  ;# Make mask accessible inside proc
   set Ms 8e5   ;# Saturation magnetization of element
   set xindex [expr {int(floor($xrel*16))}]
   set yindex [expr {5 - int(floor($yrel*6))}]
   set index [expr {$yindex*16+$xindex}]
   # index references point in mask corresponding
   # to (xrel,yrel)
   return [expr {[lindex $mask $index]*$Ms}]
}
```

The variable `mask` holds a Tcl list of 0's and 1's defining a part shape. The mask is brought into the scope of the `MyShape` proc via the Tcl `global` command. The relative x and y coordinates are converted into an index into the list, and the proc return value is either 0 or 8e5 depending on whether the corresponding point in the mask is 0 or 1. This is essentially the same technique used in the `ColorField` proc example presented in the `ReadFile` MIF extension command documented above (Sec. 17.1.1), except that there the data structure values are built from a separate image file rather than from data embedded inside the MIF file.

17.1.3 Variable Substitution

One powerful consequence of the evaluation of MIF 2.1 input files by Tcl is the ability to define and use variables. For example, the batch mode interface to Oxs (Sec. 7.2) uses the Parameter command to set variables from the command line for use inside the MIF input file. Variables in Tcl are evaluated (i.e., value substituted) by prefixing the variable name with the symbol "$". For example, if `cellsize` is a variable holding the value `5e-9`, then `$cellsize` evaluates to `5e-9`.

Unfortunately, there are complications in using variables inside Specify blocks. Consider this simple example:

```
Parameter cellsize 5e-9
Specify Oxs_RectangularMesh:BadExample {
  comment {NOTE: THIS DOESN'T WORK!!!}
  cellsize {$cellsize $cellsize $cellsize}
  atlas :atlas
}
```

This **doesn't work**, because the curly braces used to set off the `Specify` initialization string also inhibit variable substitution. There are several ways to work around this, but the easiest is usually to embed the initialization string inside a `subst` (substitution) command:

```
Parameter cellsize 5e-9
Specify Oxs_RectangularMesh:GoodExample [subst {
  comment {NOTE: This works.}
  cellsize {$cellsize $cellsize $cellsize}
  atlas :atlas
}]
```

Here the square brackets, "`[`" and "`]`", cause Tcl to perform *command substitution*, i.e., execute the string inside the square brackets as a Tcl command, in this case the `subst` command. See the Tcl documentation for `subst` for details, but the default usage illustrated above performs variable, command and backslash substitutions on the argument string.

One more example, this time involving both variable and command substitution:

```
set pi [expr {4*atan(1.0)}]
set mu0 [expr {4*$pi*1e-7}]
Specify Oxs_UZeeman [subst {
  comment {Set units to mT}
  Hscale [expr {0.001/$mu0}]
  Hrange {
     {  0  0  0   10  0  0   2 }
     { 10  0  0  -10  0  0   2 }
  }
}]
```

Note that the `subst` command is evaluated at global scope, so that the global variable `mu0` is directly accessible.

17.1.4 Sample MIF 2.1 File

```
# MIF 2.1
#
```

```
# All units are SI.
#
# This file must be a valid Tcl script.
#

# Initialize random number generators with seed=1
RandomSeed 1

# Individual Oxs_Ext objects are loaded and initialized via
# Specify command blocks.  The following block defines the
# extents (in meters) of the volume to be modeled.  The
# prefix "Oxs_BoxAtlas" specifies the type of Oxs_Ext object
# to create, and the suffix ":WorldAtlas" is the name
# assigned to this particular instance.  Each object created
# by a Specify command must have a unique full name (here
# "Oxs_BoxAtlas:WorldAtlas").  If the suffix is not
# explicitly given, then the default ":" is automatically
# assigned.  References may be made to either the full name,
# or the shorter suffix instance name (here ":WorldAtlas")
# if the latter is unique. See the Oxs_TimeDriver block for
# some reference examples.
Specify Oxs_BoxAtlas:WorldAtlas {
  xrange {0 500e-9}
  yrange {0 250e-9}
  zrange {0 10e-9}
}

# The Oxs_RectangularMesh object is initialized with the
# discretization cell size (in meters).
Specify Oxs_RectangularMesh:mesh {
  cellsize {5e-9 5e-9 5e-9}
  atlas :WorldAtlas
}

# Magnetocrystalline anisotropy block.   The setting for
# K1 (500e3 J/m^3) implicitly creates an embedded
# Oxs_UniformScalarField object.  Oxs_RandomVectorField
# is an explicit embedded Oxs_Ext object.
Specify Oxs_UniaxialAnisotropy {
  K1  530e3
  axis { Oxs_RandomVectorField {
           min_norm 1
```

```
           max_norm 1
  } }
}

# Homogeneous exchange energy.  A = 13e-12 J/m.
Specify Oxs_UniformExchange:NiFe {
  A  13e-12
}

# Define a couple of constants for later use.
set PI [expr {4*atan(1.)}]
set MU0 [expr {4*$PI*1e-7}]

# The Oxs_UZeeman class is initialized with field ranges
# in A/m.  The following block uses the multiplier option to
# allow ranges to be specified in mT.  Use the "subst"
# command to enable variable and command substitution.
Specify Oxs_UZeeman:AppliedField [subst {
  multiplier [expr 0.001/$MU0]
  Hrange {
    {  0  0  0   10  0  0   2 }
    { 10  0  0  -10  0  0   2 }
    {  0  0  0    0 10  0   4 }
    {  1  1  1    5  5  5   0 }
  }
}]

# Enable demagnetization (self-magnetostatic) field
# computation.  This block takes no parameters.
Specify Oxs_Demag {}

# First order Euler ODE solver, using default
# parameter values.
Specify Oxs_EulerEvolve {}

# The following procedure is used to set the initial spin
# configuration in the Oxs_TimeDriver block.  The arguments
# x, y, and z are coordinates relative to the min and max
# range of each dimension, e.g., 0<=x<=1, where x==0
# corresponds to xmin, x==1 corresponds to xmax.
proc UpDownSpin { x y z } {
  if { $x < 0.45 } {
```

```
    return "0 1 0"
  } elseif { $x > 0.55 } {
    return "0 -1 0"
  } else {
    return "0 0 1"
  }
}

Specify Oxs_TimeDriver {
 evolver Oxs_EulerEvolve
 stopping_dm_dt 0.01
 mesh :mesh
 Ms 8e5   comment {implicit Oxs_UniformScalarField object}
 m0 { Oxs_ScriptVectorField {
        script {UpDownSpin}
        norm  1
        atlas :WorldAtlas
 } }
 basename example
 scalar_output_format "%#.8g"
 vector_field_output_format {binary 4}
}

# Default outputs
Destination hystgraph mmGraph:Hysteresis
Destination monitor   mmGraph   new
Destination archive   mmArchive

Schedule DataTable hystgraph Stage 1
Schedule DataTable monitor   Step 5
Schedule DataTable archive   Stage 1
Schedule Oxs_TimeDriver::Magnetization archive Stage 3
```

Figure 5: Example MIF 2.1 file.

17.2 MIF 1.1

The MIF 1.1 format is an older micromagnetic problem specification format used by the 2D solver. It is completely incompatible with the MIF 2.1 format used by the 3D solver. However, the command line tool **mifconvert** (Sec. 16.7) may be used as a conversion aid.

A sample MIF 1.1 file is presented in Fig. 6. The first line of a MIF file must be of the form "# MIF x.y", where x.y represents the format revision number. (The predecessor MIF 1.0 format was not included in any released version of OOMMF.)

After the format identifier line, any line ending in a backslash, '\', is joined to the succeeding line before any other processing is performed. Lines beginning with a '#' character are comments and are ignored. Blank lines are also ignored.

All other lines must consist of a *Record Identifier* followed by a parameter list. The Record Identifier is separated from the parameter list by one or more ':' and/or '=' characters. Whitespace and case is ignored in the Record Identifier field.

The parameter list must be a proper Tcl list. The parameters are parsed (broken into separate elements) following normal Tcl rules; in short, items are separated by whitespace, except as grouped by double quotes and curly braces. The grouping characters are removed during parsing. Any '#' character that is found outside of any grouping mechanism is interpreted as a comment start character. The '#' and all following characters on that line are interpreted as a comment.

Order of the records in a MIF 1.1 file is unimportant, except as explicitly stated below. If two or more lines contain the same Record Identifier, then the last one takes precedence, with the exception of Field Range records, of which there may be several active. All records are required unless listed as optional. Some of these record types are not supported by **mmProbEd**, however you may modify a MIF 1.1 file using any plain text editor and supply it to **mmSolve2D** (Sec. 10.1) using **FileSource** (Sec. 9).

For convenience, the Record Identifier tags are organized into several groups; these groups correspond to the top-level buttons presented by **mmProbEd**. We follow this convention below.

17.2.1 Material parameters

- **# Material Name:** This is a convenience entry for **mmProbEd**; inside the MIF 1.1 file it is a comment line. It relates a symbolic name (e.g., Iron) to specific values to the next 4 items. Ignored by solvers.
- **Ms:** Saturation magnetization in A/m.
- **A:** Exchange stiffness in J/m.
- **K1:** Crystalline anisotropy constant in J/m^3. If $K1 > 0$, then the anisotropy axis (or axes) is an easy axis; if $K1 < 0$ then the anisotropy axis is a hard axis.
- **Anisotropy Type:** Crystalline anisotropy type; One of `<uniaxial|cubic>`.
- **Anisotropy Dir1:** Directional cosines of first crystalline anisotropy axis, taken with respect to the coordinate axes (3 numbers). Optional; Default is 1 0 0 (x-axis).
- **Anisotropy Dir2:** Directional cosines of second crystalline anisotropy axis, taken with respect to the coordinate axes (3 numbers). Optional; Default is 0 1 0 (y-axis).

For uniaxial materials it suffices to specify only Anisotropy Dir1. For cubic materials one must also specify Anisotropy Dir2; the third axis direction will be calculated as the cross product of the first two. The anisotropy directions will be automatically normalized if necessary, so for example 1 1 1 is valid input (it will be modified to .5774 .5774 .5774). For cubic materials, Dir2 will be adjusted to be perpendicular to Dir1 (by subtracting out the component parallel to Dir1).

- **Anisotropy Init:** Method to use to set up directions of anisotropy axes, as a function of spatial location; This is a generalization of the Anisotropy Dir1/2 records. The value for this record should be one of `<Constant|UniformXY|UniformS2>`. `Constant` uses the values specified for Anisotropy Dir1 and Dir2, with no dispersion. `UniformXY` ignores the values given for Anisotropy Dir1 and Dir2, and randomly varies the anisotropy directions uniformly in the xy-plane. `UniformS2` is similar, but randomly varies the anisotropy directions uniformly on the unit sphere (S^2). This record is optional; the default value is `Constant`.

- **Edge K1:** Anisotropy constant similar to crystalline anisotropy constant K1 described above, but applied only along the edge surface of the part. This is a uniaxial anisotropy, directed along the normal to the boundary surface. Units are J/m^3, with positive values making the surface normal an easy axis, and negative values making the surface an easy plane. The default value for Edge K1 is 0, which disables the term.

- **Do Precess:** If 1, then enable the precession term in the Landau-Lifshitz ODE. If 0, then do pure damping only. (Optional; default value is 1.)

- **Gyratio:** The Landau-Lifshitz gyromagnetic ratio, in m/(A.s). This is optional, with default value of 2.21×10^5. See the discussion of the Landau-Lifshitz ODE under the Damp Coef record identifier description.

- **Damp Coef:** The ODE solver in OOMMF integrates the Landau-Lifshitz equation [7, 9], written as

$$\frac{d\mathbf{M}}{dt} = -|\bar{\gamma}|\,\mathbf{M} \times \mathbf{H}_{\text{eff}} - \frac{|\bar{\gamma}|\alpha}{M_s}\,\mathbf{M} \times (\mathbf{M} \times \mathbf{H}_{\text{eff}})\,,$$

  where

  $\bar{\gamma}$ is the Landau-Lifshitz gyromagnetic ratio (m/(A·s)),

  α is the damping coefficient (dimensionless).

  (Compare to (2), page 54.) Here α is specified by the "Damp Coef" entry in the MIF 1.1 file. If not specified, a default value of 0.5 is used, which allows the solver to converge in a reasonable number of iterations. Physical materials will typically have a damping coefficient in the range 0.004 to 0.15. The 2D solver engine **mmSolve** (Sec. 10) requires a non-zero damping coefficient.

17.2.2 Demag specification

- **Demag Type:** Specify algorithm and demagnetization kernel used to calculate self-magnetostatic (demagnetization) field. Must be one of
  - **ConstMag:** Calculates the *average* field in each cell under the assumption that the magnetization is constant in each cell, using formulae from [12]. (The other demag options calculate the field at the center of each cell.)
  - **3dSlab:** Calculate the in-plane field components using offset blocks of constant (volume) charge. Details are given in [3]. Field components parallel to the z-axis are calculated using squares of constant (surface) charge on the upper and lower surfaces of the sample.
  - **3dCharge:** Calculate the in-plane field component using rectangles of constant (surface) charge on each cell. This is equivalent to assuming constant magnetization in each cell. The z-components of the field are calculated in the same manner as for the 3dSlab approach.
  - **FastPipe:** Algorithm suitable for simulations that have infinite extent in the z-direction. This is a 2D version of the 3dSlab algorithm.
  - **None:** No demagnetization. Fastest but least accurate method. `:-}`

  All of these algorithms except FastPipe and None require that the Part Thickness (cf. the Part Geometry section) be set. Fast Fourier Transform (FFT) techniques are used to accelerate the calculations.

17.2.3 Part geometry

- **Part Width:** Nominal part width (x-dimension) in meters. Should be an integral multiple of Cell Size.
- **Part Height:** Nominal part height (y-dimension) in meters. Should be an integral multiple of Cell Size.
- **Part Thickness:** Part thickness (z-dimension) in meters. Required for all demag types except FastPipe and None.
- **Cell Size:** In-plane (xy-plane) edge dimension of base calculation cell. This cell is a rectangular brick, with square in-plane cross-section and thickness given by Part Thickness. N.B.: Part Width and Part Height should be integral multiples of Cell Size. Part Width and Part Height will be automatically adjusted slightly (up to 0.01%) to meet this condition (affecting a small change to the problem), but if the required adjustment is too large then the problem specification is considered to be invalid, and the solver will signal an error.
- **Part Shape:** Optional. Part shape in the xy-plane; must be one of the following:

– **Rectangle**
The sample fills the area specified by Part Width and Part Height. (Default.)

– **Ellipse**
The sample (or the magnetically active portion thereof) is an ellipse inscribed into the rectangular area specified by Part Width and Part Height.

– **Ellipsoid**
Similar to the Ellipse shape, but the part thickness is varied to simulate an ellipsoid, with axis lengths of Part Width, Part Height and Part Thickness.

– **Oval r**
Shape is a rounded rectangle, where each corner is replaced by a quarter circle with radius r, where $0 \leq r \leq 1$ is relative to the half-width of the rectangle.

– **Pyramid overhang**
Shape is a truncated pyramid, with ramp transition base width (overhang) specified in meters.

– **Mask filename**
Shape and thickness are determined by a bitmap file, the name of which is specified as the second parameter. The overall size of the simulation is still determined by Part Width and Part Height (above); the bitmap is spatially scaled to fit those dimensions. Note that this scaling will not be square if the aspect ratio of the part is different from the aspect ratio of the bitmap.

The given filename must be accessible to the solver application. At present the bitmap file must be in either the PPM (portable pixmap), GIF, or BMP formats. (Formats other than the PPM P3 (text) format may be handled by spawning an **any2ppm** (Sec. 16.1) subprocess.)

White areas of the bitmap are interpreted as being non-magnetic (or having 0 thickness); all other areas are assumed to be composed of the material specified in the "Material Parameters" section. Thickness is determined by the relative darkness of the pixels in the bitmap. Black pixels are given full nominal thickness (specified by the "Part Thickness" parameter above), and gray pixels are linearly mapped to a thickness between the nominal thickness and 0. In general, bitmap pixel values are converted to a thickness relative to the nominal thickness by the formula 1-(R+G+B)/(3M), where R, G and B are the magnitudes of the red, green and blue components, respectively, and M is the maximum allowed component magnitude. For example, black has R=G=B=0, so the relative thickness is 1, and white has R=G=B=M, so the relative thickness is 0.

The code does not perform a complete 3D evaluation of thickness effects. Instead, the approximation discussed in [13] is implemented.

17.2.4 Initial magnetization

- **Init Mag:** Name of routine to use to initialize the simulation magnetization directions

(as a function of position), and routine parameters, if any. Optional, with default Random. The list of routines is long, and it is easy to add new ones. See the file `maginit.cc` for details. A few of the more useful routines are:

- **Random**
  Random directions on the unit sphere. This is somewhat like a quenched thermal demagnetized state.
- **Uniform θ ϕ**
  Uniform magnetization in the direction indicated by the two additional parameters, θ and ϕ, where the first is the angle from the z-axis (in degrees), and the second is the angle from the x-axis (in degrees) of the projection onto the xy-plane.
- **Vortex**
  Fits an idealized vortex about the center of the sample.
- **avfFile filename**
  The second parameter specifies an OVF/VIO (i.e., "any" vector field) file to use to initialize the magnetization. The grid in the input file will be scaled as necessary to fit the grid in the current simulation. The file must be accessible to the intended solver application.

17.2.5 Experiment parameters

The following records specify the applied field schedule:

- **Field Range:** Specifies a range of applied fields that are stepped though in a linear manner. The parameter list should be 7 numbers, followed by optional control point (stopping criteria) specifications. The 7 required fields are the begin field Bx By Bz in Tesla, the end field Bx By Bz in Tesla, and an integer number of steps (intervals) to take between the begin and end fields (inclusive). Use as many Field Range records as necessary—they will be stepped through in order of appearance. If the step count is 0, then the end field is ignored and only the begin field is applied. If the step count is larger than 0, and the begin field is the same as the last field from the previous range, then the begin field is not repeated.

  The optional control point specs determine the conditions that cause the applied field to be stepped, or more precisely, end the simulation of the magnetization evolution for the current applied field. The control point specs are specified as *–type value* pairs. There are 3 recognized control point types: **–torque**, **–time**, and **–iteration**. If a –torque pair is given, then the simulation at the current applied field is ended when $\|\mathbf{m} \times \mathbf{h}\|$ (i.e., $\|\mathbf{M} \times \mathbf{H}\|/M_s^2$) at all spins in the simulation is smaller than the specified –torque value (dimensionless). If a –time pair is given, then the simulation at the current field is ended when the elapsed simulation time *for the current field step* reaches the specified –time value (in seconds). Similarly, an –iteration pair steps the applied field when the iteration count for the current field step reaches the –iteration

value. If multiple control point specs are given, then the applied field is advanced when any one of the specs is met. If no control point specs are given on a range line, then the **Default Control Point Spec** is used.

For example, consider the following Field Range line:

```
Field Range: 0 0 0 .05 0 0  5  -torque 1e-5 -time 1e-9
```

This specifies 6 applied field values, (0,0,0), (0.01,0,0), (0.02,0,0), ..., (0.05,0,0) (in Tesla), with the advancement from one to the next occurring whenever $\|\mathbf{m} \times \mathbf{h}\|$ is smaller than 1e-5 for all spins, or when 1 nanosecond (simulation time) has elapsed at the current field. (If –torque was not specified, then the applied field would be stepped at 1, 2, 3 4 and 5 ns in simulation time.)

This Field Range record is optional, with a default value of 0 0 0 0 0 0 0.

- **Default Control Point Spec:** List of control point *–type value* pairs to use as stepping criteria for any field range with no control point specs. This is a generalization of and replacement for the *Converge |mxh| Value* record. Optional, with default "`-torque 1e-5`."

- **Field Type:** Applied (external) field routine and parameters, if any. This is optional, with default Uniform. At most one record of this type is allowed, but the Multi type may be used to apply a collection of fields. The nominal applied field (NAF) is stepped through the Field Ranges described above, and is made available to the applied field routines which use or ignore it as appropriate.

  The following Field Type routines are available:

  - **Uniform**
    Applied field is uniform with value specified by the NAF.
  - **Ribbon relcharge x0 y0 x1 y1 height**
    Charge "Ribbon," lying perpendicular to the xy-plane. Here relcharge is the charge strength relative to Ms, and (x0,y0), (x1,y1) are the endpoints of the ribbon (in meters). The ribbon extends height/2 above and below the calculation plane. This routine ignores the NAF.
  - **Tie rfx rfy rfz x0 y0 x1 y1 ribwidth**
    The points (x0,y0) and (x1,y1) define (in meters) the endpoints of the center spine of a rectangular ribbon of width ribwidth lying in the xy-plane. The cells with sample point inside this rectangle see an applied field of (rfx,rfy,rfz), in units relative to Ms. (If the field is large, then the magnetizations in the rectangle will be "tied" to the direction of that field.) This routine ignores the NAF.

– **OneFile filename multiplier**
Read B field in from a file. Each value in the file is multiplied by the "multiplier" value on input. This makes it simple to reverse field direction (use -1 for the multiplier), or to convert H fields to B fields (use 1.256637e-6). The input file may be any of the vector field file types recognized by **mmDisp**. The input dimensions will be scaled as necessary to fit the simulation grid, with zeroth order interpolation as necessary. This routine ignores the NAF.

– **FileSeq filename procname multiplier**
This is a generalization of the OneFile routine that reads in fields from a sequence of files. Here "filename" is the name of a file containing Tcl code to be sourced during problem initialization, and "procname" is the name of a Tcl procedure defined in filename, which takes the nominal B field components and field step count values as imports (4 values total), and returns the name of the vector field file that should be used as the applied B field for that field step.

– **Multi routinecount **
**param1count name1 param1 param2 . . . **
**param2count name2 param1 param2 . . . **
. . .
Allows a conglomeration of several field type routines. All entries must be on the same logical line, i.e., end physical lines with '\' continuation characters as necessary. Here routinecount is the number of routines, and param1count is the number parameters (including name1) needed by the first routine, etc.

Note that all lengths are in meters. The coordinates in the simulation lie in the first octant, running from (0,0,0) to (Part Width, Part Height, Part Thickness).

17.2.6 Output specification

- **Base Output Filename:** Default base name used to construct output filenames.
- **Magnetization Output Format:** Format to use in the OVF (Sec. 19.1) data block for exported magnetization files. Should be one of "binary 4" (default), "binary 8", or "text *format-spec*", where *format-spec* is a C `printf`-style format code, such as "`%# .17g`". Optional.
- **Total Field Output Format:** Analogous to the Magnetization Output Format, but for total field output files. Optional, with default "binary 4".
- **Data Table Output Format:** Format to use when producing data table style scalar output, such as that sent to **mmDataTable** (Sec. 11), **mmGraph** (Sec. 12), and **mmArchive** (Sec. 14). Should specify a C `printf`-style format code, such as the default "`%.16g`". Optional.

17.2.7 Miscellaneous

- **Converge |mxh| Value:** Nominal value to use as a stopping criterion: When $\|\mathbf{m}\times\mathbf{h}\|$ (i.e., $\|\mathbf{M}\times\mathbf{H}\|/M_s^2$) at all spins in the simulation is smaller than this value, it is assumed that a relaxed (equilibrium) state has been reached for the current applied field. This is a dimensionless value.
  NOTE: This Record Identifier is deprecated. Use *Default Control Point Spec* instead.

- **Randomizer Seed:** Value with which to seed random number generator. Optional. Default value is 0, which uses the system clock to generate a semi-random seed.

- **Max Time Step:** Limit the maximum ODE step size to no larger than this amount, in seconds. Optional.

- **Min Time Step:** Limit the minimum ODE step size to no less than this amount, in seconds. Optional.

- **User Comment:** Free-form comment string that may be used for problem identification. Optional.

```
# MIF 1.1
#
# All units are SI.
#
################## MATERIAL PARAMETERS ######################
Ms:  800e3                # Saturation magnetization in A/m.
A:   13e-12               # Exchange stiffness in J/m.
K1:  0.5e3                # Anisotropy constant in J/m^3.
Anisotropy Type: uniaxial # One of <uniaxial|cubic>.
Anisotropy Dir1: 1 0 0    # Directional cosines wrt to
                          # coordinate axes

################## DEMAG SPECIFICATION ######################
Demag Type: ConstMag # One of <ConstMag|3dSlab|2dSlab
                     #         |3dCharge|FastPipe|None>.

#################### PART GEOMETRY ##########################
Part Width:     0.25e-6    # Nominal part width in m
Part Height:    1.0e-6     # Nominal part height in m
Part Thickness: 1e-9       # Part thickness in m.
Cell Size:      8.1e-9     # Cell size in m.
#Part Shape:    # One of <Rectangle|Ellipse|Oval|Mask>.
                # Optional.
```

```
################ INITIAL MAGNETIZATION ####################
Init Mag: Uniform 90 45 # Initial magnetization routine
                        # and parameters

################ EXPERIMENT PARAMETERS ####################
# Field Range:  Start_field  Stop_field  Steps
Field Range: -.05 -.01 0. .05 .01 0. 100
Field Range: .05 .01 0. -.05 -.01 0. 100
Field Type: Multi 4 \
 7 Ribbon 1 0 1.0e-6 0.25e-6 1.0e-6 1e-9 \
 7 Ribbon 1 0 0     0.25e-6 0     1e-9 \
 9 Tie 100 0 0 0.12e-6 0.5e-6 0.13e-6 0.5e-6 8.1e-9 \
 1 Uniform
# The above positions ribbons of positive charge along the
# upper and lower edges with strength Ms, applies a large
# (100 Ms) field to the center cell, and also applies a
# uniform field across the sample stepped from
# (-.05,-.01,0.) to (.05,.01,0.) (Tesla), and back, in
# approximately 0.001 T steps.

Default Control Point Spec: -torque 1e-6
# Assume equilibrium has been reached, and step the applied
# field, when the reduced torque |mxh| drops below 1e-6.

################ OUTPUT SPECIFICATIONS ####################
Base Output Filename: samplerun
Magnetization Output Format: binary 8 # Save magnetization
# states in binary format with full (8-byte) precision.

#################### MISCELLANEOUS ########################
Randomizer Seed: 1   # Random number generator seed.
User Comment: Example MIF 1.1 file, with lots of comments.
```

Figure 6: Example MIF 1.1 file.

18 Data Table File Format (ODT)

Textual output from solver applications that is not of the vector field variety is output in the *OOMMF Data Table* (ODT) format. This is an ASCII text file format, with column information in the header and one line of data per record. Any line ending in a '\' character

is joined to the succeeding line before any other processing is performed. Any leading '#' characters on the second line are removed.

As with the OVF format (Sec. 19.1), all non-data lines begin with a '#' character, comments with two '#' characters. (This makes it easier to import the data into external programs, for example, plotting packages.) An example is shown in Fig. 7.

The first line of an ODT file should be the file type descriptor

```
# ODT 1.0
```

It is also recommended that ODT files be given names ending in the file extension `.odt` so that ODT files may be easily identified.

The remaining lines of the ODT file format should be comments, data, or any of the following 5 recognized descriptor tag lines:

- **# Table Start:** Optional, used to segment a file containing multiple data table blocks. Anything after the colon is taken as an optional label for the following data block.
- **# Title:** Optional; everything after the colon is interpreted as a title for the table.
- **# Columns:** Required. One parameter per column, designating the label for that column. Spaces may be embedded in a column label by using the normal Tcl grouping mechanisms (i.e., double-quotes and braces).
- **# Units:** Optional. If given, it should have one parameter for each column, giving a unit label for the corresponding column.
- **# Table End**: Optional, no parameters. Should be paired with a corresponding Table Start record.

Data may appear anywhere after the Columns descriptor record and before any Table End line, with one record per line. The data should be numeric values separated by whitespace.

The command line utility, **odtcols** (Sec. 16.8), can be a useful tool for examining and partitioning ODT files.

19 Vector Field File Format (OVF)

Vector field files specify vector quantities (e.g., magnetization or magnetic flux density) as a function of spatial position. The *OOMMF Vector Field* (OVF) format is the output vector field file format used by both the 2D (Sec. 10) and 3D (Sec. 7) micromagnetic solvers. It is also the input data type read by **mmDisp** (Sec. 13). There are two versions of the OVF format supported by OOMMF. The OVF 1.0 format is the preferred format and the only one written by OOMMF software. It supports both rectangular and irregular meshes, in binary and ASCII text. The OVF 0.0 format (formerly SVF) is an older, simpler format that can be useful for importing vector field data into OOMMF from other programs. (A third

```
# ODT 1.0
# Table Start
# Title: This is a small sample ODT file.
#
## This is a sample comment.  You can put anything you want
## on comment lines.
#
# Columns: Iteration "Applied Field"  {Total Energy}    Mx
# Units:      {}          "mT"           "J/m^3"       "A/m"
              103          50            0.00636      787840
             1000          32            0.00603      781120
            10300       -5000            0.00640     -800e3
# Table End
```

Figure 7: Sample ODT file.

format, the *VecFil* or *Vector Input/Output* (VIO) format, was used by some precursors to the OOMMF code. Although OOMMF is able to read the VIO format, its use is deprecated.)

The recommended file extensions for OVF files are `.omf` for magnetization files, `.ohf` for magnetic field (**H**) files, `.obf` for magnetic flux density (**B**) files, or `.ovf` for generic files.

19.1 The OVF 1.0 format

A commented sample OVF 1.0 file is provided in Fig. 8. An OVF file has an ASCII header and trailer, and a data block that may be either ASCII or binary. All non-data lines begin with a '`#`' character; double '`##`' mark the start of a comment, which continues until the end of the line. There is no line continuation character. Lines starting with a '`#`' but containing only whitespace characters are ignored.

All non-empty non-comment lines in the file header are structured as label+value pairs. The label tag consists of all characters after the initial '`#`' up to the first colon (':') character. Case is ignored, and all space and tab characters are eliminated. The value consists of all characters after the first colon, continuing up to a '`##`' comment designator or the end of the line.

The first line of an OVF file should be a file type identification line, having the form

```
# OOMMF: rectangular mesh v1.0
```

or

```
# OOMMF: irregular mesh v1.0
```

where the value "rectangular mesh v1.0" or "irregular mesh v1.0" identifies the mesh type and revision. While the OVF 1.0 format was under development in earlier OOMMF releases,

the revision strings `0.99` and `0.0a0` were sometimes recorded on the file type identification line. OOMMF treats all of these as synonyms for `1.0` when reading OVF files.

The remainder of the file is conceptually broken into Segment blocks, and each Segment block is composed of a (Segment) Header block and a Data block. Each block begins with a "`# Begin: <block type>`" line, and ends with a corresponding "`# End: <block type>`" line. The number of Segment blocks is specified in the

```
# Segment count: 1
```

line. Currently only 1 segment is allowed. This may be changed in the future to allow for multiple vector fields per file. This is followed by

```
# Begin: Segment
```

to start the first segment.

19.1.1 Segment Header block

The Segment Header block start is marked by the line "`# Begin: Header`" and the end by "`# End: Header`". Everything between these lines should be either comments or one of the following file descriptor lines. They are order independent. All are required unless otherwise stated. Numeric values are floating point values unless "integer" is explicitly stated.

- **title:** Long file name or title.
- **desc:** Description line. Optional. Use as many as desired. Description lines may be displayed by postprocessing programs, unlike comment lines which are ignored by all automated processing.
- **meshunit:** Fundamental mesh spatial unit, treated as a label. The comment marker '`##`' is not allowed in this label. Example value: "nm".
- **valueunit:** Fundamental field value unit, treated as a label. The comment marker '`##`' is not allowed in this label. Example: "kA/m."
- **valuemultiplier:** Values in the data block are multiplied by this to get true values in units of "valueunit." This simplifies the use of normalized values.
- **xmin, ymin, zmin, xmax, ymax, zmax:** Six separate lines, specifying the bounding box for the mesh, in units of "meshunit." This may be used by display programs to limit the display area, and may be used for drawing a boundary frame if "boundary" is not specified.
- **boundary:** List of *(x,y,z)* triples specifying the vertices of a boundary frame. Optional.

- **ValueRangeMaxMag, ValueRangeMinMag:** The maximum and minimum field magnitudes in the data block, in the same units and scale as used in the data block. These are for optional use as hints by postprocessing programs; for example, **mmDisp** will not display any vector with magnitude smaller than ValueRangeMinMag.

- **meshtype:** Grid structure; should be either "rectangular" or "irregular." Irregular grid files should specify "pointcount" in the header; rectangular grid files should specify instead "xbase, ybase, zbase," "xstepsize, ystepsize, zstepsize," and "xnodes, ynodes, znodes."

- **pointcount:** Number of data sample points/locations, i.e., nodes (integer). For irregular grids only.

- **xbase, ybase, zbase:** Three separate lines, denoting the position of the first point in the data section, in units of "meshunit." For rectangular grids only.

- **xstepsize, ystepsize, zstepsize:** Three separate lines, specifying the distance between adjacent grid points, in units of "meshunit." Required for rectangular grids, but may be specified as a display hint for irregular grids.

- **xnodes, ynodes, znodes:** Three separate lines, specifying the number of nodes along each axis (integers). For rectangular grids only.

19.1.2 Data block

The data block start is marked by a line of the form

```
# Begin: data <representation>
```

where `<representation>` is one of "text", "binary 4", or "binary 8". Text mode uses the ASCII specification, with individual data items separated by an arbitrary amount of whitespace (spaces, tabs and newlines). Comments are not allowed inside binary mode data blocks, but are permitted inside text data blocks.

The binary representations are IEEE floating point in network byte order (MSB). To insure that the byte order is correct, and to provide a partial check that the file hasn't been sent through a non 8-bit clean channel, the first datum is a predefined value: 1234567.0 (Hex: 49 96 B4 38) for 4-byte mode, and 123456789012345.0 (Hex: 42 DC 12 21 83 77 DE 40) for 8-byte mode. The data immediately follow the check value.

The structure of the data depends on whether the "meshtype" declared in the header is "irregular" or "rectangular". For irregular meshes, each data element is a 6-tuple, consisting of the x, y and z components of the node position, followed by the x, y and z components of the field at that position. Ordering among the nodes is not relevant. The number of nodes is specified in the "pointcount" line in the segment header.

For rectangular meshes, data input is field values only, in x, y, z component triples. These are ordered with the x index incremented first, then the y index, and the z index last.

This is nominally Fortran order, and is adopted here because commonly x will be the longest dimension, and z the shortest, so this order is more memory-access efficient than the normal C array indexing of z, y, x. The size of each dimension is specified in the "xnodes, ynodes, znodes" lines in the segment header.

In any case, the first character after the last data item should be a newline, followed by

```
# End: data <representation>
```

where `<representation>` must match the value in the "Begin: data" line. This is followed by a

```
# End: segment
```

line that ends the segment, and hence the file.

Note: An OVF 1.0 file with ASCII data and irregular meshtype is also a valid OVF 0.0 (SVF) file, although as a OVF 0.0 file the value scaling as specified by "`# valueunit`" and "`# valuemultiplier`" header lines is inactive.

```
# OOMMF: rectangular mesh v1.0
#
## This is a comment.
## No comments allowed in the first line.
#
# Segment count: 1   ## Number of segments.  Should be 1 for now.
#
# Begin: Segment
# Begin: Header
#
# Title: Long file name or title goes here
#
# Desc: 'Description' tag, which may be used or ignored by postprocessing
# Desc: programs. You can put anything you want here, and can have as many
# Desc: 'Desc' lines as you want.  The ## comment marker is disabled in
# Desc: description lines.
#
## Fundamental mesh measurement unit.  Treated as a label:
# meshunit: nm
#
# meshtype: rectangular
# xbase: 0.      ## (xbase,ybase,zbase) is the position, in
# ybase: 0.      ## 'meshunit', of the first point in the data
# zbase: 0.      ## section (below).
#
# xstepsize: 20. ## Distance between adjacent grid pts.: on the x-axis,
```

```
# ystepsize: 10. ## 20 nm, etc.  The sign on this value determines the
# zstepsize: 10. ## grid orientation relative to (xbase,ybase,zbase).
#
# xnodes: 200    ## Number of nodes along the x-axis, etc. (integers)
# ynodes: 400
# znodes:   1
#
# xmin:    0.    ## Corner points defining mesh bounding box in
# ymin:    0.    ## 'meshunit'.  Floating point values.
# zmin:  -10.
# xmax: 4000.
# ymax: 4000.
# zmax:   10.
#
## Fundamental field value unit, treated as a label:
# valueunit: kA/m
# valuemultiplier: 0.79577472  ## Multiply data block values by this
#                              ## to get true value in 'valueunits'.
#
# ValueRangeMaxMag:  1005.3096  ## These are in data block value units,
# ValueRangeMinMag:  1e-8       ## and are used as hints (or defaults)
#     ## by postprocessing programs.  The mmDisp program ignores any
#     ## points with magnitude smaller than ValueRangeMinMag, and uses
#     ## ValueRangeMaxMag to scale inputs for display.
#
# End: Header
#
## Anything between '# End: Header' and '# Begin: data text',
## '# Begin: data binary 4' or '# Begin: data binary 8' is ignored.
##
## Data input is in 'x-component y-component z-component' triples,
## ordered with x incremented first, then y, and finally z.
#
# Begin: data text
1000 0 0 724.1 0. 700.023
578.5 500.4 -652.36
<...data omitted for brevity...>
252.34 -696.42 -671.81
# End: data text
# End: segment
```

Figure 8: Commented OVF sample file.

19.2 The OVF 0.0 format

The OVF 0.0 format is a simple ASCII text format supporting irregularly sampled data. It is intended as an aid for importing data from non-OOMMF programs, and is backwards compatible with the format used for problem submissions for the first μMAG standard problem[14].

Users of early releases of OOMMF may recognize the OVF 0.0 format by its previous name, the Simple Vector Field (SVF) format. It came to the attention of the OOMMF developers that the file extension `.svf` was already registered in several MIME systems to indicate the Simple Vector Format[15], a vector graphics format. To avoid conflict, we have stopped using the name Simple Vector Field format, although OOMMF software still recognizes the `.svf` extension and you may still find example files and other references to the SVF format.

A sample OVF 0.0 file is shown in Fig. 9. Any line beginning with a '`#`' character is a comment, all others are data lines. Each data line is a whitespace separated list of 6 elements: the x, y and z components of a node position, followed by the x, y and z components of the field at that position. Input continues until the end of the file is reached.

It is recommended (but not required) that the first line of an OVF file be

```
# OOMMF: irregular mesh v0.0
```

This will aid automatic file type detection. Also, three special (extended) comments in OVF 0.0 files are recognized by **mmDisp**:

```
## File: <filename or extended filename>
## Boundary-XY: <boundary vertex pairs>
## Grid step: <cell dimension triple>
```

All these lines are optional. The "File" provides a preferred (possibly extended) filename to use for display identification. The "Boundary-XY" line specifies the ordered vertices of a bounding polygon in the xy-plane. If given, **mmDisp** will draw a frame using those points to ostensibly indicate the edges of the simulation body. Lastly, the "Grid step" line provides three values representing the average x, y and z dimensions of the volume corresponding to an individual node (field sample). It is used by **mmDisp** to help scale the display.

Note that the data section of an OVF 0.0 file takes the simple form of columns of ASCII formatted numbers. Columns of whitespace separated numbers expressed in ASCII are easy to import into other programs that process numerical datasets, and are easy to generate, so the OVF 0.0 file format is useful for exchanging vector field data between OOMMF and

[14] http://www.ctcms.nist.gov/~rdm/stdprob_1.html
[15] http://www.softsource.com/svf/

```
# OOMMF: irregular mesh v0.0
## File: sample.ovf
## Boundary-XY: 0.0 0.0 1.0 0.0 1.0 2.0 0.0 2.0 0.0 0.0
## Grid step: .25 .5 0
#  x      y      z        m_x      m_y      m_z
  0.01   0.01   0.01   -0.35537  0.93472 -0.00000
  0.01   1.00   0.01   -0.18936  0.98191 -0.00000
  0.01   1.99   0.01   -0.08112  0.99670 -0.00000
  0.50   0.50   0.01   -0.03302  0.99945 -0.00001
  0.99   0.05   0.01   -0.08141  0.99668 -0.00001
  0.75   1.50   0.01   -0.18981  0.98182 -0.00000
  0.99   1.99   0.01   -0.35652  0.93429 -0.00000
```

Figure 9: Example OVF 0.0 file.

non-OOMMF programs. Furthermore, the data section of an OVF 0.0 file is consistent with the data section of an OVF 1.0 file that has been saved as an irregular mesh using text data representation. This means that even though OOMMF software now writes only the OVF 1.0 format for vector field data, simple interchange of vector field data with other programs is still supported.

20 Troubleshooting

The OOMMF developers rely on reports from OOMMF users to alert them to problems with the software and its documentation, and to guide the selection and implementation of new features. See the Credits (Sec. 22) for instructions on how to contact the OOMMF developers.

The more complete your report, the fewer followup messages will be required to determine the cause of your problem. Usually when a problem arises there is an error message produced by the OOMMF software. A stack trace may be offered that reveals more detail about the error. When reporting an error, it will help the developers diagnose the problem if users cut and paste into their problem report the error message and stack trace exactly as reported by OOMMF software. In addition, please include a copy of the output generated by `tclsh oommf.tcl +platform` so that the OOMMF developers will know the details of your platform configuration.

Before making a report to the OOMMF developers, please check the following list of fixes for known problems:

1. When compiling (Sec. 2.2.3), there is an error about being unable to open system header files like `stdlib.h`, `time.h`, `math.h`, or system libraries and related program startup code. This usually indicates a bad compiler installation. If you running on Windows and building with the the Microsoft Visual C++ command line compiler, did you remember to run `vcvars32.bat` to set up the necessary environment variables? If you are using the Borland C++ compiler, are the `bcc32.cfg` and `ilink32.cfg` files properly configured? In all cases, check carefully any notes in the chapter Advanced Installation (Sec. 2.3) pertaining to your compiler.

2. When compiling (Sec. 2.2.3), there is an error something like:

   ```
   <30654> pimake 1.x.x.x MakeRule panic:
   Don't know how to make '/usr/include/tcl.h'
   ```

   This means the header file `tcl.h` is missing from your Tcl installation. Other missing header files might be `tk.h` from the Tk installation, or `Xlib.h` from an X Window System installation on Unix. In order to compile OOMMF, you need to have the development versions of Tcl, Tk, and (if needed) X installed. The way to achieve that is platform-dependent. On Windows you do not need an X installation, but when you install Tcl/Tk be sure to request a "full" installation, or one with "header and library files". On Linux, be sure to install developer packages (for example, the XFree86-devel RPM) as well as user packages. Other platforms are unlikely to have this problem. In the case of `Xlib.h`, it is also possible that the `tkConfig.sh` file has an incorrect entry for `TK_XINCLUDES`. A workaround for this is to add the following line to your `oommf/config/cache/`*platform* file:

   ```
   $config SetValue TK_XINCLUDES "-I/usr/X11R6/include"
   ```

Adjust the include directory as appropriate for your system.

3. When compiling (Sec. 2.2.3), there is an error indicating that exceptions are not supported.

   Parts of OOMMF are written in C++, and exceptions have been part of the C++ language for many years. If your compiler does not support them, it is time to upgrade to one that does. OOMMF 1.2 requires a compiler capable of compiling source code which uses C++ exceptions.

4. Compiling (Sec. 2.2.3) with gcc/egcs produces syntax errors on lines involving `auto_ptr` templates.

   This is known to occur on RedHat 5.2 systems. The `auto_ptr` definition in the system STL header file `memory` (located on RedHat 5.2 systems in the directory `/usr/include/g++`) is disabled by two `#if` statements. One solution is to edit this file to turn off the `#if` checks. If you do this, you will also have to fix two small typos in the definition of the `release()` member function.

5. When compiling (Sec. 2.2.3) there is an error message arising from system include directories being too early in the include search path. Try adding the offending directories to the `program_compiler_c++_system_include_path` property in the platform cache file, e.g.,

   ```
   $config SetValue program_compiler_c++_system_include_path \
         [list /usr/include /usr/local/include]
   ```

6. On Solaris, gcc reports many errors like

   ```
   ANSI C++ forbids declaration 'XSetTransientForHint' with no type
   ```

   On many Solaris systems, the header files for the X Window System are not ANSI compliant, and gcc complains about that. To work around this problem, edit the file `oommf/config/cache/solaris.tcl` to add the option `-fpermissive` to the gcc command line.

7. On Windows, when first starting `oommf.tcl`, there is an error:

   ```
   Error launching mmLaunch version 1.x.x.x:
   couldn't execute "...\omfsh.exe": invalid argument
   ```

   This cryptic message most likely means that the pre-compiled OOMMF binaries which were downloaded are for a different version of Tcl/Tk than is installed on your system. Download OOMMF again, taking care this time to retrieve the binaries that match the release of Tcl/Tk you have installed.

8. When first starting `oommf.tcl`, there is an error:

```
Error in startup script:  Neither Omf_export nor Omf_export_list
set in
```

The file `oommf/ext/net/omfExport.tcl` may be missing from your OOMMF installation. If necessary, download and install OOMMF again.

9. When launching multiple OOMMF applications on Windows 9X, there is an error:

   ```
   couldn't open socket:  no buffer space available
   ```

   OOMMF uses network sockets for communications between its various components. On Windows 9X, the socket resources are limited, and most OOMMF applications use several sockets. The only workaround is to close unneeded OOMMF applications, and any other applications that may be using network resources. The system command line utility `netstat` can be used to monitor network communications. This problem does not arise on Windows NT or Unix systems.

10. In the Cygwin environment on Windows there is an warning like:

    ```
    Tcl version mismatch:
            C:/Cygwin/lib/tclConfig.sh from 8.0p2
            Running Tcl 8.0.4
    ```

    This is caused by broken `tclConfig.sh` and `tkConfig.sh` files in the Cygwin distribution. You may ignore the warning, or edit the `tclConfig.sh` and `tkConfig.sh` files to change the `TCL_PATCH_LEVEL` and `TK_PATCH_LEVEL` values from "p2" to ".4". However, the best option is to upgrade Tcl/Tk. The stock Tcl/Tk 8.4.1 (and presumably later) release works in Cygwin.

11. I ran out of memory!

    Are you using **mmGraph** (Sec. 12) to monitor a long-running simulation? All data sent to **mmGraph** is kept in memory by default. See the **mmGraph** documentation for information on how to manage this problem.

21 References

[1] A. Aharoni, *Introduction to the Theory of Ferromagnetism* (Oxford, New York, 1996).

[2] A. Aharoni, "Demagnetizing Factors for Rectangular Ferromagnetic Prisms," J. App. Phys. **83,** 3432–3434 (1999).

[3] D. V. Berkov, K. Ramstöck, and A. Hubert, "Solving Micromagnetic Problems: Towards an Optimal Numerical Method," Phys. Stat. Sol. (a) **137,** 207–222 (1993).

[4] W. F. Brown, Jr., *Micromagnetics* (Krieger, New York, 1978).

[5] M. J. Donahue and R. D. McMichael, "Exchange Energy Representations in Computational Micromagnetics," Physica B **233,** 272–278 (1997).

[6] M. J. Donahue and D. G. Porter, "OOMMF User's Guide, Version 1.0," Technical Report No. NISTIR 6376, National Institute of Standards and Technology, Gaithersburg, MD (1999) .

[7] T. L. Gilbert, "A Lagrangian Formulation of the Gyromagnetic Equation of the Magnetization Field," Phys. Rev. **100,** 1243 (1955).

[8] P. R. Gillette and K. Oshima, "Magnetization Reversal by Rotation," J. Appl. Phys. **29,** 529–531 (1958).

[9] L. Landau and E. Lifshitz, "On the Theory of the Dispersion of Magnetic Permeability in Ferromagnetic Bodies," Physik. Z. Sowjetunion **8,** 153–169 (1935).

[10] R. D. McMichael and M. J. Donahue, "Head to Head Domain Wall Structures in Thin Magnetic Strips," IEEE Trans. Mag. **33,** 4167–4169 (1997).

[11] L. Néel, "Some Theoretical Aspects of Rock Magnetism," Adv. Phys. **4,** 191–242 (1955).

[12] A. J. Newell, W. Williams, and D. J. Dunlop, "A Generalization of the Demagnetizing Tensor for Nonuniform Magnetization," J. Geophysical Research - Solid Earth **98,** 9551–9555 (1993).

[13] D. G. Porter and M. J. Donahue, "Generalization of a Two-Dimensional Micromagnetic Model to Non-Uniform Thickness," Journal of Applied Physics **89,** 7257–7259 (2001).

[14] M. R. Scheinfein, J. Unguris, J. L. Blue, K. J. Coakley, D. T. Pierce, and R. J. Celotta, "Micromagnetics of Domain Walls at Surfaces," Phys. Rev. B **43,** 3395–3422 (1991).

[15] E. C. Stoner and E. P. Wohlfarth, "A Mechanism of Magnetic Hysteresis in Heterogeneous Alloys," Phil. Trans. Royal Soc. London **A240,** 599–642 (1948).

[16] B. B. Welch, *Practical Programming in Tcl and Tk*, 3rd ed. (Prentice Hall, Upper Saddle River, New Jersey USA, 2000).

22 Credits

The main contributors to this document are Michael J. Donahue (michael.donahue@nist.gov) and Donald G. Porter (donald.porter@nist.gov), both of ITL/NIST. Section 3 is based on notes from Dianne P. O'Leary.

The OOMMF[16] code is being developed mainly by Michael Donahue and Donald Porter. Robert D. McMichael (rmcmichael@nist.gov) made contributions to the early development of the 2D micromagnetic solver. Jason Eicke (jeicke@seas.gwu.edu) is responsible for the problem editor, and has worked on the self-magnetostatic module of the 2D micromagnetic solver.

Numerous users have contributed to the development of OOMMF by submitting bug reports, small pieces of code, or suggestions for improvements. Many thanks to all these people, including Dieter Buntinx, NgocNga Dao, Olivier Gérardin, Ping He, Michael Ho, Mansoor B. A. Jalil, Jörg Jorzick, Pierre-Olivier Jubert, Pavel Kabos, Michael Kleiber, H. T. Leung, David Lewis, Sang Ho Lim, Yi Liu, Van Luu, Andy P. Manners, Damien McGrouther, Wong Lai Mun, Edward Myers, Andrew Newell, Valentine Novosad, Andrew Perrella, Angeline Phoa, Anil Prabhakar, Robert Ravlic, Stephen E. Russek, Renat Sabirianov, Zhupei Shi, Xiaobo Tan, Stephen Thompson, Vassilios Tsiantos, Pieter Visscher, Scott L. Whittenburg, Kong Xiangyang, Tan Swee Yong, Chengtao Yu, Steven A. Zielke, and Pei Zou.

If you have bug reports, contributed code, feature requests, or other comments for the OOMMF developers, please send them in an e-mail message to <michael.donahue@nist.gov>.

Acknowledgement is appreciated if the software is used. We recommend citing the following NIST technical report:

> M. J. Donahue and D. G. Porter
> OOMMF User's Guide, Version 1.0
> Interagency Report NISTIR 6376
> National Institute of Standards and Technology, Gaithersburg, MD (Sept 1999).

and optionally include the URL of the OOMMF home page, http://math.nist.gov/oommf/. To help us keep our bibliography page[17] current, please direct publication information to <michael.donahue@nist.gov>.

[16] http://math.nist.gov/oommf/
[17] http://math.nist.gov/oommf/bibliography.html

Index