ANL-95/11 Rev 3.7

PETSc Users Manual

Revision 3.7

Mathematics and Computer Science Division

ANL-95/11 Rev 3.7

PETSc Users Manual

Revision 3.7

Prepared by
S. Balay, S. Abhyankar, M. Adams, J. Brown, P. Brune, K. Buschelman, L. Dalcin, V. Eijkhout, W. Gropp, D. Karpeyev, D. Kaushik, M. Knepley, L. Curfman McInnes, K. Rupp, B. Smith, S. Zampini, H. Zhang, and H. Zhang Mathematics and Computer Science Division, Argonne National Laboratory

April 2016

This work was supported by the Office of Advanced Scientific Computing Research,
Office of Science, U.S. Department of Energy, under Contract DE-AC02-06CH11357.

Abstract:

This manual describes the use of PETSc for the numerical solution of partial differential equations and related problems on high-performance computers. The Portable, Extensible Toolkit for Scientific Computation (PETSc) is a suite of data structures and routines that provide the building blocks for the implementation of large-scale application codes on parallel (and serial) computers. PETSc uses the MPI standard for all message-passing communication.

PETSc includes an expanding suite of parallel linear, nonlinear equation solvers and time integrators that may be used in application codes written in Fortran, C, C++, Python, and MATLAB (sequential). PETSc provides many of the mechanisms needed within parallel application codes, such as parallel matrix and vector assembly routines. The library is organized hierarchically, enabling users to employ the level of abstraction that is most appropriate for a particular problem. By using techniques of object-oriented programming, PETSc provides enormous flexibility for users.

PETSc is a sophisticated set of software tools; as such, for some users it initially has a much steeper learning curve than a simple subroutine library. In particular, for individuals without some computer science background, experience programming in C, C++ or Fortran and experience using a debugger such as `gdb` or `dbx`, it may require a significant amount of time to take full advantage of the features that enable efficient software use. However, the power of the PETSc design and the algorithms it incorporates may make the efficient implementation of many application codes simpler than "rolling them" yourself.

- For many tasks a package such as MATLAB is often the best tool; PETSc is not intended for the classes of problems for which effective MATLAB code can be written. PETSc also has a MATLAB interface, so portions of your code can be written in MATLAB to "try out" the PETSc solvers. The resulting code will not be scalable however because currently MATLAB is inherently not scalable.

- PETSc should not be used to attempt to provide a "**parallel linear solver**" in an otherwise sequential code. Certainly all parts of a previously sequential code need not be parallelized but the matrix generation portion must be parallelized to expect any kind of reasonable performance. Do not expect to generate your matrix sequentially and then "use PETSc" to solve the linear system in parallel.

Since PETSc is under continued development, small changes in usage and calling sequences of routines will occur. PETSc is supported; see the web site http://www.mcs.anl.gov/petsc for information on contacting support.

A http://www.mcs.anl.gov/petsc/publications may be found a list of publications and web sites that feature work involving PETSc.

We welcome any reports of corrections for this document.

Getting Information on PETSc:

On-line:

- Manual pages–example usage docs/index.html or http://www.mcs.anl.gov/petsc/documentation
- Installing PETSc http://www.mcs.anl.gov/petsc/documentation/installation.html

In this manual:

Acknowledgments:

We thank all PETSc users for their many suggestions, bug reports, and encouragement. We especially thank David Keyes for his valuable comments on the source code, functionality, and documentation for PETSc.

Some of the source code and utilities in PETSc have been written by

- Asbjorn Hoiland Aarrestad - the explicit Runge-Kutta implementations;
- G. Anciaux and J. Roman - the interfaces to the partitioning packages PTScotch, Chaco, and Party;
- Allison Baker - the flexible GMRES code and LGMRES;
- Chad Carroll - Win32 graphics;
- Ethan Coon - the PetscBag and many bug fixes;
- Cameron Cooper - portions of the VecScatter routines;
- Patrick Farrell - nleqerr line search for SNES
- Paulo Goldfeld - balancing Neumann-Neumann preconditioner;
- Matt Hille;
- Joel Malard - the BICGStab(l) implementation;
- Paul Mullowney, enhancements to portions of the Nvidia GPU interface;
- Dave May - the GCR implementation
- Peter Mell - portions of the DA routines;
- Richard Mills - the AIJPERM matrix format for the Cray X1 and universal F90 array interface;
- Victor Minden - the NVidia GPU interface;
- Todd Munson - the LUSOL (sparse solver in MINOS) interface and several Krylov methods;
- Adam Powell - the PETSc Debian package,
- Robert Scheichl - the MINRES implementation,
- Kerry Stevens - the pthread based Vec and Mat classes plus the various thread pools
- Karen Toonen - designed and implemented much of the PETSc web pages,
- Desire Nuentsa Wakam - the deflated GMRES implementation,
- Liyang Xu - the interface to PVODE (now Sundials/CVODE).

PETSc source code contains modfied routines from the following public domain software packages

- LINPACK - dense matrix factorization and solve; converted to C using `f2c` and then hand-optimized for small matrix sizes, for block matrix data structures;

- MINPACK - see page 127, sequential matrix coloring routines for finite difference Jacobian evaluations; converted to C using `f2c`;
- SPARSPAK - see page 81, matrix reordering routines, converted to C using `f2c`;
- libtfs - the efficient, parallel direct solver developed by Henry Tufo and Paul Fischer for the direct solution of a coarse grid problem (a linear system with very few degrees of freedom per processor).

PETSc interfaces to the following external software:

- ADIFOR - automatic differentiation for the computation of sparse Jacobians, http://www.mcs.anl.gov/adifor,
- BLAS and LAPACK - numerical linear algebra,
- Chaco - A graph partitioning package, http://www.cs.sandia.gov/CRF/chac.html
- ESSL - IBM's math library for fast sparse direct LU factorization,
- Elemental - Jack Poulson's parallel dense matrix solver package,
- hypre - the LLNL preconditioner library, http://www.llnl.gov/CASC/hypre
- LUSOL - sparse LU factorization code (part of MINOS) developed by Michael Saunders, Systems Optimization Laboratory, Stanford University, http://www.sbsi-sol-optimize.com/,
- Mathematica - see page **??**,
- MATLAB - see page 151,
- MUMPS - see page 94, MUltifrontal Massively Parallel sparse direct Solver developed by Patrick Amestoy, Iain Duff, Jacko Koster, and Jean-Yves L'Excellent, http://www.enseeiht.fr/lima/apo/MUMPS/credits.html,
- Metis/ParMeTiS - see page 70, parallel graph partitioner, http://www-users.cs.umn.edu/ karypis/metis/,
- Party - A graph partitioning package, http://www2.cs.uni-paderborn.de/cs/ag-monien/PERSONAL/ROBSY/party.html,
- PaStiX - Parallel LU and Cholesky solvers,
- PTScotch - A graph partitioning package, http://www.labri.fr/Perso/ pelegrin/scotch/
- SPAI - for parallel sparse approximate inverse preconditiong,
- SparseSuite - see page 94, developed by Timothy A. Davis, http://faculty.cse.tamu.edu/davis/suitesparse.html.
- Sundial/CVODE - see page 139, parallel ODE integrator, http://www.llnl.gov/CASC/sundials/,
- SuperLU and SuperLU_Dist - see page 94, the efficient sparse LU codes developed by Jim Demmel, Xiaoye S. Li, and John Gilbert, http://crd-legacy.lbl.gov/ xiaoye/SuperLU,
- Triangle and Tetgen - mesh generation packages,
- Trilinos/ML - Sandia's main multigrid preconditioning package, http://software.sandia.gov/trilinos/,

- Zoltan - graph partitioners from Sandia National Laboratory

These are all optional packages and do not need to be installed to use PETSc.

PETSc software is developed and maintained using

- Emacs editor
- Git revision control system
- Python

PETSc documentation has been generated using

- the text processing tools developed by Bill Gropp
- c2html
- pdflatex
- python

Contents

Part I

Introduction to PETSc

Chapter 1

Getting Started

The Portable, Extensible Toolkit for Scientific Computation (PETSc) has successfully demonstrated that the use of modern programming paradigms can ease the development of large-scale scientific application codes in Fortran, C, C++, and Python. Begun several years ago, the software has evolved into a powerful set of tools for the numerical solution of partial differential equations and related problems on high-performance computers.

PETSc consists of a variety of libraries (similar to classes in C++), which are discussed in detail in Parts II and III of the users manual. Each library manipulates a particular family of objects (for instance, vectors) and the operations one would like to perform on the objects. The objects and operations in PETSc are derived from our long experiences with scientific computation. Some of the PETSc modules deal with

- index sets (IS), including permutations, for indexing into vectors, renumbering, etc;
- vectors (Vec);
- matrices (Mat) (generally sparse);
- managing interactions between mesh data structures and vectors and matrices (DM);
- over fifteen Krylov subspace methods (KSP);
- dozens of preconditioners, including multigrid, block solvers, and sparse direct solvers (PC);
- nonlinear solvers (SNES); and
- timesteppers for solving time-dependent (nonlinear) PDEs, including support for differential algebraic equations, and the computation of adjoints (sensitivities/gradients of the solutions) (TS)

Each consists of an abstract interface (simply a set of calling sequences) and one or more implementations using particular data structures. Thus, PETSc provides clean and effective codes for the various phases of solving PDEs, with a uniform approach for each class of problems. This design enables easy comparison and use of different algorithms (for example, to experiment with different Krylov subspace methods, preconditioners, or truncated Newton methods). Hence, PETSc provides a rich environment for modeling scientific applications as well as for rapid algorithm design and prototyping.

The libraries enable easy customization and extension of both algorithms and implementations. This approach promotes code reuse and flexibility, and separates the issues of parallelism from the choice of algorithms. The PETSc infrastructure creates a foundation for building large-scale applications.

It is useful to consider the interrelationships among different pieces of PETSc. Figure 1 is a diagram of some of these pieces; Figure 2 presents several of the individual parts in more detail. These figures illustrate the library's hierarchical organization, which enables users to employ the level of abstraction that is most appropriate for a particular problem.

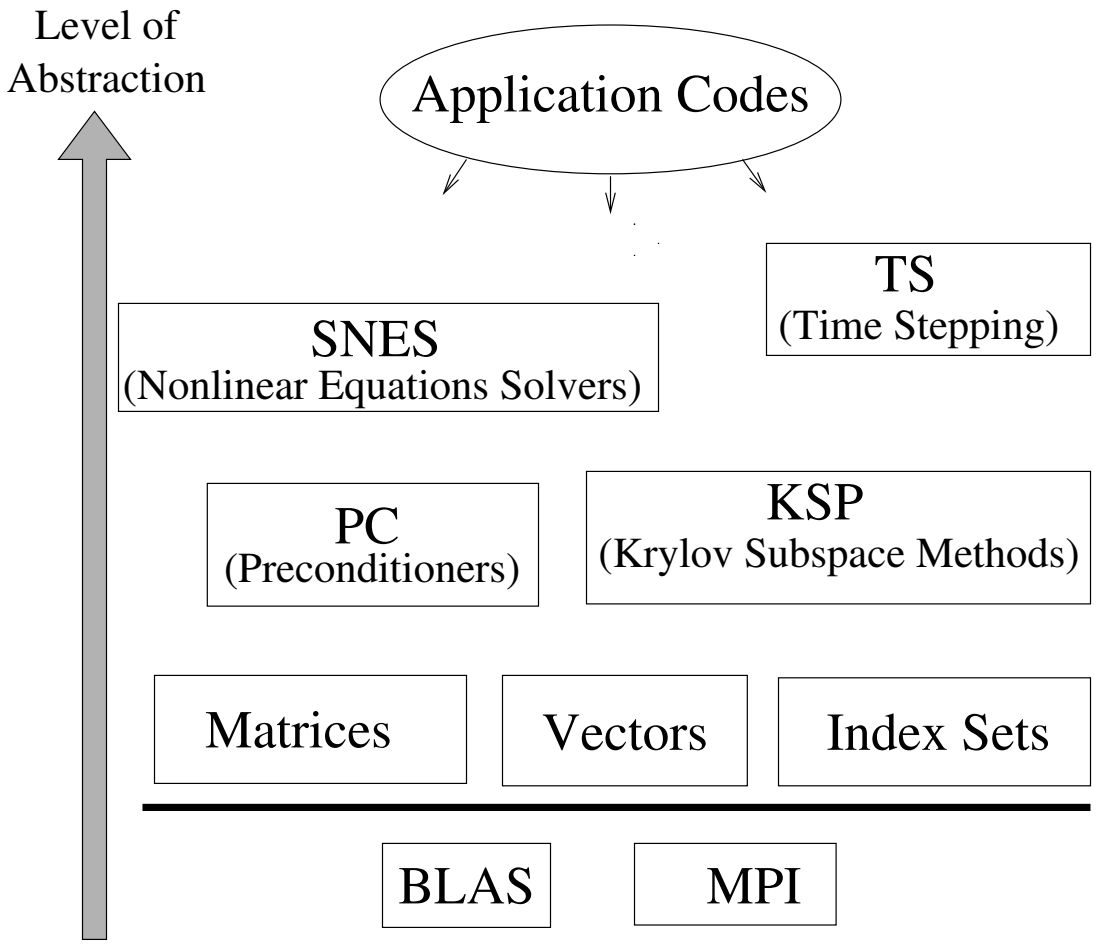

Figure 1: Organization of the PETSc Libraries

1.1 Suggested Reading

The manual is divided into three parts:

- Part I - Introduction to PETSc
- Part II - Programming with PETSc
- Part III - Additional Information

Part I describes the basic procedure for using the PETSc library and presents two simple examples of solving linear systems with PETSc. This section conveys the typical style used throughout the library and enables the application programmer to begin using the software immediately. Part I is also distributed separately for individuals interested in an overview of the PETSc software, excluding the details of library usage. Readers of this separate distribution of Part I should note that all references within the text to particular chapters and sections indicate locations in the complete users manual.

Part II explains in detail the use of the various PETSc libraries, such as vectors, matrices, index sets, linear and nonlinear solvers, and graphics. Part III describes a variety of useful information, including profiling, the options database, viewers, error handling, makefiles, and some details of PETSc design.

PETSc has evolved to become quite a comprehensive package, and therefore the *PETSc Users Manual* can be rather intimidating for new users. We recommend that one initially reads the entire document before

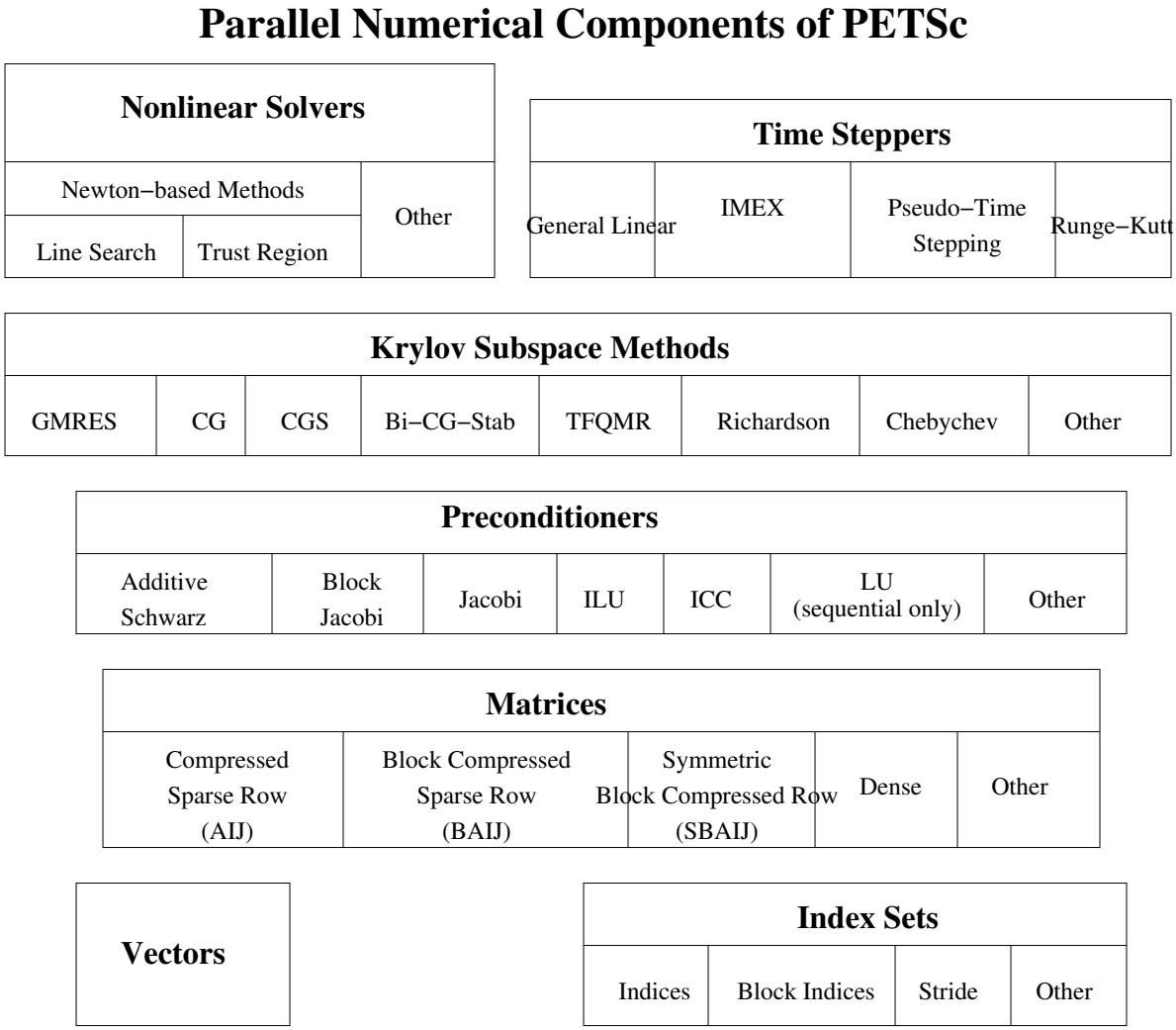

Figure 2: Numerical Libraries of PETSc

proceeding with serious use of PETSc, but bear in mind that PETSc can be used efficiently before one understands all of the material presented here. Furthermore, the definitive reference for any PETSc function is always the online manualpage.

Within the PETSc distribution, the directory `${PETSC_DIR}/docs` contains all documentation. Manual pages for all PETSc functions can be accessed at http://www.mcs.anl.gov/petsc/documentation. The manual pages provide hyperlinked indices (organized by both concepts and routine names) to the tutorial examples and enable easy movement among related topics.

Emacs and Vi/Vim users may find the `etags/ctags` option to be extremely useful for exploring the PETSc source code. Details of this feature are provided in Section 15.8.

The file `manual.pdf` contains the complete *PETSc Users Manual* in the portable document format (PDF), while `intro.pdf` includes only the introductory segment, Part I. The complete PETSc distribution, users manual, manual pages, and additional information are also available via the PETSc home page at http://www.mcs.anl.gov/petsc. The PETSc home page also contains details regarding installation, new features and changes in recent versions of PETSc, machines that we currently support, and a FAQ list for frequently asked questions.

Note to Fortran Programmers: In most of the manual, the examples and calling sequences are given for the C/C++ family of programming languages. We follow this convention because we recommend that PETSc applications be coded in C or C++. However, pure Fortran programmers can use most of the functionality of PETSc from Fortran, with only minor differences in the user interface. Chapter 12 provides a discussion of the differences between using PETSc from Fortran and C, as well as several complete Fortran examples. This chapter also introduces some routines that support direct use of Fortran90 pointers.
Note to Python Programmers: To program with PETSc in Python you need to install the PETSc4py package developed by Lisandro Dalcin. This can be done by configuring PETSc with the option `--download-petsc4py`. See the PETSc installation guide for more details http://www.mcs.anl.gov/petsc/documentation/installation.html

1.2 Running PETSc Programs

Before using PETSc, the user must first set the environmental variable `PETSC_DIR`, indicating the full path of the PETSc home directory. For example, under the UNIX bash shell a command of the form

export PETSC_DIR=$HOME/petsc

can be placed in the user's `.bashrc` or other startup file. In addition, the user may need to set the environmental variable `PETSC_ARCH` to specify a particular configuration of the PETSc libraries. Note that `PETSC_ARCH` is just a name selected by the installer to refer to the libraries compiled for a particular set of compiler options and machine type. Using different `PETSC_ARCH` allows one to switch between several different sets (say debug and optimized) of libraries easily. To determine if you need to set `PETSC_ARCH` look in the directory indicated by `PETSC_DIR`, if there are subdirectories beginning with `arch` then those subdirectories give the possible values for `PETSC_ARCH`.

All PETSc programs use the MPI (Message Passing Interface) standard for message-passing communication [22]. Thus, to execute PETSc programs, users must know the procedure for beginning MPI jobs on their selected computer system(s). For instance, when using the MPICH implementation of MPI [14] and many others, the following command initiates a program that uses eight processors:

mpiexec -n 8 ./petsc_program_name petsc_options

PETSc also comes with a script

$PETSC_DIR/bin/petscmpiexec -n 8 ./petsc_program_name petsc_options

that uses the information set in `${PETSC_DIR}/${PETSC_ARCH}/lib/petsc/conf/petscvariables` to automatically use the correct `mpiexec` for your configuration.

All PETSc-compliant programs support the use of the `-h` or `-help` option as well as the `-v` or `-version` option.

Certain options are supported by all PETSc programs. We list a few particularly useful ones below; a complete list can be obtained by running any PETSc program with the option `-help`.

- `-log_view` - summarize the program's performance
- `-fp_trap` - stop on floating-point exceptions; for example divide by zero
- `-malloc_dump` - enable memory tracing; dump list of unfreed memory at conclusion of the run
- `-malloc_debug` - enable memory tracing (by default this is activated for debugging versions)
- `-start_in_debugger [noxterm,gdb,dbx,xxgdb] [-display name]` - start all processes in debugger

- `-on_error_attach_debugger [noxterm,gdb,dbx,xxgdb] [-display name]` - start debugger only on encountering an error
- `-info` - print a great deal of information about what the programming is doing as it runs
- `-options_file filename` - read options from a file

See Section 15.4 for more information on debugging PETSc programs.

1.3 Writing PETSc Programs

Most PETSc programs begin with a call to

PetscInitialize(int *argc,char ***argv,char *file,char *help);

which initializes PETSc and MPI. The arguments `argc` and `argv` are the command line arguments delivered in all C and C++ programs. The argument `file` optionally indicates an alternative name for the PETSc options file, `.petscrc`, which resides by default in the user's home directory. Section 15.2 provides details regarding this file and the PETSc options database, which can be used for runtime customization. The final argument, `help`, is an optional character string that will be printed if the program is run with the `-help` option. In Fortran the initialization command has the form

call PetscInitialize(character(*) file,integer ierr)

PetscInitialize() automatically calls `MPI_Init()` if MPI has not been not previously initialized. In certain circumstances in which MPI needs to be initialized directly (or is initialized by some other library), the user can first call `MPI_Init()` (or have the other library do it), and then call PetscInitialize(). By default, PetscInitialize() sets the PETSc "world" communicator, given by PETSC_COMM_WORLD, to `MPI_COMM_WORLD`.

For those not familar with MPI, a *communicator* is a way of indicating a collection of processes that will be involved together in a calculation or communication. Communicators have the variable type `MPI_Comm`. In most cases users can employ the communicator PETSC_COMM_WORLD to indicate all processes in a given run and PETSC_COMM_SELF to indicate a single process.

MPI provides routines for generating new communicators consisting of subsets of processors, though most users rarely need to use these. The book *Using MPI*, by Lusk, Gropp, and Skjellum [15] provides an excellent introduction to the concepts in MPI, see also the MPI homepage http://www.mcs.anl.gov/mpi/. Note that PETSc users need not program much message passing directly with MPI, but they must be familar with the basic concepts of message passing and distributed memory computing.

All PETSc routines return an integer indicating whether an error has occurred during the call. The error code is set to be nonzero if an error has been detected; otherwise, it is zero. For the C/C++ interface, the error variable is the routine's return value, while for the Fortran version, each PETSc routine has as its final argument an integer error variable. Error tracebacks are discussed in the following section.

All PETSc programs should call PetscFinalize() as their final (or nearly final) statement, as given below in the C/C++ and Fortran formats, respectively:

PetscFinalize();
call PetscFinalize(ierr)

This routine handles options to be called at the conclusion of the program, and calls `MPI_Finalize()` if PetscInitialize() began MPI. If MPI was initiated externally from PETSc (by either the user or another software package), the user is responsible for calling `MPI_Finalize()`.

1.4 Simple PETSc Examples

To help the user start using PETSc immediately, we begin with a simple uniprocessor example in Figure 3 that solves the one-dimensional Laplacian problem with finite differences. This sequential code, which can be found in `${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex1.c`, illustrates the solution of a linear system with KSP, the interface to the preconditioners, Krylov subspace methods, and direct linear solvers of PETSc. Following the code we highlight a few of the most important parts of this example.

```
static char help[] = "Solves a tridiagonal linear system with KSP.\n\n";

/*T
   Concepts: KSP^solving a system of linear equations
   Processors: 1
T*/

/*
  Include "petscksp.h" so that we can use KSP solvers.  Note that this file
  automatically includes:
     petscsys.h       - base PETSc routines   petscvec.h - vectors
     petscmat.h - matrices
     petscis.h     - index sets            petscksp.h - Krylov subspace methods
     petscviewer.h - viewers               petscpc.h  - preconditioners

  Note:  The corresponding parallel example is ex23.c
*/
#include <petscksp.h>

#undef __FUNCT__
#define __FUNCT__ "main"
int main(int argc,char **args)
{
  Vec            x, b, u;      /* approx solution, RHS, exact solution */
  Mat            A;            /* linear system matrix */
  KSP            ksp;         /* linear solver context */
  PC             pc;           /* preconditioner context */
  PetscReal      norm;  /* norm of solution error */
  PetscErrorCode ierr;
  PetscInt       i,n = 10,col[3],its;
  PetscMPIInt    size;
  PetscScalar    neg_one      = -1.0,one = 1.0,value[3];
  PetscBool      nonzeroguess = PETSC_FALSE;

  PetscInitialize(&argc,&args,(char*)0,help);
  ierr = MPI_Comm_size(PETSC_COMM_WORLD,&size);CHKERRQ(ierr);
  if (size != 1) SETERRQ(PETSC_COMM_WORLD,1,"This is a uniprocessor example only!");
  ierr = PetscOptionsGetInt(NULL,NULL,"-n",&n,NULL);CHKERRQ(ierr);
  ierr = PetscOptionsGetBool(NULL,NULL,"-nonzero_guess",&nonzeroguess,NULL);CHKERRQ(ierr);


  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
         Compute the matrix and right-hand-side vector that define
         the linear system, Ax = b.
     - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

  /*
     Create vectors.  Note that we form 1 vector from scratch and
     then duplicate as needed.
  */
```

```
ierr = VecCreate(PETSC_COMM_WORLD,&x);CHKERRQ(ierr);
ierr = PetscObjectSetName((PetscObject) x, "Solution");CHKERRQ(ierr);
ierr = VecSetSizes(x,PETSC_DECIDE,n);CHKERRQ(ierr);
ierr = VecSetFromOptions(x);CHKERRQ(ierr);
ierr = VecDuplicate(x,&b);CHKERRQ(ierr);
ierr = VecDuplicate(x,&u);CHKERRQ(ierr);

/*
   Create matrix.  When using MatCreate(), the matrix format can
   be specified at runtime.

   Performance tuning note:  For problems of substantial size,
   preallocation of matrix memory is crucial for attaining good
   performance. See the matrix chapter of the users manual for details.
*/
ierr = MatCreate(PETSC_COMM_WORLD,&A);CHKERRQ(ierr);
ierr = MatSetSizes(A,PETSC_DECIDE,PETSC_DECIDE,n,n);CHKERRQ(ierr);
ierr = MatSetFromOptions(A);CHKERRQ(ierr);
ierr = MatSetUp(A);CHKERRQ(ierr);

/*
   Assemble matrix
*/
value[0] = -1.0; value[1] = 2.0; value[2] = -1.0;
for (i=1; i<n-1; i++) {
  col[0] = i-1; col[1] = i; col[2] = i+1;
  ierr   = MatSetValues(A,1,&i,3,col,value,INSERT_VALUES);CHKERRQ(ierr);
}
i    = n - 1; col[0] = n - 2; col[1] = n - 1;
ierr = MatSetValues(A,1,&i,2,col,value,INSERT_VALUES);CHKERRQ(ierr);
i    = 0; col[0] = 0; col[1] = 1; value[0] = 2.0; value[1] = -1.0;
ierr = MatSetValues(A,1,&i,2,col,value,INSERT_VALUES);CHKERRQ(ierr);
ierr = MatAssemblyBegin(A,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
ierr = MatAssemblyEnd(A,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);

/*
   Set exact solution; then compute right-hand-side vector.
*/
ierr = VecSet(u,one);CHKERRQ(ierr);
ierr = MatMult(A,u,b);CHKERRQ(ierr);

/* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
              Create the linear solver and set various options
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */
/*
   Create linear solver context
*/
ierr = KSPCreate(PETSC_COMM_WORLD,&ksp);CHKERRQ(ierr);

/*
   Set operators. Here the matrix that defines the linear system
   also serves as the preconditioning matrix.
*/
ierr = KSPSetOperators(ksp,A,A);CHKERRQ(ierr);

/*
   Set linear solver defaults for this problem (optional).
   - By extracting the KSP and PC contexts from the KSP context,
     we can then directly call any KSP and PC routines to set
     various options.
```

```
   - The following four statements are optional; all of these
     parameters could alternatively be specified at runtime via
     KSPSetFromOptions();
*/
ierr = KSPGetPC(ksp,&pc);CHKERRQ(ierr);
ierr = PCSetType(pc,PCJACOBI);CHKERRQ(ierr);
ierr = KSPSetTolerances(ksp,1.e-5,PETSC_DEFAULT,PETSC_DEFAULT,PETSC_DEFAULT);CHKERRQ(ierr);

/*
  Set runtime options, e.g.,
      -ksp_type <type> -pc_type <type> -ksp_monitor -ksp_rtol <rtol>
  These options will override those specified above as long as
  KSPSetFromOptions() is called _after_ any other customization
  routines.
*/
ierr = KSPSetFromOptions(ksp);CHKERRQ(ierr);

if (nonzeroguess) {
  PetscScalar p = .5;
  ierr = VecSet(x,p);CHKERRQ(ierr);
  ierr = KSPSetInitialGuessNonzero(ksp,PETSC_TRUE);CHKERRQ(ierr);
}

/* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
                   Solve the linear system
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */
/*
   Solve linear system
*/
ierr = KSPSolve(ksp,b,x);CHKERRQ(ierr);

/*
   View solver info; we could instead use the option -ksp_view to
   print this info to the screen at the conclusion of KSPSolve().
*/
ierr = KSPView(ksp,PETSC_VIEWER_STDOUT_WORLD);CHKERRQ(ierr);

/* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
                   Check solution and clean up
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */
/*
   Check the error
*/
ierr = VecAXPY(x,neg_one,u);CHKERRQ(ierr);
ierr = VecNorm(x,NORM_2,&norm);CHKERRQ(ierr);
ierr = KSPGetIterationNumber(ksp,&its);CHKERRQ(ierr);
ierr = PetscPrintf(PETSC_COMM_WORLD,"Norm of error %g, Iterations %D\n",(double)norm,its);CHKERRQ(i

/*
   Free work space.  All PETSc objects should be destroyed when they
   are no longer needed.
*/
ierr = VecDestroy(&x);CHKERRQ(ierr); ierr = VecDestroy(&u);CHKERRQ(ierr);
ierr = VecDestroy(&b);CHKERRQ(ierr); ierr = MatDestroy(&A);CHKERRQ(ierr);
ierr = KSPDestroy(&ksp);CHKERRQ(ierr);

/*
   Always call PetscFinalize() before exiting a program.  This routine
     - finalizes the PETSc libraries as well as MPI
     - provides summary and diagnostic information if certain runtime
```

```
        options are chosen (e.g., -log_summary).
  */
  ierr = PetscFinalize();
  return 0;
}
```

Figure 3: Example of Uniprocessor PETSc Code

Include Files

The C/C++ include files for PETSc should be used via statements such as

#include <petscksp.h>

where `petscksp.h` is the include file for the linear solver library. Each PETSc program must specify an include file that corresponds to the highest level PETSc objects needed within the program; all of the required lower level include files are automatically included within the higher level files. For example, `petscksp.h` includes `petscmat.h` (matrices), `petscvec.h` (vectors), and `petscsys.h` (base PETSc file). The PETSc include files are located in the directory `${PETSC_DIR}/include`. See Section 12.1.1 for a discussion of PETSc include files in Fortran programs.

The Options Database

As shown in Figure 3, the user can input control data at run time using the options database. In this example the command PetscOptionsGetInt(NULL,NULL,"-n",&n,&flg); checks whether the user has provided a command line option to set the value of `n`, the problem dimension. If so, the variable `n` is set accordingly; otherwise, `n` remains unchanged. A complete description of the options database may be found in Section 15.2.

Vectors

One creates a new parallel or sequential vector, `x`, of global dimension `M` with the commands

VecCreate(MPI_Comm comm,Vec *x);
VecSetSizes(Vec x, int m, int M);

where `comm` denotes the MPI communicator and `m` is the optional local size which may be PETSC_DECIDE. The type of storage for the vector may be set with either calls to VecSetType() or VecSetFromOptions(). Additional vectors of the same type can be formed with

VecDuplicate(Vec old,Vec *new);

The commands

VecSet(Vec x,PetscScalar value);
VecSetValues(Vec x,int n,int *indices,PetscScalar *values,INSERT_VALUES);

respectively set all the components of a vector to a particular scalar value and assign a different value to each component. More detailed information about PETSc vectors, including their basic operations, scattering/gathering, index sets, and distributed arrays, is discussed in Chapter 2.

Note the use of the PETSc variable type PetscScalar in this example. The PetscScalar is simply defined to be `double` in C/C++ (or correspondingly `double precision` in Fortran) for versions of PETSc that have *not* been compiled for use with complex numbers. The PetscScalar data type enables identical code to be used when the PETSc libraries have been compiled for use with complex numbers. Section 15.7 discusses the use of complex numbers in PETSc programs.

Matrices

Usage of PETSc matrices and vectors is similar. The user can create a new parallel or sequential matrix, `A`, which has `M` global rows and `N` global columns, with the routines

MatCreate(MPI_Comm comm,Mat *A);
MatSetSizes(Mat A,int m,int n,int M,int N);

where the matrix format can be specified at runtime. The user could alternatively specify each processes' number of local rows and columns using `m` and `n`. Generally one then sets the "type" of the matrix, with, for example,

MatSetType(Mat A,MATAIJ);

This causes the matrix to used the compressed sparse row storage format to store the matrix entries. See MatType for a list of all matrix types. Values can then be set with the command

MatSetValues(Mat A,int m,int *im,int n,int *in,PetscScalar *values,INSERT_VALUES);

After all elements have been inserted into the matrix, it must be processed with the pair of commands

MatAssemblyBegin(Mat A,MAT_FINAL_ASSEMBLY);
MatAssemblyEnd(Mat A,MAT_FINAL_ASSEMBLY);

Chapter 3 discusses various matrix formats as well as the details of some basic matrix manipulation routines.

Linear Solvers

After creating the matrix and vectors that define a linear system, `Ax = b`, the user can then use KSP to solve the system with the following sequence of commands:

KSPCreate(MPI_Comm comm,KSP *ksp);
KSPSetOperators(KSP ksp,Mat Amat,Mat Pmat);
KSPSetFromOptions(KSP ksp);
KSPSolve(KSP ksp,Vec b,Vec x);
KSPDestroy(KSP ksp);

The user first creates the KSP context and sets the operators associated with the system (matrix that defines the linear system, `Amat` and matrix from which the preconditioner is constructed, `Pmat`). The user then sets various options for customized solution, solves the linear system, and finally destroys the KSP context. We emphasize the command KSPSetFromOptions(), which enables the user to customize the linear solution method at runtime by using the options database, which is discussed in Section 15.2. Through this database, the user not only can select an iterative method and preconditioner, but also can prescribe the convergence tolerance, set various monitoring routines, etc. (see, e.g., Figure 7).

Chapter 4 describes in detail the KSP package, including the PC and KSP packages for preconditioners and Krylov subspace methods.

Nonlinear Solvers

Most PDE problems of interest are inherently nonlinear. PETSc provides an interface to tackle the nonlinear problems directly called SNES. Chapter 5 describes the nonlinear solvers in detail. We recommend most PETSc users work directly with SNES, rather than using PETSc for the linear problem within a nonlinear solver.

Error Checking

All PETSc routines return an integer indicating whether an error has occurred during the call. The PETSc macro `CHKERRQ(ierr)` checks the value of `ierr` and calls the PETSc error handler upon error detection. `CHKERRQ(ierr)` should be used in all subroutines to enable a complete error traceback. In Figure 4 we indicate a traceback generated by error detection within a sample PETSc program. The error occurred on line 1673 of the file `${PETSC_DIR}/src/mat/impls/aij/seq/aij.c` and was caused by trying to allocate too large an array in memory. The routine was called in the program `ex3.c` on line 71. See Section 12.1.2 for details regarding error checking when using the PETSc Fortran interface.

```
eagle:mpiexec -n 1 ./ex3 -m 10000
PETSC ERROR: MatCreateSeqAIJ() line 1673 in src/mat/impls/aij/seq/aij.c
PETSC ERROR: Out of memory. This could be due to allocating
PETSC ERROR: too large an object or bleeding by not properly
PETSC ERROR: destroying unneeded objects.
PETSC ERROR: Try running with -trdump for more information.
PETSC ERROR: MatSetType() line 99 in src/mat/utils/gcreate.c
PETSC ERROR: main() line 71 in src/ksp/ksp/examples/tutorials/ex3.c
MPI Abort by user Aborting program !
```

Figure 4: Example of Error Traceback

When running the debug version of the PETSc libraries, it does a great deal of checking for memory corruption (writing outside of array bounds etc). The macros `CHKMEMQ` can be called anywhere in the code to check the current status of the memory for corruption. By putting several (or many) of these macros into your code you can usually easily track down in what small segment of your code the corruption has occured.

Parallel Programming

Since PETSc uses the message-passing model for parallel programming and employs MPI for all interprocessor communication, the user is free to employ MPI routines as needed throughout an application code. However, by default the user is shielded from many of the details of message passing within PETSc, since these are hidden within parallel objects, such as vectors, matrices, and solvers. In addition, PETSc provides tools such as generalized vector scatters/gathers and distributed arrays to assist in the management of parallel data.

Recall that the user must specify a communicator upon creation of any PETSc object (such as a vector, matrix, or solver) to indicate the processors over which the object is to be distributed. For example, as mentioned above, some commands for matrix, vector, and linear solver creation are:

```
MatCreate(MPI_Comm comm,Mat *A);
VecCreate(MPI_Comm comm,Vec *x);
KSPCreate(MPI_Comm comm,KSP *ksp);
```

The creation routines are collective over all processors in the communicator; thus, all processors in the communicator *must* call the creation routine. In addition, if a sequence of collective routines is being used, they *must* be called in the same order on each processor.

The next example, given in Figure 5, illustrates the solution of a linear system in parallel. This code, corresponding to `${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex2.c`, handles the two-dimensional Laplacian discretized with finite differences, where the linear system is again solved with KSP.

The code performs the same tasks as the sequential version within Figure 3. Note that the user interface for initiating the program, creating vectors and matrices, and solving the linear system is *exactly* the same for the uniprocessor and multiprocessor examples. The primary difference between the examples in Figures 3 and 5 is that each processor forms only its local part of the matrix and vectors in the parallel case.

```
static char help[] = "Solves a linear system in parallel with KSP.\n\
Input parameters include:\n\
  -random_exact_sol : use a random exact solution vector\n\
  -view_exact_sol   : write exact solution vector to stdout\n\
  -m <mesh_x>       : number of mesh points in x-direction\n\
  -n <mesh_n>       : number of mesh points in y-direction\n\n";

/*T
   Concepts: KSP^basic parallel example;
   Concepts: KSP^Laplacian, 2d
   Concepts: Laplacian, 2d
   Processors: n
T*/

/*
  Include "petscksp.h" so that we can use KSP solvers.  Note that this file
  automatically includes:
     petscsys.h       - base PETSc routines   petscvec.h - vectors
     petscmat.h - matrices
     petscis.h     - index sets            petscksp.h - Krylov subspace methods
     petscviewer.h - viewers               petscpc.h  - preconditioners
*/
#include <petscksp.h>

#undef __FUNCT__
#define __FUNCT__ "main"
int main(int argc,char **args)
{
  Vec            x,b,u;  /* approx solution, RHS, exact solution */
  Mat            A;        /* linear system matrix */
  KSP            ksp;     /* linear solver context */
  PetscRandom    rctx;     /* random number generator context */
  PetscReal      norm;     /* norm of solution error */
  PetscInt       i,j,Ii,J,Istart,Iend,m = 8,n = 7,its;
  PetscErrorCode ierr;
  PetscBool      flg = PETSC_FALSE;
  PetscScalar    v;
#if defined(PETSC_USE_LOG)
  PetscLogStage stage;
#endif

  PetscInitialize(&argc,&args,(char*)0,help);
  ierr = PetscOptionsGetInt(NULL,NULL,"-m",&m,NULL);CHKERRQ(ierr);
  ierr = PetscOptionsGetInt(NULL,NULL,"-n",&n,NULL);CHKERRQ(ierr);
  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
         Compute the matrix and right-hand-side vector that define
         the linear system, Ax = b.
     - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */
  /*
     Create parallel matrix, specifying only its global dimensions.
     When using MatCreate(), the matrix format can be specified at
     runtime. Also, the parallel partitioning of the matrix is
     determined by PETSc at runtime.
```

```
   Performance tuning note:  For problems of substantial size,
   preallocation of matrix memory is crucial for attaining good
   performance. See the matrix chapter of the users manual for details.
*/
ierr = MatCreate(PETSC_COMM_WORLD,&A);CHKERRQ(ierr);
ierr = MatSetSizes(A,PETSC_DECIDE,PETSC_DECIDE,m*n,m*n);CHKERRQ(ierr);
ierr = MatSetFromOptions(A);CHKERRQ(ierr);
ierr = MatMPIAIJSetPreallocation(A,5,NULL,5,NULL);CHKERRQ(ierr);
ierr = MatSeqAIJSetPreallocation(A,5,NULL);CHKERRQ(ierr);
ierr = MatSeqSBAIJSetPreallocation(A,1,5,NULL);CHKERRQ(ierr);

/*
   Currently, all PETSc parallel matrix formats are partitioned by
   contiguous chunks of rows across the processors.  Determine which
   rows of the matrix are locally owned.
*/
ierr = MatGetOwnershipRange(A,&Istart,&Iend);CHKERRQ(ierr);

/*
   Set matrix elements for the 2-D, five-point stencil in parallel.
    - Each processor needs to insert only elements that it owns
      locally (but any non-local elements will be sent to the
      appropriate processor during matrix assembly).
    - Always specify global rows and columns of matrix entries.

   Note: this uses the less common natural ordering that orders first
   all the unknowns for x = h then for x = 2h etc; Hence you see J = Ii +- n
   instead of J = I +- m as you might expect. The more standard ordering
   would first do all variables for y = h, then y = 2h etc.

 */
ierr = PetscLogStageRegister("Assembly", &stage);CHKERRQ(ierr);
ierr = PetscLogStagePush(stage);CHKERRQ(ierr);
for (Ii=Istart; Ii<Iend; Ii++) {
  v = -1.0; i = Ii/n; j = Ii - i*n;
  if (i>0)   {J = Ii - n; ierr = MatSetValues(A,1,&Ii,1,&J,&v,ADD_VALUES);CHKERRQ(ierr);}
  if (i<m-1) {J = Ii + n; ierr = MatSetValues(A,1,&Ii,1,&J,&v,ADD_VALUES);CHKERRQ(ierr);}
  if (j>0)   {J = Ii - 1; ierr = MatSetValues(A,1,&Ii,1,&J,&v,ADD_VALUES);CHKERRQ(ierr);}
  if (j<n-1) {J = Ii + 1; ierr = MatSetValues(A,1,&Ii,1,&J,&v,ADD_VALUES);CHKERRQ(ierr);}
  v = 4.0; ierr = MatSetValues(A,1,&Ii,1,&Ii,&v,ADD_VALUES);CHKERRQ(ierr);
}

/*
   Assemble matrix, using the 2-step process:
     MatAssemblyBegin(), MatAssemblyEnd()
   Computations can be done while messages are in transition
   by placing code between these two statements.
*/
ierr = MatAssemblyBegin(A,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
ierr = MatAssemblyEnd(A,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
ierr = PetscLogStagePop();CHKERRQ(ierr);

/* A is symmetric. Set symmetric flag to enable ICC/Cholesky preconditioner */
ierr = MatSetOption(A,MAT_SYMMETRIC,PETSC_TRUE);CHKERRQ(ierr);

/*
   Create parallel vectors.
    - We form 1 vector from scratch and then duplicate as needed.
    - When using VecCreate(), VecSetSizes and VecSetFromOptions()
```

```
       in this example, we specify only the
       vector's global dimension; the parallel partitioning is determined
       at runtime.
     - When solving a linear system, the vectors and matrices MUST
       be partitioned accordingly.  PETSc automatically generates
       appropriately partitioned matrices and vectors when MatCreate()
       and VecCreate() are used with the same communicator.
     - The user can alternatively specify the local vector and matrix
       dimensions when more sophisticated partitioning is needed
       (replacing the PETSC_DECIDE argument in the VecSetSizes() statement
       below).
*/
ierr = VecCreate(PETSC_COMM_WORLD,&u);CHKERRQ(ierr);
ierr = VecSetSizes(u,PETSC_DECIDE,m*n);CHKERRQ(ierr);
ierr = VecSetFromOptions(u);CHKERRQ(ierr);
ierr = VecDuplicate(u,&b);CHKERRQ(ierr);
ierr = VecDuplicate(b,&x);CHKERRQ(ierr);

/*
   Set exact solution; then compute right-hand-side vector.
   By default we use an exact solution of a vector with all
   elements of 1.0;  Alternatively, using the runtime option
   -random_sol forms a solution vector with random components.
*/
ierr = PetscOptionsGetBool(NULL,NULL,"-random_exact_sol",&flg,NULL);CHKERRQ(ierr);
if (flg) {
  ierr = PetscRandomCreate(PETSC_COMM_WORLD,&rctx);CHKERRQ(ierr);
  ierr = PetscRandomSetFromOptions(rctx);CHKERRQ(ierr);
  ierr = VecSetRandom(u,rctx);CHKERRQ(ierr);
  ierr = PetscRandomDestroy(&rctx);CHKERRQ(ierr);
} else {
  ierr = VecSet(u,1.0);CHKERRQ(ierr);
}
ierr = MatMult(A,u,b);CHKERRQ(ierr);

/*
   View the exact solution vector if desired
*/
flg  = PETSC_FALSE;
ierr = PetscOptionsGetBool(NULL,NULL,"-view_exact_sol",&flg,NULL);CHKERRQ(ierr);
if (flg) {ierr = VecView(u,PETSC_VIEWER_STDOUT_WORLD);CHKERRQ(ierr);}

/* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
              Create the linear solver and set various options
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

/*
   Create linear solver context
*/
ierr = KSPCreate(PETSC_COMM_WORLD,&ksp);CHKERRQ(ierr);

/*
   Set operators. Here the matrix that defines the linear system
   also serves as the preconditioning matrix.
*/
ierr = KSPSetOperators(ksp,A,A);CHKERRQ(ierr);

/*
   Set linear solver defaults for this problem (optional).
   - By extracting the KSP and PC contexts from the KSP context,
```

```
      we can then directly call any KSP and PC routines to set
      various options.
    - The following two statements are optional; all of these
      parameters could alternatively be specified at runtime via
      KSPSetFromOptions().  All of these defaults can be
      overridden at runtime, as indicated below.
  */
  ierr = KSPSetTolerances(ksp,1.e-2/((m+1)*(n+1)),1.e-50,PETSC_DEFAULT,
                          PETSC_DEFAULT);CHKERRQ(ierr);

  /*
    Set runtime options, e.g.,
        -ksp_type <type> -pc_type <type> -ksp_monitor -ksp_rtol <rtol>
    These options will override those specified above as long as
    KSPSetFromOptions() is called _after_ any other customization
    routines.
  */
  ierr = KSPSetFromOptions(ksp);CHKERRQ(ierr);

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
                      Solve the linear system
     - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

  ierr = KSPSolve(ksp,b,x);CHKERRQ(ierr);

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
                      Check solution and clean up
     - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

  /*
     Check the error
  */
  ierr = VecAXPY(x,-1.0,u);CHKERRQ(ierr);
  ierr = VecNorm(x,NORM_2,&norm);CHKERRQ(ierr);
  ierr = KSPGetIterationNumber(ksp,&its);CHKERRQ(ierr);

  /*
     Print convergence information.  PetscPrintf() produces a single
     print statement from all processes that share a communicator.
     An alternative is PetscFPrintf(), which prints to a file.
  */
  ierr = PetscPrintf(PETSC_COMM_WORLD,"Norm of error %g iterations %D\n",(double)norm,its);CHKERRQ(ie

  /*
     Free work space.  All PETSc objects should be destroyed when they
     are no longer needed.
  */
  ierr = KSPDestroy(&ksp);CHKERRQ(ierr);
  ierr = VecDestroy(&u);CHKERRQ(ierr);  ierr = VecDestroy(&x);CHKERRQ(ierr);
  ierr = VecDestroy(&b);CHKERRQ(ierr);  ierr = MatDestroy(&A);CHKERRQ(ierr);

  /*
     Always call PetscFinalize() before exiting a program.  This routine
       - finalizes the PETSc libraries as well as MPI
       - provides summary and diagnostic information if certain runtime
         options are chosen (e.g., -log_summary).
  */
  ierr = PetscFinalize();
  return 0;
}
```

Figure 5: Example of Multiprocessor PETSc Code

Compiling and Running Programs

Figure 6 illustrates compiling and running a PETSc program using MPICH. Note that different sites may have slightly different library and compiler names. See Chapter 16 for a discussion about compiling PETSc programs. Users who are experiencing difficulties linking PETSc programs should refer to the FAQ via the PETSc WWW home page http://www.mcs.anl.gov/petsc or given in the file $PETSC_DIR/docs/faq.html.

```
eagle: make ex2
gcc -pipe -c -I../../../ -I../../..//include
-I/usr/local/mpi/include -I../../..//src -g
-DPETSC_USE_DEBUG -DPETSC_MALLOC -DPETSC_USE_LOG ex1.c
gcc -g -DPETSC_USE_DEBUG -DPETSC_MALLOC -DPETSC_USE_LOG -o ex1 ex1.o
/home/bsmith/petsc/lib/libg/sun4/libpetscksp.a
-L/home/bsmith/petsc/lib/libg/sun4 -lpetscstencil -lpetscgrid -lpetscksp
-lpetscmat -lpetscvec -lpetscsys -lpetscdraw
/usr/local/lapack/lib/lapack.a /usr/local/lapack/lib/blas.a
/usr/lang/SC1.0.1/libF77.a -lm /usr/lang/SC1.0.1/libm.a -lX11
/usr/local/mpi/lib/sun4/ch_p4/libmpi.a
/usr/lib/debug/malloc.o /usr/lib/debug/mallocmap.o
/usr/lang/SC1.0.1/libF77.a -lm /usr/lang/SC1.0.1/libm.a -lm
rm -f ex1.o
eagle: mpiexec -n 1 ./ex2
Norm of error 3.6618e-05 iterations 7
eagle:
eagle: mpiexec -n 2 ./ex2
Norm of error 5.34462e-05 iterations 9
```

Figure 6: Running a PETSc Program

As shown in Figure 7, the option `-log_view` activates printing of a performance summary, including times, floating point operation (flop) rates, and message-passing activity. Chapter 13 provides details about profiling, including interpretation of the output data within Figure 7. This particular example involves the solution of a linear system on one processor using GMRES and ILU. The low floating point operation (flop) rates in this example are due to the fact that the code solved a tiny system. We include this example merely to demonstrate the ease of extracting performance information.

```
eagle> mpiexec -n 1 ./ex1 -n 1000 -pc_type ilu -ksp_type gmres -ksp_rtol 1.e-7 -log_view
------------------------------ PETSc Performance Summary: ------------------------------

ex1 on a sun4 named merlin.mcs.anl.gov with 1 processor, by curfman Wed Aug 7 17:24 1996

                         Max         Min         Avg        Total
Time (sec):           1.150e-01      1.0   1.150e-01
Objects:              1.900e+01      1.0   1.900e+01
Flops:                3.998e+04      1.0   3.998e+04  3.998e+04
Flops/sec:            3.475e+05      1.0              3.475e+05
MPI Messages:         0.000e+00      0.0   0.000e+00  0.000e+00
MPI Messages:         0.000e+00      0.0   0.000e+00  0.000e+00 (lengths)
MPI Reductions:       0.000e+00      0.0

------------------------------------------------------------------------------------
```

```
Phase               Count      Time (sec)       Flops/sec     Messages    -- Global --
                             Max     Ratio    Max    Ratio Avg len  Redc %T %F %M %L %R
------------------------------------------------------------------------------------------
Mat Mult                2  2.553e-03    1.0  3.9e+06    1.0  0.0 0.0 0.0  2 25  0  0  0
Mat AssemblyBegin       1  2.193e-05    1.0  0.0e+00    0.0  0.0 0.0 0.0  0  0  0  0  0
Mat AssemblyEnd         1  5.004e-03    1.0  0.0e+00    0.0  0.0 0.0 0.0  4  0  0  0  0
Mat GetOrdering         1  3.004e-03    1.0  0.0e+00    0.0  0.0 0.0 0.0  3  0  0  0  0
Mat ILUFctrSymbol       1  5.719e-03    1.0  0.0e+00    0.0  0.0 0.0 0.0  5  0  0  0  0
Mat LUFactorNumer       1  1.092e-02    1.0  2.7e+05    1.0  0.0 0.0 0.0  9  7  0  0  0
Mat Solve               2  4.193e-03    1.0  2.4e+06    1.0  0.0 0.0 0.0  4 25  0  0  0
Mat SetValues        1000  2.461e-02    1.0  0.0e+00    0.0  0.0 0.0 0.0 21  0  0  0  0
Vec Dot                 1     60e-04    1.0  9.7e+06    1.0  0.0 0.0 0.0  0  5  0  0  0
Vec Norm                3  5.870e-04    1.0  1.0e+07    1.0  0.0 0.0 0.0  1 15  0  0  0
Vec Scale               1  1.640e-04    1.0  6.1e+06    1.0  0.0 0.0 0.0  0  3  0  0  0
Vec Copy                1  3.101e-04    1.0  0.0e+00    0.0  0.0 0.0 0.0  0  0  0  0  0
Vec Set                 3  5.029e-04    1.0  0.0e+00    0.0  0.0 0.0 0.0  0  0  0  0  0
Vec AXPY                3  8.690e-04    1.0  6.9e+06    1.0  0.0 0.0 0.0  1 15  0  0  0
Vec MAXPY               1  2.550e-04    1.0  7.8e+06    1.0  0.0 0.0 0.0  0  5  0  0  0
KSP Solve               1  1.288e-02    1.0  2.2e+06    1.0  0.0 0.0 0.0 11 70  0  0  0
KSP SetUp               1  2.669e-02    1.0  1.1e+05    1.0  0.0 0.0 0.0 23  7  0  0  0
KSP GMRESOrthog         1  1.151e-03    1.0  3.5e+06    1.0  0.0 0.0 0.0  1 10  0  0  0
PC SetUp                1  2.4e-02      1.0  1.5e+05    1.0  0.0 0.0 0.0 18  7  0  0  0
PC Apply                2  4.474e-03    1.0  2.2e+06    1.0  0.0 0.0 0.0  4 25  0  0  0
------------------------------------------------------------------------------------------

Memory usage is given in bytes:

Object Type      Creations   Destructions   Memory  Descendants' Mem.
Index set              3              3      12420     0
Vector                 8              8      65728     0
Matrix                 2              2     184924     4140
Krylov Solver          1              1      16892     41080
Preconditioner         1              1          0     64872
```

Figure 7: Running a PETSc Program with Profiling

Writing Application Codes with PETSc

The examples throughout the library demonstrate the software usage and can serve as templates for developing custom applications. We suggest that new PETSc users examine programs in the directories

`${PETSC_DIR}/src/<library>/examples/tutorials`,

where `<library>` denotes any of the PETSc libraries (listed in the following section), such as `SNES` or `KSP`. The manual pages located at

$PETSC_DIR/docs/index.html or
http://www.mcs.anl.gov/petsc/documentation

provide indices (organized by both routine names and concepts) to the tutorial examples.

To write a new application program using PETSc, we suggest the following procedure:

1. Install and test PETSc according to the instructions at the PETSc web site.
2. Copy one of the many PETSc examples in the directory that corresponds to the class of problem of interest (e.g., for linear solvers, see `${PETSC_DIR}/src/ksp/ksp/examples/tutorials`).

3. Copy the corresponding makefile within the example directory; compile and run the example program.

4. Use the example program as a starting point for developing a custom code.

1.5 Citing PETSc

When citing PETSc in a publication please cite the following:

@Misc{petsc-web-page,
Author = "Satish Balay and Shrirang Abhyankar and Mark F. Adams and Jed Brown
and Peter Brune and Kris Buschelman and Lisandro Dalcin and Victor Eijkhout
and William D. Gropp and Dinesh Kaushik and Matthew G. Knepley and Lois Curfman McInnes
and Karl Rupp and Barry F. Smith and Stefano Zampini and Hong Zhang and Hong Zhang",
Title = "PETSc Web page",
Note = "http://www.mcs.anl.gov/petsc",
Year = "2016"}
@TechReport{petsc-user-ref,
Author = "Satish Balay and Shrirang Abhyankar and Mark F. Adams and Jed Brown
and Peter Brune and Kris Buschelman and Lisandro Dalcin and Victor Eijkhout
and Dinesh Kaushik and Matthew G. Knepley and Lois Curfman McInnes
and William D. Gropp and Karl Rupp and Barry F. Smith and Stefano Zampini and Hong Zhang and Hong Z
Title = "PETSc Users Manual",
Number = "ANL-95/11 - Revision 3.7",
Institution = "Argonne National Laboratory",
Year = "2016"}
@InProceedings{petsc-efficient,
Author = "Satish Balay and William D. Gropp and Lois C. McInnes and Barry F. Smith",
Title = "Efficienct Management of Parallelism in Object Oriented Numerical Software Libraries",
Booktitle = "Modern Software Tools in Scientific Computing",
Editor = "E. Arge and A. M. Bruaset and H. P. Langtangen",
Pages = "163–202",
Publisher = "Birkhauser Press",
Year = "1997"}

1.6 Directory Structure

We conclude this introduction with an overview of the organization of the PETSc software. The root directory of PETSc contains the following directories:

- `docs` - All documentation for PETSc. The files `manual.pdf` contains the hyperlinked users manual, suitable for printing or on-screen viewering. Includes the subdirectory

  - `manualpages` (on-line manual pages).

- `bin` - Utilities and short scripts for use with PETSc, including

  - `petscmpiexec` (utility for setting running MPI jobs),

- `conf` - Base PETSc configuration files that define the standard make variables and rules used by PETSc

- `include` - All include files for PETSc that are visible to the user.
- `include/petsc/finclude` - PETSc include files for Fortran programmers using the .F suffix (recommended).
- `include/petsc/private` - Private PETSc include files that should *not* need to be used by application programmers.
- `share` - Some small test matrices in data files
- `src` - The source code for all PETSc libraries, which currently includes
  - `vec` - vectors,
    - `is` - index sets,
  - `mat` - matrices,
  - `dm` - data management between meshes and vectors and matrices,
  - `ksp` - complete linear equations solvers,
    - `ksp` - Krylov subspace accelerators,
    - `pc` - preconditioners,
  - `snes` - nonlinear solvers
  - `ts` - ODE solvers and timestepping,
  - `sys` - general system-related routines,
    - `logging` - PETSc logging and profiling routines,
    - `classes` - low-level classes
      - `draw` - simple graphics,
      - `viewer`
      - `bag`
      - `random` - random number generators.
  - `contrib` - contributed modules that use PETSc but are not part of the official PETSc package. We encourage users who have developed such code that they wish to share with others to let us know by writing to petsc-maint@mcs.anl.gov.

Each PETSc source code library directory has the following subdirectories:

- `examples` - Example programs for the component, including
  - `tutorials` - Programs designed to teach users about PETSc. These codes can serve as templates for the design of custom applications.
  - `tests` - Programs designed for thorough testing of PETSc. As such, these codes are not intended for examination by users.
- `interface` - The calling sequences for the abstract interface to the component. Code here does not know about particular implementations.
- `impls` - Source code for one or more implementations.
- `utils` - Utility routines. Source here may know about the implementations, but ideally will not know about implementations for other components.

Part II

Programming with PETSc

Chapter 2

Vectors and Distributing Parallel Data

The vector (denoted by Vec) is one of the simplest PETSc objects. Vectors are used to store discrete PDE solutions, right-hand sides for linear systems, etc. This chapter is organized as follows:

- (Vec) Sections 2.1 and 2.2 - basic usage of vectors
- Section 2.3 - management of the various numberings of degrees of freedom, vertices, cells, etc.
  - (AO) Mapping between different global numberings
  - (ISLocalToGlobalMapping) Mapping between local and global numberings
- (DM) Section 2.4 - management of grids
- (IS, VecScatter) Section 2.5 - management of vectors related to unstructured grids

2.1 Creating and Assembling Vectors

PETSc currently provides two basic vector types: sequential and parallel (MPI based). To create a sequential vector with `m` components, one can use the command

VecCreateSeq(PETSC_COMM_SELF,int m,Vec *x);

To create a parallel vector one can either specify the number of components that will be stored on each process or let PETSc decide. The command

VecCreateMPI(MPI_Comm comm,int m,int M,Vec *x);

creates a vector that is distributed over all processes in the communicator, comm, where `m` indicates the number of components to store on the local process, and `M` is the total number of vector components. Either the local or global dimension, but not both, can be set to PETSC_DECIDE to indicate that PETSc should determine it. More generally, one can use the routines

VecCreate(MPI_Comm comm,Vec *v);
VecSetSizes(Vec v, int m, int M);
VecSetFromOptions(Vec v);

which automatically generates the appropriate vector type (sequential or parallel) over all processes in `comm`. The option `-vec_type mpi` can be used in conjunction with VecCreate() and VecSetFromOptions() to specify the use of MPI vectors even for the uniprocess case.

We emphasize that all processes in `comm` *must* call the vector creation routines, since these routines are collective over all processes in the communicator. If you are not familiar with MPI communicators, see the discussion in Section 1.3 on page 25. In addition, if a sequence of `VecCreateXXX()` routines is used, they must be called in the same order on each process in the communicator.

One can assign a single value to all components of a vector with the command

VecSet(Vec x,PetscScalar value);

Assigning values to individual components of the vector is more complicated, in order to make it possible to write efficient parallel code. Assigning a set of components is a two-step process: one first calls

VecSetValues(Vec x,int n,int *indices,PetscScalar *values,INSERT_VALUES);

any number of times on any or all of the processes. The argument `n` gives the number of components being set in this insertion. The integer array `indices` contains the *global component indices*, and `values` is the array of values to be inserted. Any process can set any components of the vector; PETSc insures that they are automatically stored in the correct location. Once all of the values have been inserted with VecSetValues(), one must call

VecAssemblyBegin(Vec x);

followed by

VecAssemblyEnd(Vec x);

to perform any needed message passing of nonlocal components. In order to allow the overlap of communication and calculation, the user's code can perform any series of other actions between these two calls while the messages are in transition.

Example usage of VecSetValues() may be found in `${PETSC_DIR}/src/vec/vec/examples/tutorials/ex2.c` or `ex2f.F`.

Often, rather than inserting elements in a vector, one may wish to add values. This process is also done with the command

VecSetValues(Vec x,int n,int *indices, PetscScalar *values,ADD_VALUES);

Again one must call the assembly routines VecAssemblyBegin() and VecAssemblyEnd() after all of the values have been added. Note that addition and insertion calls to VecSetValues() *cannot* be mixed. Instead, one must add and insert vector elements in phases, with intervening calls to the assembly routines. This phased assembly procedure overcomes the nondeterministic behavior that would occur if two different processes generated values for the same location, with one process adding while the other is inserting its value. (In this case the addition and insertion actions could be performed in either order, thus resulting in different values at the particular location. Since PETSc does not allow the simultaneous use of INSERT_VALUES and ADD_VALUES this nondeterministic behavior will not occur in PETSc.)

You can called VecGetValues() to pull local values from a vector (but not off-process values), an alternative method for extracting some components of a vector are the vector scatter routines. See Section 2.5.2 for details; see also below for VecGetArray().

One can examine a vector with the command

VecView(Vec x,PetscViewer v);

To print the vector to the screen, one can use the viewer PETSC_VIEWER_STDOUT_WORLD, which ensures that parallel vectors are printed correctly to `stdout`. To display the vector in an X-window, one can use the default X-windows viewer PETSC_VIEWER_DRAW_WORLD, or one can create a viewer with the routine PetscViewerDrawOpenX(). A variety of viewers are discussed further in Section 15.3.

To create a new vector of the same format as an existing vector, one uses the command

VecDuplicate(Vec old,Vec *new);

To create several new vectors of the same format as an existing vector, one uses the command

VecDuplicateVecs(Vec old,int n,Vec **new);

This routine creates an array of pointers to vectors. The two routines are very useful because they allow one to write library code that does not depend on the particular format of the vectors being used. Instead, the subroutines can automatically correctly create work vectors based on the specified existing vector. As discussed in Section 12.1.6, the Fortran interface for VecDuplicateVecs() differs slightly.

When a vector is no longer needed, it should be destroyed with the command

VecDestroy(Vec *x);

To destroy an array of vectors, use the command

VecDestroyVecs(PetscInt n,Vec **vecs);

Note that the Fortran interface for VecDestroyVecs() differs slightly, as described in Section 12.1.6.

It is also possible to create vectors that use an array provided by the user, rather than having PETSc internally allocate the array space. Such vectors can be created with the routines

VecCreateSeqWithArray(PETSC_COMM_SELF,int bs,int n,PetscScalar *array,Vec *V);

and

VecCreateMPIWithArray(MPI_Comm comm,int bs,int n,int N,PetscScalar *array,Vec *vv);

Note that here one must provide the value `n`, it cannot be PETSC_DECIDE and the user is responsible for providing enough space in the array; *n*sizeof(PetscScalar)*.

2.2 Basic Vector Operations

As listed in Table 1, we have chosen certain basic vector operations to support within the PETSc vector library. These operations were selected because they often arise in application codes. The NormType argument to VecNorm() is one of NORM_1, NORM_2, or NORM_INFINITY. The 1-norm is $\sum_i |x_i|$, the 2-norm is $(\sum_i x_i^2)^{1/2}$ and the infinity norm is $\max_i |x_i|$.

For parallel vectors that are distributed across the processes by ranges, it is possible to determine a process's local range with the routine

VecGetOwnershipRange(Vec vec,int *low,int *high);

The argument `low` indicates the first component owned by the local process, while `high` specifies *one more than* the last owned by the local process. This command is useful, for instance, in assembling parallel vectors.

On occasion, the user needs to access the actual elements of the vector. The routine VecGetArray() returns a pointer to the elements local to the process:

VecGetArray(Vec v,PetscScalar **array);

When access to the array is no longer needed, the user should call

VecRestoreArray(Vec v, PetscScalar **array);

Function Name	Operation
VecAXPY(Vec y,PetscScalar a,Vec x);	$y = y + a * x$
VecAYPX(Vec y,PetscScalar a,Vec x);	$y = x + a * y$
VecWAXPY(Vec w,PetscScalar a,Vec x,Vec y);	$w = a * x + y$
VecAXPBY(Vec y,PetscScalar a,PetscScalar b,Vec x);	$y = a * x + b * y$
VecScale(Vec x, PetscScalar a);	$x = a * x$
VecDot(Vec x, Vec y, PetscScalar *r);	$r = \bar{x}' * y$
VecTDot(Vec x, Vec y, PetscScalar *r);	$r = x' * y$
VecNorm(Vec x,NormType type, PetscReal *r);	$r = \|\|x\|\|_{type}$
VecSum(Vec x, PetscScalar *r);	$r = \sum x_i$
VecCopy(Vec x, Vec y);	$y = x$
VecSwap(Vec x, Vec y);	$y = x$ while $x = y$
VecPointwiseMult(Vec w,Vec x,Vec y);	$w_i = x_i * y_i$
VecPointwiseDivide(Vec w,Vec x,Vec y);	$w_i = x_i/y_i$
VecMDot(Vec x,int n,Vec y[],PetscScalar *r);	$r[i] = \bar{x}' * y[i]$
VecMTDot(Vec x,int n,Vec y[],PetscScalar *r);	$r[i] = x' * y[i]$
VecMAXPY(Vec y,int n, PetscScalar *a, Vec x[]);	$y = y + \sum_i a_i * x[i]$
VecMax(Vec x, int *idx, PetscReal *r);	$r = \max x_i$
VecMin(Vec x, int *idx, PetscReal *r);	$r = \min x_i$
VecAbs(Vec x);	$x_i = \|x_i\|$
VecReciprocal(Vec x);	$x_i = 1/x_i$
VecShift(Vec x,PetscScalar s);	$x_i = s + x_i$
VecSet(Vec x,PetscScalar alpha);	$x_i = \alpha$

Table 1: PETSc Vector Operations

Minor differences exist in the Fortran interface for VecGetArray() and VecRestoreArray(), as discussed in Section 12.1.3. It is important to note that VecGetArray() and VecRestoreArray() do *not* copy the vector elements; they merely give users direct access to the vector elements. Thus, these routines require essentially no time to call and can be used efficiently.

The number of elements stored locally can be accessed with

```
VecGetLocalSize(Vec v,int *size);
```

The global vector length can be determined by

```
VecGetSize(Vec v,int *size);
```

In addition to VecDot() and VecMDot() and VecNorm(), PETSc provides split phase versions of these that allow several independent inner products and/or norms to share the same communication (thus improving parallel efficiency). For example, one may have code such as

```
VecDot(Vec x,Vec y,PetscScalar *dot);
VecMDot(Vec x,PetscInt nv, Vec y[],PetscScalar *dot);
VecNorm(Vec x,NormType NORM_2,PetscReal *norm2);
VecNorm(Vec x,NormType NORM_1,PetscReal *norm1);
```

This code works fine, the problem is that it performs three separate parallel communication operations. Instead one can write

VecDotBegin(Vec x,Vec y,PetscScalar *dot);
VecMDotBegin(Vec x, PetscInt nv,Vec y[],PetscScalar *dot);
VecNormBegin(Vec x,NormType NORM_2,PetscReal *norm2);
VecNormBegin(Vec x,NormType NORM_1,PetscReal *norm1);
VecDotEnd(Vec x,Vec y,PetscScalar *dot);
VecMDotEnd(Vec x, PetscInt nv,Vec y[],PetscScalar *dot);
VecNormEnd(Vec x,NormType NORM_2,PetscReal *norm2);
VecNormEnd(Vec x,NormType NORM_1,PetscReal *norm1);

With this code, the communication is delayed until the first call to `VecxxxEnd()` at which a single MPI reduction is used to communicate all the required values. It is required that the calls to the `VecxxxEnd()` are performed in the same order as the calls to the `VecxxxBegin()`; however if you mistakenly make the calls in the wrong order PETSc will generate an error, informing you of this. There are additional routines VecTDotBegin() and VecTDotEnd(), VecMTDotBegin(), VecMTDotEnd().

Note: these routines use only MPI 1 functionality; so they do not allow you to overlap computation and communication (assuming no threads are spawned within a MPI process). Once MPI 2 implementations are more common we'll improve these routines to allow overlap of inner product and norm calculations with other calculations. Also currently these routines only work for the PETSc built in vector types.

2.3 Indexing and Ordering

When writing parallel PDE codes there is extra complexity caused by having multiple ways of indexing (numbering) and ordering objects such as vertices and degrees of freedom. For example, a grid generator or partitioner may renumber the nodes, requiring adjustment of the other data structures that refer to these objects; see Figure 9. In addition, local numbering (on a single process) of objects may be different than the global (cross-process) numbering. PETSc provides a variety of tools that help to manage the mapping among the various numbering systems. The two most basic are the AO (application ordering), which enables mapping between different global (cross-process) numbering schemes and the ISLocalToGlobalMapping, which allows mapping between local (on-process) and global (cross-process) numbering.

2.3.1 Application Orderings

In many applications it is desirable to work with one or more "orderings" (or numberings) of degrees of freedom, cells, nodes, etc. Doing so in a parallel environment is complicated by the fact that each process cannot keep complete lists of the mappings between different orderings. In addition, the orderings used in the PETSc linear algebra routines (often contiguous ranges) may not correspond to the "natural" orderings for the application.

PETSc provides certain utility routines that allow one to deal cleanly and efficiently with the various orderings. To define a new application ordering (called an AO in PETSc), one can call the routine

AOCreateBasic(MPI_Comm comm,int n,const int apordering[],const int petscordering[],AO *ao);

The arrays `apordering` and `petscordering`, respectively, contain a list of integers in the application ordering and their corresponding mapped values in the PETSc ordering. Each process can provide whatever subset of the ordering it chooses, but multiple processes should never contribute duplicate values. The argument `n` indicates the number of local contributed values.

For example, consider a vector of length five, where node 0 in the application ordering corresponds to node 3 in the PETSc ordering. In addition, nodes 1, 2, 3, and 4 of the application ordering correspond, respectively, to nodes 2, 1, 4, and 0 of the PETSc ordering. We can write this correspondence as

$$0, 1, 2, 3, 4 \rightarrow 3, 2, 1, 4, 0.$$

The user can create the PETSc-AO mappings in a number of ways. For example, if using two processes, one could call

AOCreateBasic(PETSC_COMM_WORLD,2,{0,3},{3,4},&ao);

on the first process and

AOCreateBasic(PETSC_COMM_WORLD,3,{1,2,4},{2,1,0},&ao);

on the other process.

Once the application ordering has been created, it can be used with either of the commands

AOPetscToApplication(AO ao,int n,int *indices);
AOApplicationToPetsc(AO ao,int n,int *indices);

Upon input, the n-dimensional array `indices` specifies the indices to be mapped, while upon output, `indices` contains the mapped values. Since we, in general, employ a parallel database for the AO mappings, it is crucial that all processes that called AOCreateBasic() also call these routines; these routines *cannot* be called by just a subset of processes in the MPI communicator that was used in the call to AOCreateBasic().

An alternative routine to create the application ordering, AO, is

AOCreateBasicIS(IS apordering,IS petscordering,AO *ao);

where index sets (see 2.5.1) are used instead of integer arrays.

The mapping routines

AOPetscToApplicationIS(AO ao,IS indices);
AOApplicationToPetscIS(AO ao,IS indices);

will map index sets (IS objects) between orderings. Both the `AOXxxToYyy()` and `AOXxxToYyyIS()` routines can be used regardless of whether the AO was created with a AOCreateBasic() or AOCreateBasicIS().

The AO context should be destroyed with AODestroy(AO *ao) and viewed with AOView(AO ao,PetscViewer viewer).

Although we refer to the two orderings as "PETSc" and "application" orderings, the user is free to use them both for application orderings and to maintain relationships among a variety of orderings by employing several AO contexts.

The `AOxxToxx()` routines allow negative entries in the input integer array. These entries are not mapped; they simply remain unchanged. This functionality enables, for example, mapping neighbor lists that use negative numbers to indicate nonexistent neighbors due to boundary conditions, etc.

2.3.2 Local to Global Mappings

In many applications one works with a global representation of a vector (usually on a vector obtained with VecCreateMPI()) and a local representation of the same vector that includes ghost points required for local computation. PETSc provides routines to help map indices from a local numbering scheme to the PETSc global numbering scheme. This is done via the following routines

ISLocalToGlobalMappingCreate(MPI_Comm comm,int bs,int N,int* globalnum,PetscCopyMode mode,ISLocalToGlobalM
ISLocalToGlobalMappingApply(ISLocalToGlobalMapping ctx,int n,int *in,int *out);
ISLocalToGlobalMappingApplyIS(ISLocalToGlobalMapping ctx,IS isin,IS* isout);
ISLocalToGlobalMappingDestroy(ISLocalToGlobalMapping *ctx);

Here N denotes the number of local indices, globalnum contains the global number of each local number, and ISLocalToGlobalMapping is the resulting PETSc object that contains the information needed to apply the mapping with either ISLocalToGlobalMappingApply() or ISLocalToGlobalMappingApplyIS().

Note that the ISLocalToGlobalMapping routines serve a different purpose than the AO routines. In the former case they provide a mapping from a local numbering scheme (including ghost points) to a global numbering scheme, while in the latter they provide a mapping between two global numbering schemes. In fact, many applications may use both AO and ISLocalToGlobalMapping routines. The AO routines are first used to map from an application global ordering (that has no relationship to parallel processing etc.) to the PETSc ordering scheme (where each process has a contiguous set of indices in the numbering). Then in order to perform function or Jacobian evaluations locally on each process, one works with a local numbering scheme that includes ghost points. The mapping from this local numbering scheme back to the global PETSc numbering can be handled with the ISLocalToGlobalMapping routines.

If one is given a list of block indices in a global numbering, the routine

ISGlobalToLocalMappingApplyBlock(ISLocalToGlobalMapping ctx,
ISGlobalToLocalMappingType type,int nin,int idxin[],int *nout,int idxout[]);

will provide a new list of indices in the local numbering. Again, negative values in idxin are left unmapped. But, in addition, if type is set to IS_GTOLM_MASK , then nout is set to nin and all global values in idxin that are not represented in the local to global mapping are replaced by -1. When type is set to IS_GTOLM_DROP, the values in idxin that are not represented locally in the mapping are not included in idxout, so that potentially nout is smaller than nin. One must pass in an array long enough to hold all the indices. One can call ISGlobalToLocalMappingApplyBlock() with idxout equal to NULL to determine the required length (returned in nout) and then allocate the required space and call ISGlobalToLocalMappingApplyBlock() a second time to set the values.

Often it is convenient to set elements into a vector using the local node numbering rather than the global node numbering (e.g., each process may maintain its own sublist of vertices and elements and number them locally). To set values into a vector with the local numbering, one must first call

VecSetLocalToGlobalMapping(Vec v,ISLocalToGlobalMapping ctx);

and then call

VecSetValuesLocal(Vec x,int n,const int indices[],const PetscScalar values[],INSERT_VALUES);

Now the indices use the local numbering, rather than the global, meaning the entries lie in $[0, n)$ where n is the local size of the vector.

2.4 Structured Grids Using Distributed Arrays

Distributed arrays (DMDAs), which are used in conjunction with PETSc vectors, are intended for use with *logically regular rectangular grids* when communication of nonlocal data is needed before certain local computations can occur. PETSc distributed arrays are designed only for the case in which data can be thought of as being stored in a standard multidimensional array; thus, DMDAs are *not* intended for parallelizing unstructured grid problems, etc. DAs are intended for communicating vector (field) information; they are not intended for storing matrices.

For example, a typical situation one encounters in solving PDEs in parallel is that, to evaluate a local function, f(x), each process requires its local portion of the vector x as well as its ghost points (the bordering portions of the vector that are owned by neighboring processes). Figure 8 illustrates the ghost points for the seventh process of a two-dimensional, regular parallel grid. Each box represents a process; the ghost points for the seventh process's local part of a parallel array are shown in gray.

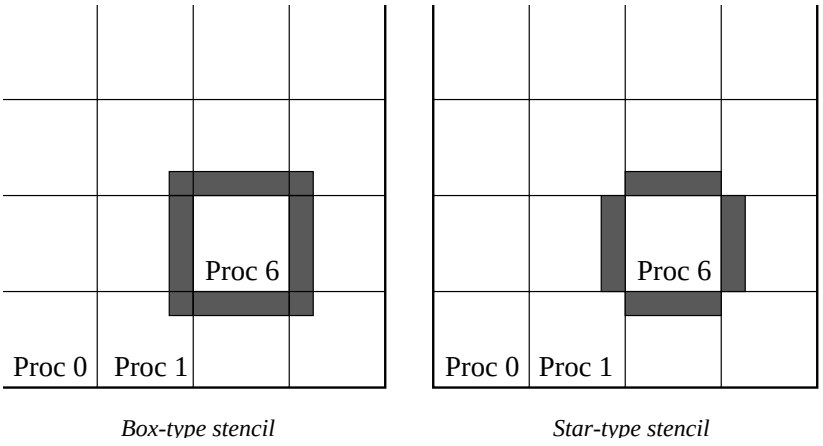

Figure 8: Ghost Points for Two Stencil Types on the Seventh Process

2.4.1 Creating Distributed Arrays

The PETSc DMDA object manages the parallel communication required while working with data stored in regular arrays. The actual data is stored in appropriately sized vector objects; the DMDA object only contains the parallel data layout information and communication information, however it may be used to create vectors and matrices with the proper layout.

One creates a distributed array communication data structure in two dimensions with the command

DMDACreate2d(MPI_Comm comm,DMDABoundaryType xperiod,DMDABoundaryType yperiod,DMDAStencilType st,in
int N,int m,int n,int dof,int s,int *lx,int *ly,DM *da);

The arguments `M` and `N` indicate the global numbers of grid points in each direction, while `m` and `n` denote the process partition in each direction; `m*n` must equal the number of processes in the MPI communicator, `comm`. Instead of specifying the process layout, one may use PETSC_DECIDE for `m` and `n` so that PETSc will determine the partition using MPI. The type of periodicity of the array is specified by `xperiod` and `yperiod`, which can be DMDA_BOUNDARY_NONE (no periodicity), DMDA_BOUNDARY_PERIODIC (periodic in that direction), DMDA_BOUNDARY_TWIST (periodic in that direction, but identified in reverse order), DMDA_BOUNDARY_GHOSTED , or DMDA_BOUNDARY_MIRROR. The argument `dof` indicates the number of degrees of freedom at each array point, and `s` is the stencil width (i.e., the width of the ghost point region). The optional arrays `lx` and `ly` may contain the number of nodes along the x and y axis for each cell, i.e. the dimension of `lx` is `m` and the dimension of `ly` is `n`; or NULL may be passed in.

Two types of distributed array communication data structures can be created, as specified by `st`. Star-type stencils that radiate outward only in the coordinate directions are indicated by DMDA_STENCIL_STAR, while box-type stencils are specified by DA_STENCIL_BOX. For example, for the two-dimensional case, DA_STENCIL_STAR with width 1 corresponds to the standard 5-point stencil, while DMDA_STENCIL_BOX with width 1 denotes the standard 9-point stencil. In both instances the ghost points are identical, the only difference being that with star-type stencils certain ghost points are ignored, decreasing substantially the number of messages sent. Note that the DMDA_STENCIL_STAR stencils can save interprocess communication in two and three dimensions.

These DMDA stencils have nothing directly to do with any finite difference stencils one might chose to use for a discretization; they only ensure that the correct values are in place for application of a user-defined finite difference stencil (or any other discretization technique).

The commands for creating distributed array communication data structures in one and three dimensions are analogous:

DMDACreate1d(MPI_Comm comm,DMDABoundaryType xperiod,int M,int w,int s,int *lc,DM *inra);
DMDACreate3d(MPI_Comm comm,DMDABoundaryType xperiod,DMDABoundaryType yperiod,
DMDABoundaryType zperiod, DMDAStencilType stencil_type,
int M,int N,int P,int m,int n,int p,int w,int s,int *lx,int *ly,int *lz,DM *inra);

The routines to create distributed arrays are collective, so that all processes in the communicator `comm` must call `DACreateXXX()`.

2.4.2 Local/Global Vectors and Scatters

Each DMDA object defines the layout of two vectors: a distributed global vector and a local vector that includes room for the appropriate ghost points. The DMDA object provides information about the size and layout of these vectors, but does not internally allocate any associated storage space for field values. Instead, the user can create vector objects that use the DMDA layout information with the routines

DMCreateGlobalVector(DM da,Vec *g);
DMCreateLocalVector(DM da,Vec *l);

These vectors will generally serve as the building blocks for local and global PDE solutions, etc. If additional vectors with such layout information are needed in a code, they can be obtained by duplicating `l` or `g` via VecDuplicate() or VecDuplicateVecs().

We emphasize that a distributed array provides the information needed to communicate the ghost value information between processes. In most cases, several different vectors can share the same communication information (or, in other words, can share a given DMDA). The design of the DMDA object makes this easy, as each DMDA operation may operate on vectors of the appropriate size, as obtained via DMCreateLocalVector() and DMCreateGlobalVector() or as produced by VecDuplicate(). As such, the DMDA scatter/gather operations (e.g., DMGlobalToLocalBegin()) require vector input/output arguments, as discussed below.

PETSc currently provides no container for multiple arrays sharing the same distributed array communication; note, however, that the `dof` parameter handles many cases of interest.

At certain stages of many applications, there is a need to work on a local portion of the vector, including the ghost points. This may be done by scattering a global vector into its local parts by using the two-stage commands

DMGlobalToLocalBegin(DM da,Vec g,InsertMode iora,Vec l);
DMGlobalToLocalEnd(DM da,Vec g,InsertMode iora,Vec l);

which allow the overlap of communication and computation. Since the global and local vectors, given by `g` and `l`, respectively, must be compatible with the distributed array, `da`, they should be generated by DMCreateGlobalVector() and DMCreateLocalVector() (or be duplicates of such a vector obtained via VecDuplicate()). The InsertMode can be either ADD_VALUES or INSERT_VALUES.

One can scatter the local patches into the distributed vector with the command

DMLocalToGlobalBegin(DM da,Vec l,InsertMode mode,Vec g); DMLocalToGlobalEnd(DM da,Vec l,InsertMode mode,Ve

In general this is used with an InsertMode of ADD_VALUES, because if one wishes to insert values into the global vector they should just access the global vector directly and put in the values.

A third type of distributed array scatter is from a local vector (including ghost points that contain irrelevant values) to a local vector with correct ghost point values. This scatter may be done by commands

DMDALocalToLocalBegin(DM da,Vec l1,InsertMode iora,Vec l2);
DMDALocalToLocalEnd(DM da,Vec l1,InsertMode iora,Vec l2);

Since both local vectors, `l1` and `l2`, must be compatible with the distributed array, `da`, they should be generated by DMCreateLocalVector() (or be duplicates of such vectors obtained via VecDuplicate()). The InsertMode can be either `ADD_VALUES` or `INSERT_VALUES`.

It is possible to directly access the vector scatter contexts (see below) used in the local-to-global (`ltog`), global-to-local (`gtol`), and local-to-local (`ltol`) scatters with the command

```
DMDAGetScatter(DM da,VecScatter *ltog,VecScatter *gtol,VecScatter *ltol);
```

Most users should not need to use these contexts.

2.4.3 Local (Ghosted) Work Vectors

In most applications the local ghosted vectors are only needed during user "function evaluations". PETSc provides an easy light-weight (requiring essentially no CPU time) way to obtain these work vectors and return them when they are no longer needed. This is done with the routines

```
DMGetLocalVector(DM da,Vec *l);
.... use the local vector l
DMRestoreLocalVector(DM da,Vec *l);
```

2.4.4 Accessing the Vector Entries for DMDA Vectors

PETSc provides an easy way to set values into the DMDA Vectors and access them using the natural grid indexing. This is done with the routines

```
DMDAVecGetArray(DM da,Vec l,void *array);
... use the array indexing it with 1 or 2 or 3 dimensions
... depending on the dimension of the DMDA
DMDAVecRestoreArray(DM da,Vec l,void *array); DMDAVecGetArrayRead(DM da,Vec l,void *array);
... use the array indexing it with 1 or 2 or 3 dimensions
... depending on the dimension of the DMDA
DMDAVecRestoreArrayRead(DM da,Vec l,void *array);

DMDAVecGetArrayDOF(DM da,Vec l,void *array);
... use the array indexing it with 1 or 2 or 3 dimensions
... depending on the dimension of the DMDA
DMDAVecRestoreArrayDOF(DM da,Vec l,void *array); DMDAVecGetArrayDOFRead(DM da,Vec l,void *array);
... use the array indexing it with 1 or 2 or 3 dimensions
... depending on the dimension of the DMDA
DMDAVecRestoreArrayDOFRead(DM da,Vec l,void *array);
```

where `array` is a multidimensional C array with the same dimension as `da`. The vector l can be either a global vector or a local vector. The `array` is accessed using the usual **global** indexing on the entire grid, but the user may **only** refer to the local and ghost entries of this array as all other entries are undefined. For example for a scalar problem in two dimensions one could do

```
PetscScalar **f,**u;
...
DMDAVecGetArray(DM da,Vec local,&u);
DMDAVecGetArray(DM da,Vec global,&f);
...
```

```
f[i][j] = u[i][j] - ...
...
DMDAVecRestoreArray(DM da,Vec local,&u);
DMDAVecRestoreArray(DM da,Vec global,&f);
```

The recommended approach for multi-component PDEs is to declare a struct representing the fields defined at each node of the grid, e.g.

```
typedef struct {
PetscScalar u,v,omega,temperature;
} Node;
```

and write residual evaluation using

```
Node **f,**u;
DMDAVecGetArray(DM da,Vec local,&u);
DMDAVecGetArray(DM da,Vec global,&f);
...
f[i][j].omega = ...
...
DMDAVecRestoreArray(DM da,Vec local,&u);
DMDAVecRestoreArray(DM da,Vec global,&f);
```

See `${PETSC_DIR}/src/snes/examples/tutorials/ex5.c` for a complete example and see `${PETSC_DIR}/src/snes/examples/tutorials/ex19.c` for an example for a multi-component PDE.

2.4.5 Grid Information

The global indices of the lower left corner of the local portion of the array as well as the local array size can be obtained with the commands

```
DMDAGetCorners(DM da,int *x,int *y,int *z,int *m,int *n,int *p);
DMDAGetGhostCorners(DM da,int *x,int *y,int *z,int *m,int *n,int *p);
```

The first version excludes any ghost points, while the second version includes them. The routine DMDAGetGhostCorners() deals with the fact that subarrays along boundaries of the problem domain have ghost points only on their interior edges, but not on their boundary edges.

When either type of stencil is used, DMDA_STENCIL_STAR or DA_STENCIL_BOX, the local vectors (with the ghost points) represent rectangular arrays, including the extra corner elements in the DMDA_STENCIL_STAR case. This configuration provides simple access to the elements by employing two- (or three-) dimensional indexing. The only difference between the two cases is that when DMDA_STENCIL_STAR is used, the extra corner components are *not* scattered between the processes and thus contain undefined values that should *not* be used.

To assemble global stiffness matrices, one needs either

- the global node number of each local node including the ghost nodes cqn be obtained by first calling

  ```
  DMGetLocalToGlobalMapping(DM da,ISLocalToGlobalMapping *map);
  ```

  followed by

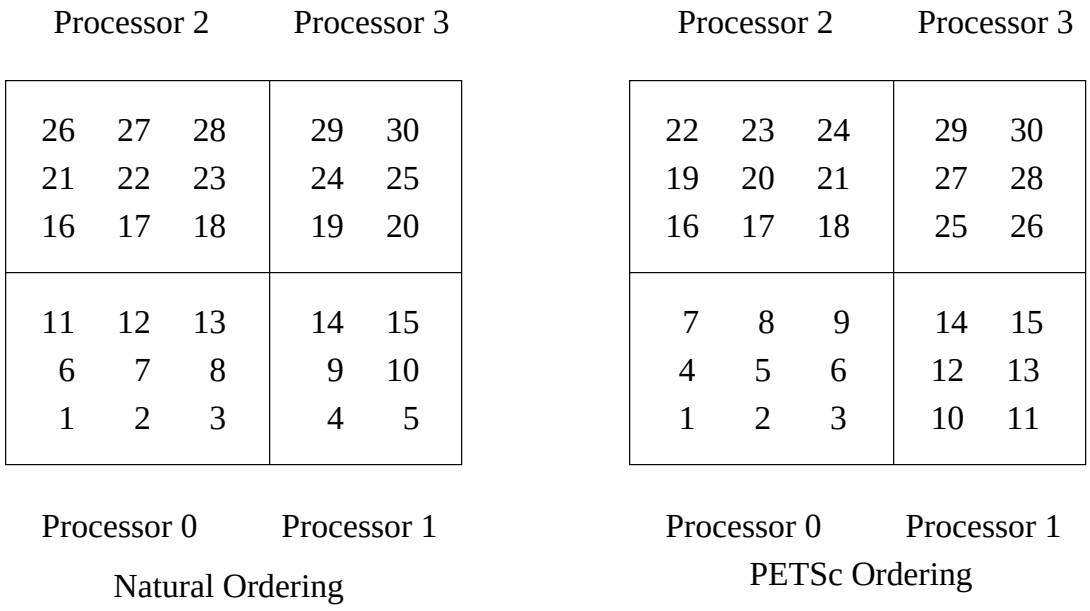

Figure 9: Natural Ordering and PETSc Ordering for a 2D Distributed Array (Four Processes)

VecSetLocalToGlobalMapping(Vec v,ISLocalToGlobalMapping map);
MatSetLocalToGlobalMapping(Mat A,ISLocalToGlobalMapping map);

Now entries may be added to the vector and matrix using the local numbering and VecSetValuesLocal() and MatSetValuesLocal().

Since the global ordering that PETSc uses to manage its parallel vectors (and matrices) does not usually correspond to the "natural" ordering of a two- or three-dimensional array, the DMDA structure provides an application ordering AO (see Section 2.3.1) that maps between the natural ordering on a rectangular grid and the ordering PETSc uses to parallelize. This ordering context can be obtained with the command

DMDAGetAO(DM da,AO *ao);

In Figure 9 we indicate the orderings for a two-dimensional distributed array, divided among four processes.

The example `${PETSC_DIR}/src/snes/examples/tutorials/ex5.c`, illustrates the use of a distributed array in the solution of a nonlinear problem. The analogous Fortran program is `${PETSC_DIR}/src/snes/examples/tutorials/ex5f.F`; see Chapter 5 for a discussion of the nonlinear solvers.

2.5 Software for Managing Vectors Related to Unstructured Grids

2.5.1 Index Sets

To facilitate general vector scatters and gathers used, for example, in updating ghost points for problems defined on unstructured grids, PETSc employs the concept of an index set. An index set, which is a generalization of a set of integer indices, is used to define scatters, gathers, and similar operations on vectors and matrices.

The following command creates a index set based on a list of integers:

ISCreateGeneral(MPI_Comm comm,int n,int *indices,PetscCopyMode mode, IS *is);

When mode is PETSC_COPY_VALUES this routine copies the `n` indices passed to it by the integer array `indices`. Thus, the user should be sure to free the integer array `indices` when it is no longer needed, perhaps directly after the call to ISCreateGeneral(). The communicator, `comm`, should consist of all processes that will be using the IS.

Another standard index set is defined by a starting point (`first`) and a stride (`step`), and can be created with the command

ISCreateStride(MPI_Comm comm,int n,int first,int step,IS *is);

Index sets can be destroyed with the command

ISDestroy(IS &is);

On rare occasions the user may need to access information directly from an index set. Several commands assist in this process:

ISGetSize(IS is,int *size);
ISStrideGetInfo(IS is,int *first,int *stride);
ISGetIndices(IS is,int **indices);

The function ISGetIndices() returns a pointer to a list of the indices in the index set. For certain index sets, this may be a temporary array of indices created specifically for a given routine. Thus, once the user finishes using the array of indices, the routine

ISRestoreIndices(IS is, int **indices);

should be called to ensure that the system can free the space it may have used to generate the list of indices.

A blocked version of the index sets can be created with the command

ISCreateBlock(MPI_Comm comm,int bs,int n,int *indices,PetscCopyMode mode, IS *is);

This version is used for defining operations in which each element of the index set refers to a block of `bs` vector entries. Related routines analogous to those described above exist as well, including ISBlockGetIndices(), ISBlockGetSize(), ISBlockGetLocalSize(), ISGetBlockSize(). See the man pages for details.

2.5.2 Scatters and Gathers

PETSc vectors have full support for general scatters and gathers. One can select any subset of the components of a vector to insert or add to any subset of the components of another vector. We refer to these operations as generalized scatters, though they are actually a combination of scatters and gathers.

To copy selected components from one vector to another, one uses the following set of commands:

VecScatterCreate(Vec x,IS ix,Vec y,IS iy,VecScatter *ctx);
VecScatterBegin(VecScatter ctx,Vec x,Vec y,INSERT_VALUES,SCATTER_FORWARD);
VecScatterEnd(VecScatter ctx,Vec x,Vec y,INSERT_VALUES,SCATTER_FORWARD);
VecScatterDestroy(VecScatter *ctx);

Here `ix` denotes the index set of the first vector, while `iy` indicates the index set of the destination vector. The vectors can be parallel or sequential. The only requirements are that the number of entries in the index set of the first vector, `ix`, equal the number in the destination index set, `iy`, and that the vectors be long enough to contain all the indices referred to in the index sets. The argument INSERT_VALUES specifies that the vector elements will be inserted into the specified locations of the destination vector, overwriting any existing values. To add the components, rather than insert them, the user should select the option ADD_VALUES instead of INSERT_VALUES.

```
Vec p, x; /* initial vector, destination vector */
VecScatter scatter; /* scatter context */
IS from, to; /* index sets that define the scatter */
PetscScalar *values;
int idx_from[] = {100,200}, idx_to[] = {0,1};
VecCreateSeq(PETSC_COMM_SELF,2,&x);
ISCreateGeneral(PETSC_COMM_SELF,2,idx_from,PETSC_COPY_VALUES,&from);
ISCreateGeneral(PETSC_COMM_SELF,2,idx_to,PETSC_COPY_VALUES,&to);
VecScatterCreate(p,from,x,to,&scatter);
VecScatterBegin(scatter,p,x,INSERT_VALUES,SCATTER_FORWARD);
VecScatterEnd(scatter,p,x,INSERT_VALUES,SCATTER_FORWARD);
VecGetArray(x,&values);
ISDestroy(&from);
ISDestroy(&to);
VecScatterDestroy(&scatter);
```

Figure 10: Example Code for Vector Scatters

To perform a conventional gather operation, the user simply makes the destination index set, `iy`, be a stride index set with a stride of one. Similarly, a conventional scatter can be done with an initial (sending) index set consisting of a stride. The scatter routines are collective operations (i.e. all processes that own a parallel vector *must* call the scatter routines). When scattering from a parallel vector to sequential vectors, each process has its own sequential vector that receives values from locations as indicated in its own index set. Similarly, in scattering from sequential vectors to a parallel vector, each process has its own sequential vector that makes contributions to the parallel vector.

Caution: When INSERT_VALUES is used, if two different processes contribute different values to the same component in a parallel vector, either value may end up being inserted. When ADD_VALUES is used, the correct sum is added to the correct location.

In some cases one may wish to "undo" a scatter, that is perform the scatter backwards switching the roles of the sender and receiver. This is done by using

```
VecScatterBegin(VecScatter ctx,Vec y,Vec x,INSERT_VALUES,SCATTER_REVERSE);
VecScatterEnd(VecScatter ctx,Vec y,Vec x,INSERT_VALUES,SCATTER_REVERSE);
```

Note that the roles of the first two arguments to these routines must be swapped whenever the SCATTER_REVERSE option is used.

Once a VecScatter object has been created it may be used with any vectors that have the appropriate parallel data layout. That is, one can call VecScatterBegin() and VecScatterEnd() with different vectors than used in the call to VecScatterCreate() so long as they have the same parallel layout (number of elements on each process are the same). Usually, these "different" vectors would have been obtained via calls to VecDuplicate() from the original vectors used in the call to VecScatterCreate().

There is a PETSc routine that is nearly the opposite of VecSetValues(), that is, VecGetValues(), but it can only get local values from the vector. To get off process values, the user should create a new vector where the components are to be stored and perform the appropriate vector scatter. For example, if one desires to obtain the values of the 100th and 200th entries of a parallel vector, `p`, one could use a code such as that within Figure 10. In this example, the values of the 100th and 200th components are placed in the array values. In this example each process now has the 100th and 200th component, but obviously each process could gather any elements it needed, or none by creating an index set with no entries.

The scatter comprises two stages, in order to allow overlap of communication and computation. The introduction of the VecScatter context allows the communication patterns for the scatter to be computed once and then reused repeatedly. Generally, even setting up the communication for a scatter requires communication; hence, it is best to reuse such information when possible.

2.5.3 Scattering Ghost Values

The scatters provide a very general method for managing the communication of required ghost values for unstructured grid computations. One scatters the global vector into a local "ghosted" work vector, performs the computation on the local work vectors, and then scatters back into the global solution vector. In the simplest case this may be written as

Function: (Input Vec globalin, Output Vec globalout)

```
VecScatterBegin(VecScatter scatter,Vec globalin,Vec localin,InsertMode INSERT_VALUES,
                ScatterMode SCATTER_FORWARD);
VecScatterEnd(VecScatter scatter,Vec globalin,Vec localin,InsertMode INSERT_VALUES,
                ScatterMode SCATTER_FORWARD);
/* For example, do local calculations from localin to localout */
VecScatterBegin(VecScatter scatter,Vec localout,Vec globalout,InsertMode ADD_VALUES,
                ScatterMode SCATTER_REVERSE);
VecScatterEnd(VecScatter scatter,Vec localout,Vec globalout,InsertMode ADD_VALUES,
                ScatterMode SCATTER_REVERSE);
```

2.5.4 Vectors with Locations for Ghost Values

There are two minor drawbacks to the basic approach described above:

- the extra memory requirement for the local work vector, `localin`, which duplicates the memory in `globalin`, and
- the extra time required to copy the local values from `localin` to `globalin`.

An alternative approach is to allocate global vectors with space preallocated for the ghost values; this may be done with either

```
VecCreateGhost(MPI_Comm comm,int n,int N,int nghost,int *ghosts,Vec *vv)
```

or

```
VecCreateGhostWithArray(MPI_Comm comm,int n,int N,int nghost,int *ghosts,
PetscScalar *array,Vec *vv)
```

Here `n` is the number of local vector entries, `N` is the number of global entries (or NULL) and `nghost` is the number of ghost entries. The array `ghosts` is of size `nghost` and contains the global vector location for each local ghost location. Using VecDuplicate() or VecDuplicateVecs() on a ghosted vector will generate additional ghosted vectors.

In many ways a ghosted vector behaves just like any other `MPI` vector created by VecCreateMPI(), the difference is that the ghosted vector has an additional "local" representation that allows one to access the ghost locations. This is done through the call to

```
VecGhostGetLocalForm(Vec g,Vec *l);
```

The vector `l` is a sequential representation of the parallel vector `g` that shares the same array space (and hence numerical values); but allows one to access the "ghost" values past "the end of the" array. Note that one access the entries in `l` using the local numbering of elements and ghosts, while they are accessed in `g` using the global numbering.

A common usage of a ghosted vector is given by

```
VecGhostUpdateBegin(Vec globalin,InsertMode INSERT_VALUES,
                    ScatterMode SCATTER_FORWARD);
VecGhostUpdateEnd(Vec globalin,InsertMode INSERT_VALUES,
                    ScatterMode SCATTER_FORWARD);
VecGhostGetLocalForm(Vec globalin,Vec *localin);
VecGhostGetLocalForm(Vec globalout,Vec *localout);
/*
Do local calculations from localin to localout
*/
VecGhostRestoreLocalForm(Vec globalin,Vec *localin);
VecGhostRestoreLocalForm(Vec globalout,Vec *localout);
VecGhostUpdateBegin(Vec globalout,InsertMode ADD_VALUES,
                    ScatterMode SCATTER_REVERSE);
VecGhostUpdateEnd(Vec globalout,InsertMode ADD_VALUES,
                    ScatterMode SCATTER_REVERSE);
```

The routines VecGhostUpdateBegin() and VecGhostUpdateEnd() are equivalent to the routines VecScatterBegin() and VecScatterEnd() above except that since they are scattering into the ghost locations, they do not need to copy the local vector values, which are already in place. In addition, the user does not have to allocate the local work vector, since the ghosted vector already has allocated slots to contain the ghost values.

The input arguments INSERT_VALUES and SCATTER_FORWARD cause the ghost values to be correctly updated from the appropriate process. The arguments ADD_VALUES and SCATTER_REVERSE update the "local" portions of the vector from all the other processes' ghost values. This would be appropriate, for example, when performing a finite element assembly of a load vector.

Section 3.5 discusses the important topic of partitioning an unstructured grid.

Chapter 3

Matrices

PETSc provides a variety of matrix implementations because no single matrix format is appropriate for all problems. Currently we support dense storage and compressed sparse row storage (both sequential and parallel versions), as well as several specialized formats. Additional formats can be added.

This chapter describes the basics of using PETSc matrices in general (regardless of the particular format chosen) and discusses tips for efficient use of the several simple uniprocess and parallel matrix types. The use of PETSc matrices involves the following actions: create a particular type of matrix, insert values into it, process the matrix, use the matrix for various computations, and finally destroy the matrix. The application code does not need to know or care about the particular storage formats of the matrices.

3.1 Creating and Assembling Matrices

The simplest routine for forming a PETSc matrix, `A`, is followed by

MatCreate(MPI_Comm comm,Mat *A) MatSetSizes(Mat A,int m,int n,int M,int N)

This routine generates a sequential matrix when running one process and a parallel matrix for two or more processes; the particular matrix format is set by the user via options database commands. The user specifies either the global matrix dimensions, given by `M` and `N` or the local dimensions, given by `m` and `n` while PETSc completely controls memory allocation. This routine facilitates switching among various matrix types, for example, to determine the format that is most efficient for a certain application. By default, MatCreate() employs the sparse AIJ format, which is discussed in detail Section 3.1.1. See the manual pages for further information about available matrix formats.

To insert or add entries to a matrix, one can call a variant of MatSetValues(), either

MatSetValues(Mat A,int m,const int idxm[],int n,const int idxn[],const PetscScalar values[],
INSERT_VALUES);

or

MatSetValues(Mat A,int m,const int idxm[],int n,const int idxn[],const PetscScalar values[],
ADD_VALUES);

This routine inserts or adds a logically dense subblock of dimension `m*n` into the matrix. The integer indices `idxm` and `idxn`, respectively, indicate the global row and column numbers to be inserted. MatSetValues() uses the standard C convention, where the row and column matrix indices begin with zero *regardless of the storage format employed*. The array `values` is logically two-dimensional, containing the values that are to be inserted. By default the values are given in row major order, which is the opposite of the Fortran convention, meaning that the value to be put in row `idxm[i]` and column `idxn[j]` is located in `values[i*n+j]`. To allow the insertion of values in column major order, one can call the command

MatSetOption(Mat A,MAT_ROW_ORIENTED,PETSC_FALSE);

Warning: Several of the sparse implementations do *not* currently support the column-oriented option.

This notation should not be a mystery to anyone. For example, to insert one matrix into another when using MATLAB, one uses the command `A(im,in) = B;` where `im` and `in` contain the indices for the rows and columns. This action is identical to the calls above to MatSetValues().

When using the block compressed sparse row matrix format (MATSEQBAIJ or MATMPIBAIJ), one can insert elements more efficiently using the block variant, MatSetValuesBlocked() or MatSetValuesBlocked-Local().

The function MatSetOption() accepts several other inputs; see the manual page for details.

After the matrix elements have been inserted or added into the matrix, they must be processed (also called assembled) before they can be used. The routines for matrix processing are

MatAssemblyBegin(Mat A,MAT_FINAL_ASSEMBLY);
MatAssemblyEnd(Mat A,MAT_FINAL_ASSEMBLY);

By placing other code between these two calls, the user can perform computations while messages are in transit. Calls to MatSetValues() with the INSERT_VALUES and ADD_VALUES options *cannot* be mixed without intervening calls to the assembly routines. For such intermediate assembly calls the second routine argument typically should be `MAT_FLUSH_ASSEMBLY`, which omits some of the work of the full assembly process. `MAT_FINAL_ASSEMBLY` is required only in the last matrix assembly before a matrix is used.

Even though one may insert values into PETSc matrices without regard to which process eventually stores them, for efficiency reasons we usually recommend generating most entries on the process where they are destined to be stored. To help the application programmer with this task for matrices that are distributed across the processes by ranges, the routine

MatGetOwnershipRange(Mat A,int *first_row,int *last_row);

informs the user that all rows from `first_row` to `last_row-1` (since the value returned in `last_row` is one more than the global index of the last local row) will be stored on the local process.

In the sparse matrix implementations, once the assembly routines have been called, the matrices are compressed and can be used for matrix-vector multiplication, etc. Any space for preallocated nonzeros that was not filled by a call to MatSetValues() or a related routine is compressed out by assembling with `MAT_FINAL_ASSEMBLY`. If you intend to use that extra space later, be sure to insert explicit zeros before assembling with `MAT_FINAL_ASSEMBLY` so the space will not be compressed out. Once the matrix has been assembled, inserting new values will be expensive since it will require copies and possible memory allocation.

If one wishes to repeatedly assemble matrices that retain the same nonzero pattern (such as within a nonlinear or time-dependent problem), the option

MatSetOption(Mat A,MAT_NEW_NONZERO_LOCATIONS,PETSC_FALSE);

should be specified after the first matrix has been fully assembled. This option ensures that certain data structures and communication information will be reused (instead of regenerated) during successive steps, thereby increasing efficiency. See `${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex5.c` for a simple example of solving two linear systems that use the same matrix data structure.

3.1.1 Sparse Matrices

The default matrix representation within PETSc is the general sparse AIJ format (also called the Yale sparse matrix format or compressed sparse row format, CSR). This section discusses tips for *efficiently* using this matrix format for large-scale applications. Additional formats (such as block compressed row and block

diagonal storage, which are generally much more efficient for problems with multiple degrees of freedom per node) are discussed below. Beginning users need not concern themselves initially with such details and may wish to proceed directly to Section 3.2. However, when an application code progresses to the point of tuning for efficiency and/or generating timing results, it is *crucial* to read this information.

Sequential AIJ Sparse Matrices

In the PETSc AIJ matrix formats, we store the nonzero elements by rows, along with an array of corresponding column numbers and an array of pointers to the beginning of each row. Note that the diagonal matrix entries are stored with the rest of the nonzeros (not separately).

To create a sequential AIJ sparse matrix, `A`, with `m` rows and `n` columns, one uses the command

MatCreateSeqAIJ(PETSC_COMM_SELF,int m,int n,int nz,int *nnz,Mat *A);

where `nz` or `nnz` can be used to preallocate matrix memory, as discussed below. The user can set `nz=0` and `nnz`=NULL for PETSc to control all matrix memory allocation.

The sequential and parallel AIJ matrix storage formats by default employ *i-nodes* (identical nodes) when possible. We search for consecutive rows with the same nonzero structure, thereby reusing matrix information for increased efficiency. Related options database keys are `-mat_no_inode` (do not use inodes) and `-mat_inode_limit <limit>` (set inode limit (max limit=5)). Note that problems with a single degree of freedom per grid node will automatically not use I-nodes.

By default the internal data representation for the AIJ formats employs zero-based indexing. For compatibility with standard Fortran storage, thus enabling use of external Fortran software packages such as SPARSKIT, the option `-mat_aij_oneindex` enables one-based indexing, where the stored row and column indices begin at one, not zero. All user calls to PETSc routines, regardless of this option, use zero-based indexing.

Preallocation of Memory for Sequential AIJ Sparse Matrices

The dynamic process of allocating new memory and copying from the old storage to the new is *intrinsically very expensive*. Thus, to obtain good performance when assembling an AIJ matrix, it is crucial to preallocate the memory needed for the sparse matrix. The user has two choices for preallocating matrix memory via MatCreateSeqAIJ().

One can use the scalar `nz` to specify the expected number of nonzeros for each row. This is generally fine if the number of nonzeros per row is roughly the same throughout the matrix (or as a quick and easy first step for preallocation). If one underestimates the actual number of nonzeros in a given row, then during the assembly process PETSc will automatically allocate additional needed space. However, this extra memory allocation can slow the computation,

If different rows have very different numbers of nonzeros, one should attempt to indicate (nearly) the exact number of elements intended for the various rows with the optional array, `nnz` of length `m`, where `m` is the number of rows, for example

```
int nnz[m];
nnz[0] = <nonzeros in row 0>
nnz[1] = <nonzeros in row 1>
....
nnz[m-1] = <nonzeros in row m-1>
```

In this case, the assembly process will require no additional memory allocations if the `nnz` estimates are correct. If, however, the `nnz` estimates are incorrect, PETSc will automatically obtain the additional needed space, at a slight loss of efficiency.

Using the array `nnz` to preallocate memory is especially important for efficient matrix assembly if the number of nonzeros varies considerably among the rows. One can generally set `nnz` either by knowing in advance the problem structure (e.g., the stencil for finite difference problems on a structured grid) or by precomputing the information by using a segment of code similar to that for the regular matrix assembly. The overhead of determining the `nnz` array will be quite small compared with the overhead of the inherently expensive mallocs and moves of data that are needed for dynamic allocation during matrix assembly. Always guess high if exact value is not known (since extra space is cheaper than too little).

Thus, when assembling a sparse matrix with very different numbers of nonzeros in various rows, one could proceed as follows for finite difference methods:

- Allocate integer array `nnz`.
- Loop over grid, counting the expected number of nonzeros for the row(s)
  associated with the various grid points.
- Create the sparse matrix via MatCreateSeqAIJ() or alternative.
- Loop over the grid, generating matrix entries and inserting in matrix via MatSetValues().

For (vertex-based) finite element type calculations, an analogous procedure is as follows:

- Allocate integer array `nnz`.
- Loop over vertices, computing the number of neighbor vertices, which determines the
  number of nonzeros for the corresponding matrix row(s).
- Create the sparse matrix via MatCreateSeqAIJ() or alternative.
- Loop over elements, generating matrix entries and inserting in matrix via MatSetValues().

The `-info` option causes the routines MatAssemblyBegin() and MatAssemblyEnd() to print information about the success of the preallocation. Consider the following example for the MATSEQAIJ matrix format:

```
MatAssemblyEnd_SeqAIJ:Matrix size 10 X 10; storage space:20 unneeded, 100 used
MatAssemblyEnd_SeqAIJ:Number of mallocs during MatSetValues is 0
```

The first line indicates that the user preallocated 120 spaces but only 100 were used. The second line indicates that the user preallocated enough space so that PETSc did not have to internally allocate additional space (an expensive operation). In the next example the user did not preallocate sufficient space, as indicated by the fact that the number of mallocs is very large (bad for efficiency):

```
MatAssemblyEnd_SeqAIJ:Matrix size 10 X 10; storage space:47 unneeded, 1000 used
MatAssemblyEnd_SeqAIJ:Number of mallocs during MatSetValues is 40000
```

Although at first glance such procedures for determining the matrix structure in advance may seem unusual, they are actually very efficient because they alleviate the need for dynamic construction of the matrix data structure, which can be very expensive.

Parallel AIJ Sparse Matrices

Parallel sparse matrices with the AIJ format can be created with the command

```
MatCreateAIJ(MPI_Comm comm,int m,int n,int M,int N,int d_nz,
             int *d_nnz, int o_nz,int *o_nnz,Mat *A);
```

A is the newly created matrix, while the arguments m, M, and N, indicate the number of local rows and the number of global rows and columns, respectively. In the PETSc partitioning scheme, all the matrix columns are local and n is the number of columns corresponding to local part of a parallel vector. Either the local or global parameters can be replaced with PETSC_DECIDE, so that PETSc will determine them. The matrix is stored with a fixed number of rows on each process, given by m, or determined by PETSc if m is PETSC_DECIDE.

If PETSC_DECIDE is not used for the arguments m and n, then the user must ensure that they are chosen to be compatible with the vectors. To do this, one first considers the matrix-vector product $y = Ax$. The m that is used in the matrix creation routine MatCreateAIJ() must match the local size used in the vector creation routine VecCreateMPI() for y. Likewise, the n used must match that used as the local size in VecCreateMPI() for x.

The user must set d_nz=0, o_nz=0, d_nnz=NULL, and o_nnz=NULL for PETSc to control dynamic allocation of matrix memory space. Analogous to nz and nnz for the routine MatCreateSeqAIJ(), these arguments optionally specify nonzero information for the diagonal (d_nz and d_nnz) and off-diagonal (o_nz and o_nnz) parts of the matrix. For a square global matrix, we define each process's diagonal portion to be its local rows and the corresponding columns (a square submatrix); each process's off-diagonal portion encompasses the remainder of the local matrix (a rectangular submatrix). The rank in the MPI communicator determines the absolute ordering of the blocks. That is, the process with rank 0 in the communicator given to MatCreateAIJ() contains the top rows of the matrix; the i^{th} process in that communicator contains the i^{th} block of the matrix.

Preallocation of Memory for Parallel AIJ Sparse Matrices

As discussed above, preallocation of memory is critical for achieving good performance during matrix assembly, as this reduces the number of allocations and copies required. We present an example for three processes to indicate how this may be done for the MATMPIAIJ matrix format. Consider the 8 by 8 matrix, which is partitioned by default with three rows on the first process, three on the second and two on the third.

$$\left(\begin{array}{ccc|ccc|cc} 1 & 2 & 0 & 0 & 3 & 0 & 0 & 4 \\ 0 & 5 & 6 & 7 & 0 & 0 & 8 & 0 \\ 9 & 0 & 10 & 11 & 0 & 0 & 12 & 0 \\ \hline 13 & 0 & 14 & 15 & 16 & 17 & 0 & 0 \\ 0 & 18 & 0 & 19 & 20 & 21 & 0 & 0 \\ 0 & 0 & 0 & 22 & 23 & 0 & 24 & 0 \\ \hline 25 & 26 & 27 & 0 & 0 & 28 & 29 & 0 \\ 30 & 0 & 0 & 31 & 32 & 33 & 0 & 34 \end{array}\right)$$

The "diagonal" submatrix, d, on the first process is given by

$$\left(\begin{array}{ccc} 1 & 2 & 0 \\ 0 & 5 & 6 \\ 9 & 0 & 10 \end{array}\right),$$

while the "off-diagonal" submatrix, o, matrix is given by

$$\left(\begin{array}{ccccc} 0 & 3 & 0 & 0 & 4 \\ 7 & 0 & 0 & 8 & 0 \\ 11 & 0 & 0 & 12 & 0 \end{array}\right).$$

For the first process one could set d_nz to 2 (since each row has 2 nonzeros) or, alternatively, set d_nnz to {2,2,2}. The o_nz could be set to 2 since each row of the o matrix has 2 nonzeros, or o_nnz could be set to {2,2,2}.

For the second process the d submatrix is given by

$$\begin{pmatrix} 15 & 16 & 17 \\ 19 & 20 & 21 \\ 22 & 23 & 0 \end{pmatrix}.$$

Thus, one could set d_nz to 3, since the maximum number of nonzeros in each row is 3, or alternatively one could set d_nnz to {3,3,2}, thereby indicating that the first two rows will have 3 nonzeros while the third has 2. The corresponding o submatrix for the second process is

$$\begin{pmatrix} 13 & 0 & 14 & 0 & 0 \\ 0 & 18 & 0 & 0 & 0 \\ 0 & 0 & 0 & 24 & 0 \end{pmatrix}$$

so that one could set o_nz to 2 or o_nnz to {2,1,1}.

Note that the user never directly works with the d and o submatrices, except when preallocating storage space as indicated above. Also, the user need not preallocate exactly the correct amount of space; as long as a sufficiently close estimate is given, the high efficiency for matrix assembly will remain.

As described above, the option -info will print information about the success of preallocation during matrix assembly. For the MATMPIAIJ and MATMPIBAIJ formats, PETSc will also list the number of elements owned by on each process that were generated on a different process. For example, the statements

MatAssemblyBegin_MPIAIJ:Stash has 10 entries, uses 0 mallocs
MatAssemblyBegin_MPIAIJ:Stash has 3 entries, uses 0 mallocs
MatAssemblyBegin_MPIAIJ:Stash has 5 entries, uses 0 mallocs

indicate that very few values have been generated on different processes. On the other hand, the statements

MatAssemblyBegin_MPIAIJ:Stash has 100000 entries, uses 100 mallocs
MatAssemblyBegin_MPIAIJ:Stash has 77777 entries, uses 70 mallocs

indicate that many values have been generated on the "wrong" processes. This situation can be very inefficient, since the transfer of values to the "correct" process is generally expensive. By using the command MatGetOwnershipRange() in application codes, the user should be able to generate most entries on the owning process.

Note: It is fine to generate some entries on the "wrong" process. Often this can lead to cleaner, simpler, less buggy codes. One should never make code overly complicated in order to generate all values locally. Rather, one should organize the code in such a way that *most* values are generated locally.

3.1.2 Dense Matrices

PETSc provides both sequential and parallel dense matrix formats, where each process stores its entries in a column-major array in the usual Fortran style. To create a sequential, dense PETSc matrix, A of dimensions m by n, the user should call

MatCreateSeqDense(PETSC_COMM_SELF,int m,int n,PetscScalar *data,Mat *A);

The variable data enables the user to optionally provide the location of the data for matrix storage (intended for Fortran users who wish to allocate their own storage space). Most users should merely set data to NULL for PETSc to control matrix memory allocation. To create a parallel, dense matrix, A, the user should call

MatCreateDense(MPI_Comm comm,int m,int n,int M,int N,PetscScalar *data,Mat *A)

The arguments m, n, M, and N, indicate the number of local rows and columns and the number of global rows and columns, respectively. Either the local or global parameters can be replaced with PETSC_DECIDE, so that PETSc will determine them. The matrix is stored with a fixed number of rows on each process, given by m, or determined by PETSc if m is PETSC_DECIDE.

PETSc does not provide parallel dense direct solvers. Our focus is on sparse iterative solvers.

3.1.3 Block Matrices

Block matrices arise when coupling variables with different meaning, especially when solving problems with constraints (e.g. incompressible flow) and "multi-physics" problems. Usually the number of blocks is small and each block is partitioned in parallel. We illustrate for a 3×3 system with components labeled a, b, c. With some numbering of unknowns, the matrix could be written as

$$\begin{pmatrix} A_{aa} & A_{ab} & A_{ac} \\ A_{ba} & A_{bb} & A_{bc} \\ A_{ca} & A_{cb} & A_{cc} \end{pmatrix}.$$

There are two fundamentally different ways that this matrix could be stored, as a single assembled sparse matrix where entries from all blocks are merged together ("monolithic"), or as separate assembled matrices for each block ("nested"). These formats have different performance characteristics depending on the operation being performed. In particular, many preconditioners require a monolithic format, but some that are very effective for solving block systems (see Section 4.5) are more efficient when a nested format is used. In order to stay flexible, we would like to be able to use the same code to assemble block matrices in both monolithic and nested formats. Additionally, for software maintainability and testing, especially in a multi-physics context where different groups might be responsible for assembling each of the blocks, it is desirable to be able to use exactly the same code to assemble a single block independently as to assemble it as part of a larger system. To do this, we introduce the four spaces shown in Figure 11.

- The monolithic global space is the space in which the Krylov and Newton solvers operate, with collective semantics across the entire block system.
- The split global space splits the blocks apart, but each split still has collective semantics.
- The split local space adds ghost points and separates the blocks. Operations in this space can be performed with no parallel communication. This is often the most natural, and certainly the most powerful, space for matrix assembly code.
- The monolithic local space can be thought of as adding ghost points to the monolithic global space, but it is often more natural to use it simply as a concatenation of split local spaces on each process. It is not common to explicitly manipulate vectors or matrices in this space (at least not during assembly), but it is a useful for declaring which part of a matrix is being assembled.

The key to format-independent assembly is the function

MatGetLocalSubMatrix(Mat A,IS isrow,IS iscol,Mat *submat);

which provides a "view" submat into a matrix A that operates in the monolithic global space. The submat transforms from the split local space defined by iscol to the split local space defined by isrow. The index sets specify the parts of the monolithic local space that submat should operate in. If a nested matrix format is used, then MatGetLocalSubMatrix() finds the nested block and returns it without making any copies. In this case, submat is fully functional and has a parallel communicator. If a monolithic matrix format is used, then MatGetLocalSubMatrix() returns a proxy matrix on PETSC_COMM_SELF that does not provide values

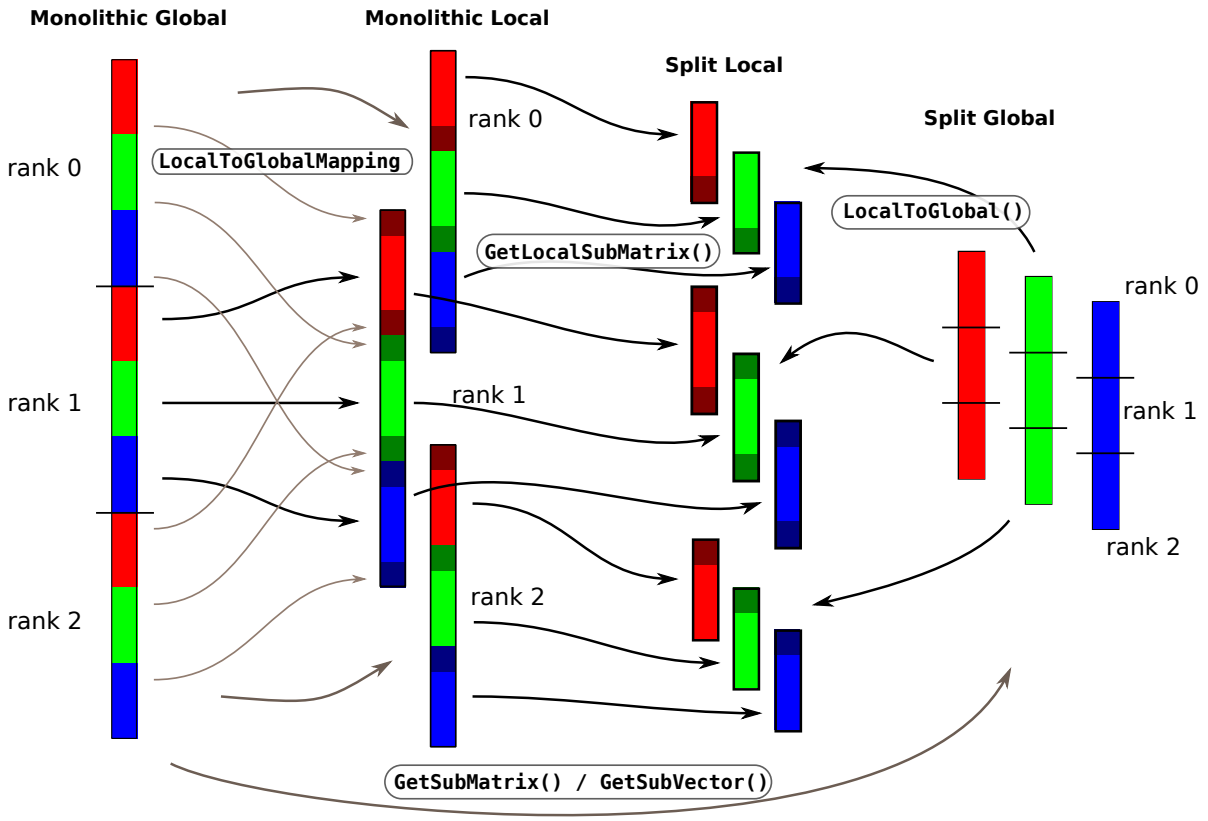

Figure 11: The relationship between spaces used for coupled assembly.

or implement MatMult(), but does implement MatSetValuesLocal() and, if `isrow,iscol` have a constant block size, MatSetValuesBlockedLocal(). Note that although `submat` may not be a fully functional matrix and the caller does not even know a priori which communicator it will reside on, it always implements the local assembly functions (which are not collective). The index sets `isrow,iscol` can be obtained using DMCompositeGetLocalISs() if DMComposite is being used. DMComposite can also be used to create matrices, in which case the MATNEST format can be specified using `-prefix_dm_mat_type nest` and MATAIJ can be specified using `-prefix_dm_mat_type aij`. See `$PETSC_DIR/src/snes/examples/tutorials/ex28.c` for a simple example using this interface.

3.2 Basic Matrix Operations

Table 2 summarizes basic PETSc matrix operations. We briefly discuss a few of these routines in more detail below.

The parallel matrix can multiply a vector with `n` local entries, returning a vector with `m` local entries. That is, to form the product

```
MatMult(Mat A,Vec x,Vec y);
```

the vectors `x` and `y` should be generated with

```
VecCreateMPI(MPI_Comm comm,n,N,&x);
VecCreateMPI(MPI_Comm comm,m,M,&y);
```

By default, if the user lets PETSc decide the number of components to be stored locally (by passing in PETSC_DECIDE as the second argument to VecCreateMPI() or using VecCreate()), vectors and matrices of the same dimension are automatically compatible for parallel matrix-vector operations.

Along with the matrix-vector multiplication routine, there is a version for the transpose of the matrix,

```
MatMultTranspose(Mat A,Vec x,Vec y);
```

There are also versions that add the result to another vector:

```
MatMultAdd(Mat A,Vec x,Vec y,Vec w);
MatMultTransposeAdd(Mat A,Vec x,Vec y,Vec w);
```

These routines, respectively, produce $w = A * x + y$ and $w = A^T * x + y$. In C it is legal for the vectors `y` and `w` to be identical. In Fortran, this situation is forbidden by the language standard, but we allow it anyway.

One can print a matrix (sequential or parallel) to the screen with the command

```
MatView(Mat mat,PETSC_VIEWER_STDOUT_WORLD);
```

Other viewers can be used as well. For instance, one can draw the nonzero structure of the matrix into the default X-window with the command

```
MatView(Mat mat,PETSC_VIEWER_DRAW_WORLD);
```

Also one can use

```
MatView(Mat mat,PetscViewer viewer);
```

where `viewer` was obtained with PetscViewerDrawOpen(). Additional viewers and options are given in the MatView() man page and Section 15.3.

The NormType argument to MatNorm() is one of NORM_1, NORM_INFINITY, and NORM_FROBENIUS.

Function Name	Operation
MatAXPY(Mat Y, PetscScalar a,Mat X,MatStructure);	$Y = Y + a * X$
MatMult(Mat A,Vec x, Vec y);	$y = A * x$
MatMultAdd(Mat A,Vec x, Vec y,Vec z);	$z = y + A * x$
MatMultTranspose(Mat A,Vec x, Vec y);	$y = A^T * x$
MatMultTransposeAdd(Mat A,Vec x, Vec y,Vec z);	$z = y + A^T * x$
MatNorm(Mat A,NormType type, double *r);	$r = \|\|A\|\|_{type}$
MatDiagonalScale(Mat A,Vec l,Vec r);	$A = \text{diag}(l) * A * \text{diag}(r)$
MatScale(Mat A,PetscScalar a);	$A = a * A$
MatConvert(Mat A,MatType type,Mat *B);	$B = A$
MatCopy(Mat A,Mat B,MatStructure);	$B = A$
MatGetDiagonal(Mat A,Vec x);	$x = \text{diag}(A)$
MatTranspose(Mat A,MatReuse,Mat* B);	$B = A^T$
MatZeroEntries(Mat A);	$A = 0$
MatShift(Mat Y,PetscScalar a);	$Y = Y + a * I$

Table 2: PETSc Matrix Operations

3.3 Matrix-Free Matrices

Some people like to use matrix-free methods, which do not require explicit storage of the matrix, for the numerical solution of partial differential equations. To support matrix-free methods in PETSc, one can use the following command to create a Mat structure without ever actually generating the matrix:

MatCreateShell(MPI_Comm comm,int m,int n,int M,int N,void *ctx,Mat *mat);

Here `M` and `N` are the global matrix dimensions (rows and columns), `m` and `n` are the local matrix dimensions, and `ctx` is a pointer to data needed by any user-defined shell matrix operations; the manual page has additional details about these parameters. Most matrix-free algorithms require only the application of the linear operator to a vector. To provide this action, the user must write a routine with the calling sequence

UserMult(Mat mat,Vec x,Vec y);

and then associate it with the matrix, `mat`, by using the command

MatShellSetOperation(Mat mat,MatOperation MATOP_MULT,
(void(*)(void)) PetscErrorCode (*UserMult)(Mat,Vec,Vec));

Here `MATOP_MULT` is the name of the operation for matrix-vector multiplication. Within each user-defined routine (such as `UserMult()`), the user should call MatShellGetContext() to obtain the user-defined context, `ctx`, that was set by MatCreateShell(). This shell matrix can be used with the iterative linear equation solvers discussed in the following chapters.

The routine MatShellSetOperation() can be used to set any other matrix operations as well. The file `${PETSC_DIR}/include/petscmat.h` provides a complete list of matrix operations, which have the form `MATOP_<OPERATION>`, where `<OPERATION>` is the name (in all capital letters) of the user interface routine (for example, MatMult() → `MATOP_MULT`). All user-provided functions have the same calling sequence as the usual matrix interface routines, since the user-defined functions are intended to be accessed through the same interface, e.g., MatMult(Mat,Vec,Vec) → `UserMult(Mat,Vec,Vec)`. The final argument for MatShellSetOperation() needs to be cast to a `void *`, since the final argument could (depending on the MatOperation) be a variety of different functions.

Note that MatShellSetOperation() can also be used as a "backdoor" means of introducing user-defined changes in matrix operations for other storage formats (for example, to override the default LU factorization routine supplied within PETSc for the MATSEQAIJ format). However, we urge anyone who introduces such changes to use caution, since it would be very easy to accidentally create a bug in the new routine that could affect other routines as well.

See also Section 5.5 for details on one set of helpful utilities for using the matrix-free approach for nonlinear solvers.

3.4 Other Matrix Operations

In many iterative calculations (for instance, in a nonlinear equations solver), it is important for efficiency purposes to reuse the nonzero structure of a matrix, rather than determining it anew every time the matrix is generated. To retain a given matrix but reinitialize its contents, one can employ

MatZeroEntries(Mat A);

This routine will zero the matrix entries in the data structure but keep all the data that indicates where the nonzeros are located. In this way a new matrix assembly will be much less expensive, since no memory allocations or copies will be needed. Of course, one can also explicitly set selected matrix elements to zero by calling MatSetValues().

By default, if new entries are made in locations where no nonzeros previously existed, space will be allocated for the new entries. To prevent the allocation of additional memory and simply discard those new entries, one can use the option

MatSetOption(Mat A,MAT_NEW_NONZERO_LOCATIONS,PETSC_FALSE);

Once the matrix has been assembled, one can factor it numerically without repeating the ordering or the symbolic factorization. This option can save some computational time, although it does require that the factorization is not done in-place.

In the numerical solution of elliptic partial differential equations, it can be cumbersome to deal with Dirichlet boundary conditions. In particular, one would like to assemble the matrix without regard to boundary conditions and then at the end apply the Dirichlet boundary conditions. In numerical analysis classes this process is usually presented as moving the known boundary conditions to the right-hand side and then solving a smaller linear system for the interior unknowns. Unfortunately, implementing this requires extracting a large submatrix from the original matrix and creating its corresponding data structures. This process can be expensive in terms of both time and memory.

One simple way to deal with this difficulty is to replace those rows in the matrix associated with known boundary conditions, by rows of the identity matrix (or some scaling of it). This action can be done with the command

MatZeroRows(Mat A,PetscInt numRows,PetscInt rows[],PetscScalar diag_value,Vec x,Vec b),

or equivalently,

MatZeroRowsIS(Mat A,IS rows,PetscScalar diag_value,Vec x,Vec b);

For sparse matrices this removes the data structures for certain rows of the matrix. If the pointer `diag_value` is NULL, it even removes the diagonal entry. If the pointer is not null, it uses that given value at the pointer location in the diagonal entry of the eliminated rows.

One nice feature of this approach is that when solving a nonlinear problem such that at each iteration the Dirichlet boundary conditions are in the same positions and the matrix retains the same nonzero structure, the user can call MatZeroRows() in the first iteration. Then, before generating the matrix in the second iteration the user should call

MatSetOption(Mat A,MAT_NEW_NONZERO_LOCATIONS,PETSC_FALSE);

From that point, no new values will be inserted into those (boundary) rows of the matrix.

The functions MatZeroRowsLocal() and MatZeroRowsLocalIS() can also be used if for each process one provides the Dirichlet locations in the local numbering of the matrix. A drawback of MatZeroRows() is that it destroys the symmetry of a matrix. Thus one can use

MatZeroRowsColumns(Mat A,PetscInt numRows,PetscInt rows[],PetscScalar diag_value,Vec x,Vec b),

or equivalently,

MatZeroRowsColumnsIS(Mat A,IS rows,PetscScalar diag_value,Vec x,Vec b);

Note that with all of these for a given assembled matrix it can be only called once to update the x and b vector. It cannot be used if one wishes to solve multiple right hand side problems for the same matrix since the matrix entries needed for updating the b vector are removed in its first use.

Once the zeroed rows are removed the new matrix has possibly many rows with only a diagonal entry affecting the parallel load balancing. The PCREDISTRIBUTE preconditioner removes all the zeroed rows (and associated columns and adjusts the right hand side based on the removed columns) and then rebalances the resulting rows of smaller matrix across the processes. Thus one can use MatZeroRows() to set the Dirichlet points and then solve with the preconditioner PCREDISTRIBUTE. Note if the original matrix was symmetric the smaller solved matrix will also be symmetric.

Another matrix routine of interest is

MatConvert(Mat mat,MatType newtype,Mat *M)

which converts the matrix `mat` to new matrix, `M`, that has either the same or different format. Set `newtype` to MATSAME to copy the matrix, keeping the same matrix format. See `${PETSC_DIR}/include/petscmat.h` for other available matrix types; standard ones are MATSEQDENSE, MATSEQAIJ, MATMPIAIJ, MATSEQBAIJ and MATMPIBAIJ.

In certain applications it may be necessary for application codes to directly access elements of a matrix. This may be done by using the the command (for local rows only)

MatGetRow(Mat A,int row, int *ncols,const PetscInt (*cols)[],const PetscScalar (*vals)[]);

The argument `ncols` returns the number of nonzeros in that row, while `cols` and `vals` returns the column indices (with indices starting at zero) and values in the row. If only the column indices are needed (and not the corresponding matrix elements), one can use NULL for the `vals` argument. Similarly, one can use NULL for the `cols` argument. The user can only examine the values extracted with MatGetRow(); the values *cannot* be altered. To change the matrix entries, one must use MatSetValues().

Once the user has finished using a row, he or she *must* call

MatRestoreRow(Mat A,int row,int *ncols,int **cols,PetscScalar **vals);

to free any space that was allocated during the call to MatGetRow().

3.5 Partitioning

For almost all unstructured grid computation, the distribution of portions of the grid across the process's work load and memory can have a very large impact on performance. In most PDE calculations the grid partitioning and distribution across the processes can (and should) be done in a "pre-processing" step before the numerical computations. However, this does not mean it need be done in a separate, sequential program,

rather it should be done before one sets up the parallel grid data structures in the actual program. PETSc provides an interface to the ParMETIS (developed by George Karypis; see the docs/installation/index.htm file for directions on installing PETSc to use ParMETIS) to allow the partitioning to be done in parallel. PETSc does not currently provide directly support for dynamic repartitioning, load balancing by migrating matrix entries between processes, etc. For problems that require mesh refinement, PETSc uses the "rebuild the data structure" approach, as opposed to the "maintain dynamic data structures that support the insertion/deletion of additional vector and matrix rows and columns entries" approach.

Partitioning in PETSc is organized around the MatPartitioning object. One first creates a parallel matrix that contains the connectivity information about the grid (or other graph-type object) that is to be partitioned. This is done with the command

```
MatCreateMPIAdj(MPI_Comm comm,int mlocal,int n,const int ia[],const int ja[],
                int *weights,Mat *Adj);
```

The argument `mlocal` indicates the number of rows of the graph being provided by the given process, `n` is the total number of columns; equal to the sum of all the `mlocal`. The arguments `ia` and `ja` are the row pointers and column pointers for the given rows, these are the usual format for parallel compressed sparse row storage, using indices starting at 0, **not** 1.

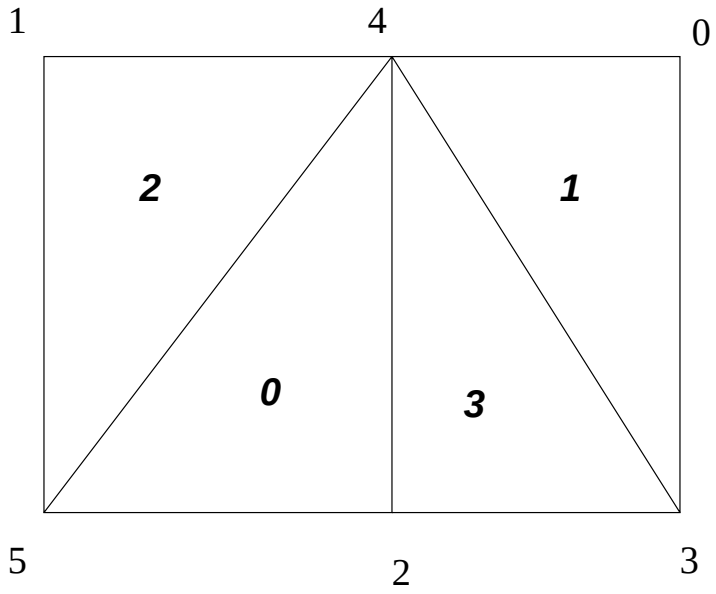

Figure 12: Numbering on Simple Unstructured Grid

This, of course, assumes that one has already distributed the grid (graph) information among the processes. The details of this initial distribution is not important; it could be simply determined by assigning to the first process the first n_0 nodes from a file, the second process the next n_1 nodes, etc.

For example, we demonstrate the form of the `ia` and `ja` for a triangular grid where we

(1) partition by element (triangle)

- Process 0, $mlocal = 2, n = 4, ja = \{2, 3, |3\}, ia = \{0, 2, 3\}$
- Process 1, $mlocal = 2, n = 4, ja = \{0, |0, 1\}, ia = \{0, 1, 3\}$

Note that elements are not connected to themselves and we only indicate edge connections (in some contexts single vertex connections between elements may also be included). We use a | above to denote the transition between rows in the matrix.

and (2) partition by vertex.

- Process 0, $mlocal = 3, n = 6, ja = \{3, 4, |4, 5, |3, 4, 5\}, ia = \{0, 2, 4, 7\}$
- Process 1, $mlocal = 3, n = 6, ja = \{0, 2, 4, |0, 1, 2, 3, 5, |1, 2, 4\}, ia = \{0, 3, 8, 11\}$.

Once the connectivity matrix has been created the following code will generate the renumbering required for the new partition

```
MatPartitioningCreate(MPI_Comm comm,MatPartitioning *part);
MatPartitioningSetAdjacency(MatPartitioning part,Mat Adj);
MatPartitioningSetFromOptions(MatPartitioning part);
MatPartitioningApply(MatPartitioning part,IS *is);
MatPartitioningDestroy(MatPartitioning *part);
MatDestroy(Mat *Adj);
ISPartitioningToNumbering(IS is,IS *isg);
```

The resulting `isg` contains for each local node the new global number of that node. The resulting `is` contains the new process number that each local node has been assigned to.

Now that a new numbering of the nodes has been determined, one must renumber all the nodes and migrate the grid information to the correct process. The command

```
AOCreateBasicIS(isg,NULL,&ao);
```

generates, see Section 2.3.1, an AO object that can be used in conjunction with the `is` and `isg` to move the relevant grid information to the correct process and renumber the nodes etc. In this context, the new ordering is the "application" ordering so AOPetscToApplication() converts old global indices to new global indices and AOApplicationToPetsc() converts new global indices back to old global indices.

PETSc does not currently provide tools that completely manage the migration and node renumbering, since it will be dependent on the particular data structure you use to store the grid information and the type of grid information that you need for your application. We do plan to include more support for this in the future, but designing the appropriate general user interface and providing a scalable implementation that can be used for a wide variety of different grids requires a great deal of time.

Chapter 4

KSP: Linear Equations Solvers

The object KSP is the heart of PETSc, because it provides uniform and efficient access to all of the package's linear system solvers, including parallel and sequential, direct and iterative. KSP is intended for solving nonsingular systems of the form

$$Ax = b, \tag{4.1}$$

where A denotes the matrix representation of a linear operator, b is the right-hand-side vector, and x is the solution vector. KSP uses the same calling sequence for both direct and iterative solution of a linear system. In addition, particular solution techniques and their associated options can be selected at runtime.

The combination of a Krylov subspace method and a preconditioner is at the center of most modern numerical codes for the iterative solution of linear systems. See, for example, [11] for an overview of the theory of such methods. KSP creates a simplified interface to the lower-level KSP and PC modules within the PETSc package. The KSP package, discussed in Section 4.3, provides many popular Krylov subspace iterative methods; the PC module, described in Section 4.4, includes a variety of preconditioners. Although both KSP and PC can be used directly, users should employ the interface of KSP.

4.1 Using KSP

To solve a linear system with KSP, one must first create a solver context with the command

KSPCreate(MPI_Comm comm,KSP *ksp);

Here `comm` is the MPI communicator, and `ksp` is the newly formed solver context. Before actually solving a linear system with KSP, the user must call the following routine to set the matrices associated with the linear system:

KSPSetOperators(KSP ksp,Mat Amat,Mat Pmat);

The argument `Amat`, representing the matrix that defines the linear system, is a symbolic place holder for any kind of matrix. In particular, KSP *does* support matrix-free methods. The routine MatCreateShell() in Section 3.3 provides further information regarding matrix-free methods. Typically the matrix from which the preconditioner is to be constructed, `Pmat`, is the same as the matrix that defines the linear system, `Amat`; however, occasionally these matrices differ (for instance, when a preconditioning matrix is obtained from a lower order method than that employed to form the linear system matrix).

Much of the power of KSP can be accessed through the single routine

KSPSetFromOptions(KSP ksp);

This routine accepts the options `-h` and `-help` as well as any of the KSP and PC options discussed below. To solve a linear system, one sets the rhs and solution vectors using and executes the command

KSPSolve(KSP ksp,Vec b,Vec x);

where `b` and `x` respectively denote the right-hand-side and solution vectors. On return, the iteration number at which the iterative process stopped can be obtained using

KSPGetIterationNumber(KSP ksp, int *its);

Note that this does not state that the method converged at this iteration: it can also have reached the maximum number of iterations, or have diverged. Section 4.3.2 gives more details regarding convergence testing. Note that multiple linear solves can be performed by the same KSP context. Once the KSP context is no longer needed, it should be destroyed with the command

KSPDestroy(KSP *ksp);

The above procedure is sufficient for general use of the KSP package. One additional step is required for users who wish to customize certain preconditioners (e.g., see Section 4.4.4) or to log certain performance data using the PETSc profiling facilities (as discussed in Chapter 13). In this case, the user can optionally explicitly call

KSPSetUp(KSP ksp)

before calling KSPSolve() to perform any setup required for the linear solvers. The explicit call of this routine enables the separate monitoring of any computations performed during the set up phase, such as incomplete factorization for the ILU preconditioner.

The default solver within KSP is restarted GMRES, preconditioned for the uniprocess case with ILU(0), and for the multiprocess case with the block Jacobi method (with one block per process, each of which is solved with ILU(0)). A variety of other solvers and options are also available. To allow application programmers to set any of the preconditioner or Krylov subspace options directly within the code, we provide routines that extract the PC and KSP contexts,

KSPGetPC(KSP ksp,PC *pc);

The application programmer can then directly call any of the PC or KSP routines to modify the corresponding default options.

To solve a linear system with a direct solver (currently supported by PETSc for sequential matrices, and by several external solvers through PETSc interfaces (see Section 4.7)) one may use the options `-ksp_type preonly -pc_type lu` (see below).

By default, if a direct solver is used, the factorization is *not* done in-place. This approach prevents the user from the unexpected surprise of having a corrupted matrix after a linear solve. The routine PCFactorSetUseInPlace(), discussed below, causes factorization to be done in-place.

4.2 Solving Successive Linear Systems

When solving multiple linear systems of the same size with the same method, several options are available. To solve successive linear systems having the *same* preconditioner matrix (i.e., the same data structure with exactly the same matrix elements) but different right-hand-side vectors, the user should simply call KSPSolve,() multiple times. The preconditioner setup operations (e.g., factorization for ILU) will be done during the first call to KSPSolve() only; such operations will *not* be repeated for successive solves.

To solve successive linear systems that have *different* preconditioner matrices (i.e., the matrix elements and/or the matrix data structure change), the user *must* call KSPSetOperators() and KSPSolve() for each solve. See Section 4.1 for a description of various flags for KSPSetOperators() that can save work for such cases.

4.3 Krylov Methods

The Krylov subspace methods accept a number of options, many of which are discussed below. First, to set the Krylov subspace method that is to be used, one calls the command

KSPSetType(KSP ksp,KSPType method);

The type can be one of KSPRICHARDSON, KSPCHEBYSHEV, KSPCG, KSPGMRES, KSPTCQMR, KSPBCGS, KSPCGS, KSPTFQMR, KSPCR, KSPLSQR, KSPBICG, or KSPPREONLY. The KSP method can also be set with the options database command `-ksp_type`, followed by one of the options `richardson`, `chebyshev`, `cg`, `gmres`, `tcqmr`, `bcgs`, `cgs`, `tfqmr`, `cr`, `lsqr`, `bicg`, or `preonly`. There are method-specific options for the Richardson, Chebyshev, and GMRES methods:

KSPRichardsonSetScale(KSP ksp,double damping_factor);
KSPChebyshevSetEigenvalues(KSP ksp,double emax,double emin);
KSPGMRESSetRestart(KSP ksp,int max_steps);

The default parameter values are `damping_factor=1.0, emax=0.01, emin=100.0`, and `max_steps=30`. The GMRES restart and Richardson damping factor can also be set with the options `-ksp_gmres_restart <n>` and `-ksp_richardson_scale <factor>`.

The default technique for orthogonalization of the Hessenberg matrix in GMRES is the unmodified (classical) Gram-Schmidt method, which can be set with

KSPGMRESSetOrthogonalization(KSP ksp,KSPGMRESClassicalGramSchmidtOrthogonalization);

or the options database command `-ksp_gmres_classicalgramschmidt`. By default this will **not** use iterative refinement to improve the stability of the orthogonalization. This can be changed with the option

KSPGMRESSetCGSRefinementType(KSP ksp,KSPGMRESCGSRefinementType type)

or via the options database with

-ksp_gmres_cgs_refinement_type none,ifneeded,always

The values for KSPGMRESCGSRefinementType are `KSP_GMRES_CGS_REFINEMENT_NONE`, `KSP_GMRES_CGS_REFINEMENT_IFNEEDED` and `KSP_GMRES_CGS_REFINEMENT_ALWAYS`.

One can also use modified Gram-Schmidt, by setting the orthogonalization routine, KSPGMRESModifiedGramSchmidtOrthogonalization(), by using the command line option `-ksp_gmres_modifiedgramschmidt`.

For the conjugate gradient method with complex numbers, there are two slightly different algorithms depending on whether the matrix is Hermitian symmetric or truly symmetric (the default is to assume that it is Hermitian symmetric). To indicate that it is symmetric, one uses the command

KSPCGSetType(KSP ksp,KSPCGType `KSP_CG_SYMMETRIC`);

Note that this option is not valid for all matrices.

The LSQR algorithm does not involve a preconditioner, any preconditioner set to work with the KSP object is ignored if LSQR was selected.

By default, KSP assumes an initial guess of zero by zeroing the initial value for the solution vector that is given; this zeroing is done at the call to KSPSolve() (or KSPSolve()). To use a nonzero initial guess, the user *must* call

KSPSetInitialGuessNonzero(KSP ksp,PetscBool flg);

4.3.1 Preconditioning within KSP

Since the rate of convergence of Krylov projection methods for a particular linear system is strongly dependent on its spectrum, preconditioning is typically used to alter the spectrum and hence accelerate the convergence rate of iterative techniques. Preconditioning can be applied to the system (4.1) by

$$\left(M_L^{-1} A M_R^{-1}\right)\left(M_R x\right) = M_L^{-1} b, \tag{4.2}$$

where M_L and M_R indicate preconditioning matrices (or, matrices from which the preconditioner is to be constructed). If $M_L = I$ in (4.2), right preconditioning results, and the residual of (4.1),

$$r \equiv b - Ax = b - A M_R^{-1} M_R x,$$

is preserved. In contrast, the residual is altered for left ($M_R = I$) and symmetric preconditioning, as given by

$$r_L \equiv M_L^{-1} b - M_L^{-1} A x = M_L^{-1} r.$$

By default, all KSP implementations use left preconditioning. Right preconditioning can be activated for some methods by using the options database command `-ksp_pc_side right` or calling the routine

KSPSetPCSide(KSP ksp,PCSide PC_RIGHT);

Attempting to use right preconditioning for a method that does not currently support it results in an error message of the form

KSPSetUp_Richardson:No right preconditioning for KSPRICHARDSON

We summarize the defaults for the residuals used in KSP convergence monitoring within Table 3. Details regarding specific convergence tests and monitoring routines are presented in the following sections. The preconditioned residual is used by default for convergence testing of all left-preconditioned KSP methods. For the conjugate gradient, Richardson, and Chebyshev methods the true residual can be used by the options database command `ksp_norm_type unpreconditioned` or by calling the routine

KSPSetNormType(KSP ksp,KSP_NORM_UNPRECONDITIONED);

Note: the bi-conjugate gradient method requires application of both the matrix and its transpose plus the preconditioner and its transpose. Currently not all matrices and preconditioners provide this support and thus the KSPBICG cannot always be used.

4.3.2 Convergence Tests

The default convergence test, KSPConvergedDefault(), is based on the l_2-norm of the residual. Convergence (or divergence) is decided by three quantities: the decrease of the residual norm relative to the norm of the right hand side, `rtol`, the absolute size of the residual norm, `atol`, and the relative increase in the residual, `dtol`. Convergence is detected at iteration k if

$$\|r_k\|_2 < \max(rtol * \|b\|_2, atol),$$

where $r_k = b - Ax_k$. Divergence is detected if

$$\|r_k\|_2 > dtol * \|b\|_2.$$

These parameters, as well as the maximum number of allowable iterations, can be set with the routine

KSPSetTolerances(KSP ksp,double rtol,double atol,double dtol,int maxits);

Method	KSPType	Options Database Name
Richardson	KSPRICHARDSON	richardson
Chebyshev	KSPCHEBYSHEV	chebyshev
Conjugate Gradient [17]	KSPCG	cg
BiConjugate Gradient	KSPBICG	bicg
Generalized Minimal Residual [26]	KSPGMRES	gmres
Flexible Generalized Minimal Residual	KSPFGMRES	fgmres
Deflated Generalized Minimal Residual	KSPDGMRES	dgmres
Generalized Conjugate Residual	KSPGCR	gcr
BiCGSTAB [30]	KSPBCGS	bcgs
Conjugate Gradient Squared [29]	KSPCGS	cgs
Transpose-Free Quasi-Minimal Residual (1) [12]	KSPTFQMR	tfqmr
Transpose-Free Quasi-Minimal Residual (2)	KSPTCQMR	tcqmr
Conjugate Residual	KSPCR	cr
Least Squares Method	KSPLSQR	lsqr
Shell for no KSP method	KSPPREONLY	preonly

Table 3: KSP Objects.

The user can retain the default value of any of these parameters by specifying PETSC_DEFAULT as the corresponding tolerance; the defaults are `rtol`=10^{-5}, `atol`=10^{-50}, `dtol`=10^{5}, and `maxits`=10^{4}. These parameters can also be set from the options database with the commands `-ksp_rtol <rtol>`, `-ksp_atol <atol>`, `-ksp_divtol <dtol>`, and `-ksp_max_it <its>`.

In addition to providing an interface to a simple convergence test, KSP allows the application programmer the flexibility to provide customized convergence-testing routines. The user can specify a customized routine with the command

KSPSetConvergenceTest(KSP ksp,PetscErrorCode (*test)(KSP ksp,int it,double rnorm,
KSPConvergedReason *reason,void *ctx),void *ctx,PetscErrorCode (*destroy)(void *ctx));

The final routine argument, `ctx`, is an optional context for private data for the user-defined convergence routine, `test`. Other `test` routine arguments are the iteration number, `it`, and the residual's l_2 norm, `rnorm`. The routine for detecting convergence, `test`, should set reason to positive for convergence, 0 for no convergence, and negative for failure to converge. A list of possible KSPConvergedReason is given in `include/petscksp.h`. You can use KSPGetConvergedReason() after KSPSolve() to see why convergence/divergence was detected.

4.3.3 Convergence Monitoring

By default, the Krylov solvers run silently without displaying information about the iterations. The user can indicate that the norms of the residuals should be displayed by using `-ksp_monitor` within the options database. To display the residual norms in a graphical window (running under X Windows), one should use `-ksp_monitor_lg_residualnorm [x,y,w,h]`, where either all or none of the options must be specified. Application programmers can also provide their own routines to perform the monitoring by using the command

KSPMonitorSet(KSP ksp,PetscErrorCode (*mon)(KSP ksp,int it,double rnorm,void *ctx),

void *ctx,PetscErrorCode (*mondestroy)(void**));

The final routine argument, `ctx`, is an optional context for private data for the user-defined monitoring routine, `mon`. Other `mon` routine arguments are the iteration number (`it`) and the residual's l_2 norm (`rnorm`). A helpful routine within user-defined monitors is `PetscObjectGetComm((PetscObject)ksp,MPI_Comm *comm)`, which returns in `comm` the MPI communicator for the KSP context. See section 1.3 for more discussion of the use of MPI communicators within PETSc.

Several monitoring routines are supplied with PETSc, including

KSPMonitorDefault(KSP,int,double, void *);
KSPMonitorSingularValue(KSP,int,double, void *);
KSPMonitorTrueResidualNorm(KSP,int,double, void *);

The default monitor simply prints an estimate of the l_2-norm of the residual at each iteration. The routine KSPMonitorSingularValue() is appropriate only for use with the conjugate gradient method or GMRES, since it prints estimates of the extreme singular values of the preconditioned operator at each iteration. Since KSPMonitorTrueResidualNorm() prints the true residual at each iteration by actually computing the residual using the formula $r = b - Ax$, the routine is slow and should be used only for testing or convergence studies, not for timing. These monitors may be accessed with the command line options `-ksp_monitor`, `-ksp_monitor_singular_value`, and `-ksp_monitor_true_residual`.

To employ the default graphical monitor, one should use the commands

PetscDrawLG lg;
KSPMonitorLGResidualNormCreate(MPI_Comm comm,char *display,char *title,int x,int y,int w,int h,PetscDrawLG *lg);
KSPMonitorSet(KSP ksp,KSPMonitorLGResidualNorm,lg,0);

When no longer needed, the line graph should be destroyed with the command

PetscDrawLGDestroy(PetscDrawLG *lg);

The user can change aspects of the graphs with the `PetscDrawLG*()` and `PetscDrawAxis*()` routines. One can also access this functionality from the options database with the command `-ksp_monitor_lg_residualnorm [x,y,w,h].`, where `x, y, w, h` are the optional location and size of the window.

One can cancel hardwired monitoring routines for KSP at runtime with `-ksp_monitor_cancel`.

Unless the Krylov method converges so that the residual norm is small, say 10^{-10}, many of the final digits printed with the `-ksp_monitor` option are meaningless. Worse, they are different on different machines; due to different round-off rules used by, say, the IBM RS6000 and the Sun SPARC. This makes testing between different machines difficult. The option `-ksp_monitor_short` causes PETSc to print fewer of the digits of the residual norm as it gets smaller; thus on most of the machines it will always print the same numbers making cross system testing easier.

4.3.4 Understanding the Operator's Spectrum

Since the convergence of Krylov subspace methods depends strongly on the spectrum (eigenvalues) of the preconditioned operator, PETSc has specific routines for eigenvalue approximation via the Arnoldi or Lanczos iteration. First, before the linear solve one must call

KSPSetComputeEigenvalues(KSP ksp,PETSC_TRUE);

Then after the KSP solve one calls

KSPComputeEigenvalues(KSP ksp, int n,double *realpart,double *complexpart,int *neig);

Here, n is the size of the two arrays and the eigenvalues are inserted into those two arrays. Neig is the number of eigenvalues computed; this number depends on the size of the Krylov space generated during the linear system solution, for GMRES it is never larger than the restart parameter. There is an additional routine

KSPComputeEigenvaluesExplicitly(KSP ksp, int n,double *realpart,double *complexpart);

that is useful only for very small problems. It explicitly computes the full representation of the preconditioned operator and calls LAPACK to compute its eigenvalues. It should be only used for matrices of size up to a couple hundred. The PetscDrawSP*() routines are very useful for drawing scatter plots of the eigenvalues.

The eigenvalues may also be computed and displayed graphically with the options data base commands -ksp_plot_eigenvalues and -ksp_plot_eigenvalues_explicitly. Or they can be dumped to the screen in ASCII text via -ksp_compute_eigenvalues and -ksp_compute_eigenvalues_explicitly.

4.3.5 Other KSP Options

To obtain the solution vector and right hand side from a KSP context, one uses

KSPGetSolution(KSP ksp,Vec *x);
KSPGetRhs(KSP ksp,Vec *rhs);

During the iterative process the solution may not yet have been calculated or it may be stored in a different location. To access the approximate solution during the iterative process, one uses the command

KSPBuildSolution(KSP ksp,Vec w,Vec *v);

where the solution is returned in v. The user can optionally provide a vector in w as the location to store the vector; however, if w is NULL, space allocated by PETSc in the KSP context is used. One should not destroy this vector. For certain KSP methods, (e.g., GMRES), the construction of the solution is expensive, while for many others it requires not even a vector copy.

Access to the residual is done in a similar way with the command

KSPBuildResidual(KSP ksp,Vec t,Vec w,Vec *v);

Again, for GMRES and certain other methods this is an expensive operation.

4.4 Preconditioners

As discussed in Section 4.3.1, the Krylov space methods are typically used in conjunction with a preconditioner. To employ a particular preconditioning method, the user can either select it from the options database using input of the form -pc_type <methodname> or set the method with the command

PCSetType(PC pc,PCType method);

In Table 4 we summarize the basic preconditioning methods supported in PETSc. The PCSHELL preconditioner uses a specific, application-provided preconditioner. The direct preconditioner, PCLU, is, in fact, a direct solver for the linear system that uses LU factorization. PCLU is included as a preconditioner so that PETSc has a consistent interface among direct and iterative linear solvers.

Each preconditioner may have associated with it a set of options, which can be set with routines and options database commands provided for this purpose. Such routine names and commands are all of the form PC<TYPE>Option and -pc_<type>_option [value]. A complete list can be found by consulting the manual pages; we discuss just a few in the sections below.

Method	PCType	Options Database Name
Jacobi	PCJACOBI	jacobi
Block Jacobi	PCBJACOBI	bjacobi
SOR (and SSOR)	PCSOR	sor
SOR with Eisenstat trick	PCEISENSTAT	eisenstat
Incomplete Cholesky	PCICC	icc
Incomplete LU	PCILU	ilu
Additive Schwarz	PCASM	asm
Generalized Additive Schwarz	PCGASM	gasm
Algebraic Multigrid	PCGAMG	gamg
Balancing Domain Decomposition by Constraints	PCBDDC	bddc
Linear solver	PCKSP	ksp
Combination of preconditioners	PCCOMPOSITE	composite
LU	PCLU	lu
Cholesky	PCCHOLESKY	cholesky
No preconditioning	PCNONE	none
Shell for user-defined PC	PCSHELL	shell

Table 4: PETSc Preconditioners

4.4.1 ILU and ICC Preconditioners

Some of the options for ILU preconditioner are

PCFactorSetLevels(PC pc,int levels);
PCFactorSetReuseOrdering(PC pc,PetscBool flag);
PCFactorSetDropTolerance(PC pc,double dt,double dtcol,int dtcount);
PCFactorSetReuseFill(PC pc,PetscBool flag);
PCFactorSetUseInPlace(PC pc,PetscBool flg);
PCFactorSetAllowDiagonalFill(PC pc,PetscBool flg);

When repeatedly solving linear systems with the same KSP context, one can reuse some information computed during the first linear solve. In particular, PCFactorSetReuseOrdering() causes the ordering (for example, set with `-pc_factor_mat_ordering_type order`) computed in the first factorization to be reused for later factorizations. PCFactorSetUseInPlace() is often used with PCASM or PCBJACOBI when zero fill is used, since it reuses the matrix space to store the incomplete factorization it saves memory and copying time. Note that in-place factorization is not appropriate with any ordering besides natural and cannot be used with the drop tolerance factorization. These options may be set in the database with

```
-pc_factor_levels <levels>
-pc_factor_reuse_ordering
-pc_factor_reuse_fill
-pc_factor_in_place
-pc_factor_nonzeros_along_diagonal
-pc_factor_diagonal_fill
```

See Section 14.4.2 for information on preallocation of memory for anticipated fill during factorization. By alleviating the considerable overhead for dynamic memory allocation, such tuning can significantly enhance performance.

PETSc supports incomplete factorization preconditioners for several matrix types for sequential matrix (for example MATSEQAIJ, MATSEQBAIJ, MATSEQSBAIJ).

4.4.2 SOR and SSOR Preconditioners

PETSc provides only a sequential SOR preconditioner that can only be used on sequential matrices or as the subblock preconditioner when using block Jacobi or ASM preconditioning (see below).

The options for SOR preconditioning are

PCSORSetOmega(PC pc,double omega);
PCSORSetIterations(PC pc,int its,int lits);
PCSORSetSymmetric(PC pc,MatSORType type);

The first of these commands sets the relaxation factor for successive over (under) relaxation. The second command sets the number of inner iterations `its` and local iterations `lits` (the number of smoothing sweeps on a process before doing a ghost point update from the other processes) to use between steps of the Krylov space method. The total number of SOR sweeps is given by `its*lits`. The third command sets the kind of SOR sweep, where the argument `type` can be one of `SOR_FORWARD_SWEEP`, `SOR_BACKWARD_SWEEP` or `SOR_SYMMETRIC_SWEEP`, the default being `SOR_FORWARD_SWEEP`. Setting the type to be `SOR_SYMMETRIC_SWEEP` produces the SSOR method. In addition, each process can locally and independently perform the specified variant of SOR with the types `SOR_LOCAL_FORWARD_SWEEP`, `SOR_LOCAL_BACKWARD_SWEEP`, and `SOR_LOCAL_SYMMETRIC_SWEEP`. These variants can also be set with the options `-pc_sor_omega <omega>`, `-pc_sor_its <its>`, `-pc_sor_lits <lits>`, `-pc_sor_backward`, `-pc_sor_symmetric`, `-pc_sor_local_forward`, `-pc_sor_local_backward`, and `-pc_sor_local_symmetric`.

The Eisenstat trick [9] for SSOR preconditioning can be employed with the method PCEISENSTAT (`-pc_type eisenstat`). By using both left and right preconditioning of the linear system, this variant of SSOR requires about half of the floating-point operations for conventional SSOR. The option `-pc_eisenstat_no_diagonal_scaling`) (or the routine PCEisenstatSetNoDiagonalScaling()) turns off diagonal scaling in conjunction with Eisenstat SSOR method, while the option `-pc_eisenstat_omega <omega>` (or the routine PCEisenstatSetOmega(PC pc,double omega)) sets the SSOR relaxation coefficient, `omega`, as discussed above.

4.4.3 LU Factorization

The LU preconditioner provides several options. The first, given by the command

PCFactorSetUseInPlace(PC pc,PetscBool flg);

causes the factorization to be performed in-place and hence destroys the original matrix. The options database variant of this command is `-pc_factor_in_place`. Another direct preconditioner option is selecting the ordering of equations with the command

`-pc_factor_mat_ordering_type <ordering>`

The possible orderings are

- `MATORDERINGNATURAL` - Natural
- `MATORDERINGND` - Nested Dissection
- `MATORDERING1WD` - One-way Dissection
- `MATORDERINGRCM` - Reverse Cuthill-McKee
- `MATORDERINGQMD` - Quotient Minimum Degree

These orderings can also be set through the options database by specifying one of the following: `-pc_factor_mat_ordering_type natural`, or `nd`, or `1wd`, or `rcm`, or `qmd`. In addition, see MatGetOrdering(), discussed in Section 17.2.

The sparse LU factorization provided in PETSc does not perform pivoting for numerical stability (since they are designed to preserve nonzero structure), thus occasionally a LU factorization will fail with a zero pivot when, in fact, the matrix is non-singular. The option `-pc_factor_nonzeros_along_diagonal <tol>` will often help eliminate the zero pivot, by preprocessing the the column ordering to remove small values from the diagonal. Here, `tol` is an optional tolerance to decide if a value is nonzero; by default it is $1.e-10$.

In addition, Section 14.4.2 provides information on preallocation of memory for anticipated fill during factorization. Such tuning can significantly enhance performance, since it eliminates the considerable overhead for dynamic memory allocation.

4.4.4 Block Jacobi and Overlapping Additive Schwarz Preconditioners

The block Jacobi and overlapping additive Schwarz methods in PETSc are supported in parallel; however, only the uniprocess version of the block Gauss-Seidel method is currently in place. By default, the PETSc implementations of these methods employ ILU(0) factorization on each individual block (that is, the default solver on each subblock is PCType=PCILU, KSPType=KSPPREONLY); the user can set alternative linear solvers via the options `-sub_ksp_type` and `-sub_pc_type`. In fact, all of the KSP and PC options can be applied to the subproblems by inserting the prefix `-sub_` at the beginning of the option name. These options database commands set the particular options for *all* of the blocks within the global problem. In addition, the routines

PCBJacobiGetSubKSP(PC pc,int *n_local,int *first_local,KSP **subksp);
PCASMGetSubKSP(PC pc,int *n_local,int *first_local,KSP **subksp);

extract the KSP context for each local block. The argument `n_local` is the number of blocks on the calling process, and `first_local` indicates the global number of the first block on the process. The blocks are numbered successively by processes from zero through $gb-1$, where gb is the number of global blocks. The array of KSP contexts for the local blocks is given by `subksp`. This mechanism enables the user to set different solvers for the various blocks. To set the appropriate data structures, the user *must* explicitly call KSPSetUp() before calling PCBJacobiGetSubKSP() or PCASMGetSubKSP(). For further details, see the examples `${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex7.c` or `${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex8.c`.

The block Jacobi, block Gauss-Seidel, and additive Schwarz preconditioners allow the user to set the number of blocks into which the problem is divided. The options database commands to set this value are `-pc_bjacobi_blocks n` and `-pc_bgs_blocks n`, and, within a program, the corresponding routines are

PCBJacobiSetTotalBlocks(PC pc,int blocks,int *size);
PCASMSetTotalSubdomains(PC pc,int n,IS *is,IS *islocal);
PCASMSetType(PC pc,PCASMType type);

The optional argument `size`, is an array indicating the size of each block. Currently, for certain parallel matrix formats, only a single block per process is supported. However, the MATMPIAIJ and MATMPIBAIJ formats support the use of general blocks as long as no blocks are shared among processes. The `is` argument contains the index sets that define the subdomains.

The object PCASMType is one of `PC_ASM_BASIC`, `PC_ASM_INTERPOLATE`, `PC_ASM_RESTRICT`, `PC_ASM_NONE` and may also be set with the options database `-pc_asm_type [basic, interpolate,`

restrict, none]. The type PC_ASM_BASIC (or -pc_asm_type basic) corresponds to the standard additive Schwarz method that uses the full restriction and interpolation operators. The type PC_ASM_RESTRICT (or -pc_asm_type restrict) uses a full restriction operator, but during the interpolation process ignores the off-process values. Similarly, PC_ASM_INTERPOLATE (or -pc_asm_type interpolate) uses a limited restriction process in conjunction with a full interpolation, while PC_ASM_NONE (or -pc_asm_type none) ignores off-process values for both restriction and interpolation. The ASM types with limited restriction or interpolation were suggested by Xiao-Chuan Cai and Marcus Sarkis [6]. PC_ASM_RESTRICT is the PETSc default, as it saves substantial communication and for many problems has the added benefit of requiring fewer iterations for convergence than the standard additive Schwarz method.

The user can also set the number of blocks and sizes on a per-process basis with the commands

PCBJacobiSetLocalBlocks(PC pc,int blocks,int *size);
PCASMSetLocalSubdomains(PC pc,int N,IS *is,IS *islocal);

For the ASM preconditioner one can use the following command to set the overlap to compute in constructing the subdomains.

PCASMSetOverlap(PC pc,int overlap);

The overlap defaults to 1, so if one desires that no additional overlap be computed beyond what may have been set with a call to PCASMSetTotalSubdomains() or PCASMSetLocalSubdomains(), then overlap must be set to be 0. In particular, if one does *not* explicitly set the subdomains in an application code, then all overlap would be computed internally by PETSc, and using an overlap of 0 would result in an ASM variant that is equivalent to the block Jacobi preconditioner. Note that one can define initial index sets is with *any* overlap via PCASMSetTotalSubdomains() or PCASMSetLocalSubdomains(); the routine PCASMSetOverlap() merely allows PETSc to extend that overlap further if desired.

PCGASM is an experimental generalization of PCASM that allows the user to specify subdomains that span multiple MPI ranks. This can be useful for problems where small subdomains result in poor convergence. To be effective, the multirank subproblems must be solved using a sufficient strong subsolver, such as LU, for which SuperLU_DIST or a similar parallel direct solver could be used; other choices may include a multigrid solver on the subdomains.

The interface for PCGASM is similar to that of PCASM. In particular, PCGASMType is one of PC_GASM_BASIC, PC_GASM_INTERPOLATE, PC_GASM_RESTRICT, PC_GASM_NONE. These options have the same meaning as with PCASM and may also be set with the options database -pc_gasm_type [basic, interpolate, restrict, none].

Unlike PCASM, however, PCGASM allows the user to define subdomains that span multiple MPI ranks. The simplest way to do this is using a call to PCGASMSetTotalSubdomains(PC pc,PetscInt N) with the total number of subdomains N that is smaller than the MPI communicator size. In this case PCGASM will coalesce size/N concecutive single-rank subdomains into a single multi-rank subdomain. The single-rank subdomains contain the degrees of freedom corresponding to the locally-owned rows of the PCGASM preconditioning matrix – these are the subdomains PCASM and PCGASM use by default.

Each of the multirank subdomain subproblems is defined on the subcommunicator that contains the coalesced PCGASM ranks. In general this might not result in a very good subproblem if the single-rank problems corresponding to the coalesced ranks are not very strongly connected. In the future this will be addressed with a hierarchical partitioner that generates well-connected coarse subdomains first before subpartitioning them into the single-rank subdomains.

In the meantime the user can provide his or her own multi-rank subdomains by calling PCGASMSetSubdomains(PC,IS[],IS[]) where each of the IS objects on the list defines the inner (without the overlap) or the outer (including the overlap) subdomain on the subcommunicator of the IS object. A helper

subroutine `PCGASMCreateSubdomains2D()` is similar to PCASM's but is capable of constructing multi-rank subdomains that can be then used with `PCGASMSetSubdomains()`. An alternative way of creating multi-rank subdomains is by using the underlying DM object, if it is capable of generating such decompositions via `DMCreateDomainDecomposition()`. Ordinarily the decomposition specified by the user via `PCGASMSetSubdomains()` takes precedence, unless `PCGASMSetUseDMSubdomains()` instructs PCGASM to prefer DM-created decompositions.

Currently there is no support for increasing the overlap of multi-rank subdomains via `PCGASMSetOverlap()` – this functionality works only for subdomains that fit within a single MPI rank, exactly as in PCASM.

Examples of the described PCGASM usage can be found in `${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex62.c`. In particular, `runex62_superlu_dist` illustrates the use of `SuperLU_DIST` as the subdomain solver on coalesced multi-rank subdomains. The `runex62_2D_*` examples illustrate the use of `PCGASMCreateSubdomains2D()`.

4.4.5 Algebraic Multigrid (AMG) Preconditioners

PETSc provides a native algebraic multigrid preconditioner PCGAMG – *"gamg"* – and interfaces to two external AMG packages: *hypre* and *ML*.

PETSc provides a (smoothed) aggregation AMG, (`-pc_type gamg -pc_gamg_type agg` or PCSetType(pc,PCGAMG) and PCGAMGSetType(pc,PCGAMGAgg). There are also less developed implementations of a classical AMG method (`-pc_gamg_type classical`) and a hybrid geometric AMG method (`-pc_gamg_type geo`).

GAMG provides unsmoothed aggregation (`-pc_gamg_agg_nsmooths 0`) and smoothed aggregation (`-pc_gamg_agg_nsmooths 1` or PCGAMGSetNSmooths(pc,1)). Smoothed aggregation is recommended for symmetric positive definite systems; unsmoothed aggregation can be useful for asymmetric problems and problems where highest eigen estimates are problematic. Try `-pc_gamg_agg_nsmooths 0` if poor convergence rates are observed using the smoothed version. AMG methods requires knowledge of the number of degrees of freedom per vertex, the default is one (a scalar problem) but vector problems like elasticity should set the block size of the matrix appropriately with MatSetBlockSize(mat,bs). Smoothed aggregation relies an explicit representation of the (near) null space of the operator. One can provide this with MatSetNearNullSpace(). For elasticity, where rotational rigid body modes define the near null space you can call PCSetCoordinates() instead of MatSetNearNullSpace() and it will set the near null space for you.

Other options for GAMG include PCGAMGSetProcEqLim(), PCGAMGSetCoarseEqLim(), PCGAMGSetRepartitioning(), PCGAMGRegister(), PCGAMGSetReuseInterpolation(), PCGAMGSetUseASMAggs(), PCGAMGSetNlevels(), PCGAMGSetThreshold(), and PCGAMGGetType().

Trouble shooting algebraic multigrid methods: If GAMG, ML, or hypre does not perform well the first thing to try is the other methods. You can also try `-mg_levels_ksp_type richardson -mg_levels_pc_type sor`. AMG methods are sensitive to coarsening rates and methods; for GAMG use `-pc_gamg_threshold` x to regulate coarsening rates (PCGAMGSetThreshold()). A high threshold (eg, $x = 0.08$) will result in an expensive but potentially powerful preconditioner; a low threshold (eg, $x = 0.0$) will result in faster coarsening, fewer levels, cheaper solves, and generally worse convergence rates. `-pc_gamg_threshold 0.0` is the most robust option (the reason for this is not obvious) and is recommended if poor convergence rates are observed, at least until the source of the problem is discovered. Optimizing parameters for AMG is tricky; we can help. To get help with your problem, first tell us the PDE that you are solving and any other useful information like discretizations, stretched grids, high contrast media, etc. Run with `-info` and pipe the output, which will be large, to `grep GAMG`, and send us the output (about 20 lines). This will give us useful high level information that will often give us a direction to start optimizing parameters.

Finally, there are several sources of poor performance of AMG solvers and often special purpose methods must be developed to achieve the full potential of multigrid. To name just a few sources of performance degradation that may not be fixed with parameters or bug fixes in PETSc currently: non-elliptic operators, curl/curl operators, highly stretched grids or highly anisotropic problems, large jumps in material coefficients with complex geometry (AMG is particularly well suited to jumps in coefficients but it is not a perfect solution), highly incompressible elasticity, not to mention ill-posed problems, and many others. For div/div and curl/curl operators, you may want to try the Auxilary Maxwell Space (AMS, `-pc_type hypre -pc_hypre_set_type ams`) or the Auxilary Divergence Space (ADS, `-pc_type hypre -pc_hypre_set_type ads`) solvers. These solvers need some additional information on the underlying mesh; specifically, AMS needs the so-called discrete gradient operator, which can be specified via PCHYPRESetDiscreteGradient(). In addition to the the discrete gradient, ADS also needs the specification of the discrete curl operator, which can be set using PCHYPRESetDiscreteCurl().

PETSc's AMG solver is constructed as a framework for developers to easily add AMG capabilities, like a new AMG algorithm or a new matrix triple product, etc. Contact us directly if you are interested in extending/improving PETSc's algebraic multigrid solvers.

4.4.6 Balancing Domain Decomposition by Constraints

PETSc provides the Balancing Domain Decomposition by Constraints (BDDC) method for preconditioning parallel finite element problems stored in unassembled format (see MATIS). BDDC is a 2-level non-overlapping domain decomposition method which can be easily adapted to different problems and discretizations by means of few user customizations. The application of the preconditioner to a vector consists in the static condensation of the residual at the interior of the subdomains by means of local Dirichet solves, followed by an additive combination of Neumann local corrections and the solution of a global coupled coarse problem. Command line options for the underlying KSP objects are prefixed by `-pc_bddc_dirichlet`, `-pc_bddc_neumann`, and `-pc_bddc_coarse` respectively.

The current implementation supports any kind of linear system, and assumes a one-to-one mapping between subdomains and MPI processes. Complex numbers are supported as well. For non-symmetric problems, use the runtime option `-pc_bddc_symmetric 0`.

Unlike conventional non-overlapping methods that iterates just on the degrees of freedom at the interface between subdomain, PCBDDC iterates on the whole set of degrees of freedom allowing the use of approximate subdomain solvers. When using approximate solvers with singular problems, the nullspace of the operator should be provided using PCBDDCSetNullSpace().

At the basis of the method there's the analysis of the connected components of the interface for the detection of vertices, edges and faces equivalence classes. Additional information on the degrees of freedom can be supplied to PCBDDC by using the following functions: PCBDDCSetDofsSplitting(), PCBDDCSetLocalAdjacencyGraph(), PCBDDCSetPrimalVerticesLocalIS(), PCBDDCSetNeumannBoundaries(), PCBDDCSetDirichletBoundaries(), PCBDDCSetNeumannBoundariesLocal(), PCBDDCSetDirichletBoundariesLocal().

Crucial for the convergence of the iterative process is the specification of the primal constraints to be imposed at the interface between subdomains. PCBDDC uses by default vertex continuities and edge arithmetic averages, which are enough for the three-dimensional Poisson problem with constant coeffcients. The user can switch on and off the usage of vertices, edges or face constraints by using the command line switches `-pc_bddc_use_vertices`, `-pc_bddc_use_edges`, `-pc_bddc_use_faces`. A customization of the constraints is available by attaching a MatNullSpace object to the preconditioning matrix via MatSetNearNullSpace(). The vectors of the MatNullSpace object should represent the constraints in the form of quadrature rules; quadrature rules for different classes of the interface can be listed in the same vector. The number of vectors of the MatNullSpace object corresponds to the maximum number of constraints that can be imposed for each class. Once all the quadrature rules for a given interface class have been ex-

tracted, an SVD operation is performed to retain the non-singular modes. As an example, the rigid body modes represent an effective choice for elasticity, even in the almost incompressible case. For particular problems, like e.g. edge-based discretization with Nedelec elements, a user defined change of basis of the degrees of freedom can be beneficial for PCBDDC; use PCBDDCSetChangeOfBasisMat() to customize the change of basis.

The BDDC method is usually robust with respect to jumps in the material parameters aligned with the interface; for PDEs with more than one material parameter you may also consider to use the so-called deluxe scaling, available via the command line switch `-pc_bddc_use_deluxe_scaling`. Other scalings are available, see PCISSetSubdomainScalingFactor(), PCISSetSubdomainDiagonalScaling() or PCISSetUseStiffnessScaling(). However, the convergence properties of the BDDC method degrades in presence of large jumps in the material coefficients not aligned with the interface; for such cases, PETSc has the capability of adaptively computing the primal constraints. Adaptive selection of constraints could be requested by specifying a threshold value at command line by using `-pc_bddc_adaptive_threshold x`. Valid values for the threshold `x` ranges from 1 to infinity, with smaller values corresponding to more robust preconditioners. For SPD problems in 2D, or in 3D with only face degrees of freedom (like in the case of Raviart-Thomas or Brezzi-Douglas-Marini elements), such a threshold is a very accurate estimator of the condition number of the resulting preconditioned operator. Since the adaptive selection of constraints for BDDC methods is still an active topic of research, its implementation is currently limited to SPD problems; moreover, because the technique requires the explicit knowledge of the local Schur complements, it needs the external package MUMPS.

When solving problems decomposed in thousands of subdomains or more, the solution of the BDDC coarse problem could become a bottleneck; in order to overcome this issue, the user could either consider to solve the parallel coarse problem on a subset of the communicator associated with PCBDDC by using the command line switch `-pc_bddc_coarse_redistribute`, or instead use a multilevel approach. The latter can be requested by specifying the number of requested level at command line (`-pc_bddc_levels`) or by using PCBDDCSetLevels(). An additional parameter (see PCBDDCSetCoarseningRatio()) controls the number of subdomains that will be generated at the next level; the larger the coarsening ratio, the lower the number of coarser subdomains.

PETSc currently provides an experimental interface to the FETI-DP method, which is the dual method of BDDC. Users interested in experimenting with FETI-DP should use PCBDDCCreateFETIDPOperators() in order access the Mat and PC objects needed by the FETI-DP method. The latter call should be performed after having called PCSetUp() on the PCBDDC object. PCBDDCMatFETIDPGetRHS() maps the original right-hand side to a suitable right-hand side for the FETI-DP solver. The physical solution can be recovered by using PCBDDCMatFETIDPGetSolution(). A stand-alone class for FETI-DP will be provided in future releases.

For further details, see the example `${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex59.c` and the online documentation for PCBDDC.

4.4.7 Shell Preconditioners

The shell preconditioner simply uses an application-provided routine to implement the preconditioner. To set this routine, one uses the command

PCShellSetApply(PC pc,PetscErrorCode (*apply)(PC,Vec,Vec));

Often a preconditioner needs access to an application-provided data structured. For this, one should use

PCShellSetContext(PC pc,void *ctx);

to set this data structure and

PCShellGetContext(PC pc,void **ctx);

to retrieve it in `apply`. The three routine arguments of `apply()` are the PC, the input vector, and the output vector, respectively.

For a preconditioner that requires some sort of "setup" before being used, that requires a new setup every time the operator is changed, one can provide a "setup" routine that is called every time the operator is changed (usually via KSPSetOperators()).

PCShellSetSetUp(PC pc,PetscErrorCode (*setup)(PC));

The argument to the "setup" routine is the same PC object which can be used to obtain the operators with PCGetOperators() and the application-provided data structure that was set with PCShellSetContext().

4.4.8 Combining Preconditioners

The PC type PCCOMPOSITE allows one to form new preconditioners by combining already defined preconditioners and solvers. Combining preconditioners usually requires some experimentation to find a combination of preconditioners that works better than any single method. It is a tricky business and is not recommended until your application code is complete and running and you are trying to improve performance. In many cases using a single preconditioner is better than a combination; an exception is the multigrid/multilevel preconditioners (solvers) that are always combinations of some sort, see Section 4.4.9.

Let B_1 and B_2 represent the application of two preconditioners of type `type1` and `type2`. The preconditioner $B = B_1 + B_2$ can be obtained with

PCSetType(pc,PCCOMPOSITE);
PCCompositeAddPC(pc,type1);
PCCompositeAddPC(pc,type2);

Any number of preconditioners may added in this way.

This way of combining preconditioners is called additive, since the actions of the preconditioners are added together. This is the default behavior. An alternative can be set with the option

PCCompositeSetType(PC pc,PCCompositeType PC_COMPOSITE_MULTIPLICATIVE);

In this form the new residual is updated after the application of each preconditioner and the next preconditioner applied to the next residual. For example, with two composed preconditioners: B_1 and B_2; $y = Bx$ is obtained from

$$\begin{aligned} y &= B_1 x \\ w_1 &= x - Ay \\ y &= y + B_2 w_1 \end{aligned}$$

Loosely, this corresponds to a Gauss-Seidel iteration, while additive corresponds to a Jacobi iteration.

Under most circumstances the multiplicative form requires one-half the number of iterations as the additive form; but the multiplicative form does require the application of A inside the preconditioner.

In the multiplicative version, the calculation of the residual inside the preconditioner can be done in two ways: using the original linear system matrix or using the matrix used to build the preconditioners `B1`, `B2`, etc. By default it uses the "preconditioner matrix", to use the `Amat` matrix use the option

PCSetUseAmat(PC pc);

The individual preconditioners can be accessed (in order to set options) via

PCCompositeGetPC(PC pc,int count,PC *subpc);

For example, to set the first sub preconditioners to use ILU(1)

PC subpc;
PCCompositeGetPC(pc,0,&subpc);
PCFactorSetFill(subpc,1);

These various options can also be set via the options database. For example, `-pc_type composite -pc_composite_pcs jacobi,ilu` causes the composite preconditioner to be used with two preconditioners: Jacobi and ILU. The option `-pc_composite_type multiplicative` initiates the multiplicative version of the algorithm, while `-pc_composite_type additive` the additive version. Using the `Amat` matrix is obtained with the option `-pc_use_amat`. One sets options for the sub-preconditioners with the extra prefix `-sub_N_` where `N` is the number of the sub-preconditioner. For example, `-sub_0_pc_ifactor_fill 0`.

PETSc also allows a preconditioner to be a complete linear solver. This is achieved with the PCKSP type.

PCSetType(PC pc,PCKSP PCKSP);
PCKSPGetKSP(pc,&ksp);
/* set any KSP/PC options */

From the command line one can use 5 iterations of bi-CG-stab with ILU(0) preconditioning as the preconditioner with `-pc_type ksp -ksp_pc_type ilu -ksp_ksp_max_it 5 -ksp_ksp_type bcgs`.

By default the inner KSP preconditioner uses the outer preconditioner matrix, `Pmat`, as the matrix to be solved in the linear system; to use the matrix that defines the linear system, `Amat` use the option

PCSetUseAmat(PC pc);

or at the command line with `-pc_use_amat`.

Naturally one can use a KSP preconditioner inside a composite preconditioner. For example, `-pc_type composite -pc_composite_pcs ilu,ksp -sub_1_pc_type jacobi -sub_1_ksp_max_it 10` uses two preconditioners: ILU(0) and 10 iterations of GMRES with Jacobi preconditioning. Though it is not clear whether one would ever wish to do such a thing.

4.4.9 Multigrid Preconditioners

A large suite of routines is available for using multigrid as a preconditioner. In the PC framework the user is required to provide the coarse grid solver, smoothers, restriction, and interpolation, as well as the code to calculate residuals. The PC package allows all of that to be wrapped up into a PETSc compliant preconditioner. We fully support both matrix-free and matrix-based multigrid solvers.

A multigrid preconditioner is created with the four commands

KSPCreate(MPI_Comm comm,KSP *ksp);
KSPGetPC(KSP ksp,PC *pc);
PCSetType(PC pc,PCMG);
PCMGSetLevels(pc,int levels,MPI_Comm *comms);

A large number of parameters affect the multigrid behavior. The command

PCMGSetType(PC pc,PCMGType mode);

indicates which form of multigrid to apply [28].

For standard V or W-cycle multigrids, one sets the `mode` to be `PC_MG_MULTIPLICATIVE`; for the additive form (which in certain cases reduces to the BPX method, or additive multilevel Schwarz, or multilevel diagonal scaling), one uses `PC_MG_ADDITIVE` as the `mode`. For a variant of full multigrid, one can use `PC_MG_FULL`, and for the Kaskade algorithm `PC_MG_KASKADE`. For the multiplicative and full multigrid options, one can use a W-cycle by calling

PCMGSetCycleType(PC pc,PCMGCycleType ctype);

with a value of `PC_MG_CYCLE_W` for `ctype`. The commands above can also be set from the options database. The option names are `-pc_mg_type [multiplicative, additive, full, kaskade]`, and `-pc_mg_cycle_type <ctype>`.

The user can control the amount of pre- and postsmoothing by using either the options `-pc_mg_smoothup m` and `-pc_mg_smoothdown n` or the routines

PCMGSetNumberSmoothUp(PC pc,int m);
PCMGSetNumberSmoothDown(PC pc,int n);

The multigrid routines, which determine the solvers and interpolation/restriction operators that are used, are mandatory. To set the coarse grid solver, one must call

PCMGGetCoarseSolve(PC pc,KSP *ksp);

and set the appropriate options in `ksp`. Similarly, the smoothers are setcontrolled by first calling

PCMGGetSmoother(PC pc,int level,KSP *ksp);

and then setting the various options in the `ksp`. For example,

PCMGGetSmoother(pc,1,&ksp); KSPSetOperators(ksp,A1,A1);

sets the matrix that defines the smoother on level 1 of the multigrid. While

PCMGGetSmoother(pc,1,&ksp); KSPGetPC(ksp,&pc); PCSetType(pc,PCSOR);

sets SOR as the smoother to use on level 1.

To use a different pre- and postsmoother, one should call the following routines instead.

PCMGGetSmootherUp(PC pc,int level,KSP *upksp);

and

PCMGGetSmootherDown(PC pc,int level,KSP *downksp);

Use

PCMGSetInterpolation(PC pc,int level,Mat P);

and

PCMGSetRestriction(PC pc,int level,Mat R);

to define the intergrid transfer operations. If only one of these is set, it's transpose will be used for the other.

It is possible for these interpolation operations to be matrix free (see Section 3.3), he or she should make sure that these operations are defined for the (matrix-free) matrices passed in. Note that this system is arranged so that if the interpolation is the transpose of the restriction, you can pass the same `mat` argument to both PCMGSetRestriction() and PCMGSetInterpolation().

On each level except the coarsest, one must also set the routine to compute the residual. The following command suffices:

PCMGSetResidual(PC pc,int level,PetscErrorCode (*residual)(Mat,Vec,Vec,Vec),Mat mat);

The residual() function normally does not need to be set if one's operator is stored in a Mat format. In certain circumstances, where it is much cheaper to calculate the residual directly, rather than through the usual formula $b - Ax$, the user may wish to provide an alternative.

Finally, the user may provide three work vectors for each level (except on the finest, where only the residual work vector is required). The work vectors are set with the commands

PCMGSetRhs(PC pc,int level,Vec b);
PCMGSetX(PC pc,int level,Vec x);
PCMGSetR(PC pc,int level,Vec r);

The PC references these vectors so you should call VecDestroy() when you are finished with them. If any of these vectors are not provided, the preconditioner will allocate them.

One can control the KSP and PC options used on the various levels (as well as the coarse grid) using the prefix `mg_levels_` (`mg_coarse_` for the coarse grid). For example,

-mg_levels_ksp_type cg

will cause the CG method to be used as the Krylov method for each level. Or

-mg_levels_pc_type ilu -mg_levels_pc_factor_levels 2

will cause the ILU preconditioner to be used on each level with two levels of fill in the incomplete factorization.

4.5 Solving Block Matrices

Block matrices represent an important class of problems in numerical linear algebra and offer the possibility of far more efficient iterative solvers than just treating the entire matrix as black box. In this section we are using the common linear algebra definition of block matrices where matrices are divided in a small (two, three or so) number of very large blocks. Where the blocks arise naturally from the underlying physics or discretization of the problem, for example, the velocity and pressure. Under a certain numbering of unknowns the matrix can be written as

$$\begin{pmatrix} A_{00} & A_{01} & A_{02} & A_{03} \\ A_{10} & A_{11} & A_{12} & A_{13} \\ A_{20} & A_{21} & A_{22} & A_{23} \\ A_{30} & A_{31} & A_{32} & A_{33} \end{pmatrix}.$$

Where each A_{ij} is an entire block. In a parallel computer the matrices are not stored explicitly this way however, each process will own some of the rows of A_{0*}, A_{1*} etc. On a process the blocks may be stored one block followed by another

$$\begin{pmatrix} A_{00_{00}} & A_{00_{01}} & A_{00_{02}} & \dots & A_{01_{00}} & A_{01_{02}} & \dots \\ A_{00_{10}} & A_{00_{11}} & A_{00_{12}} & \dots & A_{01_{10}} & A_{01_{12}} & \dots \\ A_{00_{20}} & A_{00_{21}} & A_{00_{22}} & \dots & A_{01_{20}} & A_{01_{22}} & \dots \\ \dots & & & & & & \\ A_{10_{00}} & A_{10_{01}} & A_{10_{02}} & \dots & A_{11_{00}} & A_{11_{02}} & \dots \\ A_{10_{10}} & A_{10_{11}} & A_{10_{12}} & \dots & A_{11_{10}} & A_{11_{12}} & \dots \\ \dots & & & & & & \end{pmatrix}$$

or interlaced, for example, with two blocks

$$\begin{pmatrix} A00_{00} & A01_{00} & A00_{01} & A01_{01} & \dots \\ A10_{00} & A11_{00} & A10_{01} & A11_{01} & \dots \\ \dots & & & & \\ A00_{10} & A01_{10} & A00_{11} & A01_{11} & \dots \\ A10_{10} & A11_{10} & A10_{11} & A11_{11} & \dots \\ \dots & & & & \end{pmatrix}.$$

Note that for interlaced storage the number of rows/columns of each block must be the same size. Matrices obtained with DMCreateMatrix() where the DM is a a DMDA are always stored interlaced. Block matrices can also be stored using the MATNEST format which holds separate assembled blocks. Each of these nested matrices is itself distributed in parallel. It is more efficient to use MATNEST with the methods described in this section because there are fewer copies and better formats (e.g. BAIJ or SBAIJ) can be used for the components, but it is not possible to use many other methods with MATNEST. See Section 3.1.3 for more on assembling block matrices without depending on a specific matrix format.

The PETSc PCFIELDSPLIT preconditioner is used to implement the "block" solvers in PETSc. There are three ways to provide the information that defines the blocks. If the matrices are stored as interlaced then PCFieldSplitSetFields() can be called repeatedly to indicate which fields belong to each block. More generally PCFieldSplitSetIS() can be used to indicate exactly which rows/columns of the matrix belong to a particular block. You can provide names for each block with these routines, if you do not provide names they are numbered from 0. With these two approaches the blocks may overlap (though generally they will not). If only one block is defined then the complement of the matrices is used to define the other block. Finally the option `-pc_fieldsplit_detect_saddle_point` causes two diagonal blocks to be found, one associated with all rows/columns that have zeros on the diagonals and the rest.

For simplicity in the rest of the section we restrict our matrices to two by two blocks. So the matrix is

$$\begin{pmatrix} A_{00} & A_{01} \\ A_{10} & A_{11} \end{pmatrix}.$$

On occasion the user may provide another matrix that is used to construct parts of the preconditioner

$$\begin{pmatrix} Ap_{00} & Ap_{01} \\ Ap_{10} & Ap_{11} \end{pmatrix}.$$

For notational simplicity define $ksp(A, Ap)$ to mean approximately solving a linear system using KSP with operator A and preconditioner built from matrix Ap.

For matrices defined with any number of blocks there are three "block" algorithms available: block Jacobi,

$$\begin{pmatrix} ksp(A_{00}, Ap_{00}) & 0 \\ 0 & ksp(A_{11}, Ap_{11}) \end{pmatrix}$$

block Gauss-Seidel,

$$\begin{pmatrix} I & 0 \\ 0 & A_{11}^{-1} \end{pmatrix} \begin{pmatrix} I & 0 \\ -A_{10} & I \end{pmatrix} \begin{pmatrix} A_{00}^{-1} & 0 \\ 0 & I \end{pmatrix}$$

which is implemented[1] as

$$\begin{pmatrix} I & 0 \\ 0 & ksp(A_{11}, Ap_{11}) \end{pmatrix} \left[\begin{pmatrix} 0 & 0 \\ 0 & I \end{pmatrix} + \begin{pmatrix} I & 0 \\ -A_{10} & -A_{11} \end{pmatrix} \begin{pmatrix} I & 0 \\ 0 & 0 \end{pmatrix} \right] \begin{pmatrix} ksp(A_{00}, Ap_{00}) & 0 \\ 0 & I \end{pmatrix}$$

[1] This may seem an odd way to implement since it involves the "extra" multiply by $-A_{11}$. The reason is this is implemented this way is that this approach works for any number of blocks that may overlap.

and symmetric block Gauss-Seidel

$$\begin{pmatrix} A_{00}^{-1} & 0 \\ 0 & I \end{pmatrix} \begin{pmatrix} I & -A_{01} \\ 0 & I \end{pmatrix} \begin{pmatrix} A_{00} & 0 \\ 0 & A_{11}^{-1} \end{pmatrix} \begin{pmatrix} I & 0 \\ -A_{10} & I \end{pmatrix} \begin{pmatrix} A_{00}^{-1} & 0 \\ 0 & I \end{pmatrix}.$$

These can be accessed with `-pc_fieldsplit_type <additive,multiplicative,symmetric_multiplicative>` or the function PCFieldSplitSetType(). The option prefixes for the internal KSPs are given by `-fieldsplit_name_`.

By default blocks A_{00}, A_{01} and so on are extracted out of `Pmat`, the matrix that the KSP uses to build the preconditioner, and not out of `Amat` (i.e., A itself). As discussed above in Sec. 4.4.8, however, it is possible to use `Amat` instead of `Pmat` by calling PCSetUseAmat(pc) or using `-pc_use_amat` on the command line. Alternatively, you can have PCFieldSplit extract the diagonal blocks A_{00}, A_{11} etc. out of `Amat` by calling PCFieldSplitSetDiagUseAmat(pc,`PETSC_TRUE`) or supplying command-line argument `-pc_fieldsplit_diag_use_amat`. Similarly, PCFieldSplitSetOffDiagUseAmat(pc,`PETSC_TRUE`) or `-pc_fieldsplit_off_diag_use_amat` will cause the off-diagonal blocks A_{01}, A_{10} etc. to be extracted out of `Amat`.

For two by two blocks only there are another family of solvers, based on Schur complements. The inverse of the Schur complement factorization is

$$\left[\begin{pmatrix} I & 0 \\ A_{10}A_{00}^{-1} & I \end{pmatrix} \begin{pmatrix} A_{00} & 0 \\ 0 & S \end{pmatrix} \begin{pmatrix} I & A_{00}^{-1}A_{01} \\ 0 & I \end{pmatrix}\right]^{-1}$$

$$\begin{pmatrix} I & A_{00}^{-1}A_{01} \\ 0 & I \end{pmatrix}^{-1} \begin{pmatrix} A_{00}^{-1} & 0 \\ 0 & S^{-1} \end{pmatrix} \begin{pmatrix} I & 0 \\ A_{10}A_{00}^{-1} & I \end{pmatrix}^{-1}$$

$$\begin{pmatrix} I & -A_{00}^{-1}A_{01} \\ 0 & I \end{pmatrix} \begin{pmatrix} A_{00}^{-1} & 0 \\ 0 & S^{-1} \end{pmatrix} \begin{pmatrix} I & 0 \\ -A_{10}A_{00}^{-1} & I \end{pmatrix}$$

$$\begin{pmatrix} A_{00}^{-1} & 0 \\ 0 & I \end{pmatrix} \begin{pmatrix} I & -A_{01} \\ 0 & I \end{pmatrix} \begin{pmatrix} A_{00} & 0 \\ 0 & S^{-1} \end{pmatrix} \begin{pmatrix} I & 0 \\ -A_{10} & I \end{pmatrix} \begin{pmatrix} A_{00}^{-1} & 0 \\ 0 & I \end{pmatrix}.$$

The preconditioner is accessed with `-pc_fieldsplit_type schur` and is implemented as

$$\begin{pmatrix} ksp(A_{00}, Ap_{00}) & 0 \\ 0 & I \end{pmatrix} \begin{pmatrix} I & -A_{01} \\ 0 & I \end{pmatrix} \begin{pmatrix} I & 0 \\ 0 & ksp(\hat{S}, \hat{S}p) \end{pmatrix} \begin{pmatrix} I & 0 \\ -A_{10}ksp(A_{00}, Ap_{00}) & I \end{pmatrix}.$$

Where $\hat{S} = A_{11} - A_{10}ksp(A_{00}, Ap_{00})A_{01}$ is the approximate Schur complement.

There are several variants of the Schur complement preconditioner obtained by dropping some of the terms, these can be obtained with `-pc_fieldsplit_schur_fact_type <diag,lower,upper,full>` or the function PCFieldSplitSetSchurFactType(). Note that the `diag` form uses the preconditioner

$$\begin{pmatrix} ksp(A_{00}, Ap_{00}) & 0 \\ 0 & -ksp(\hat{S}, \hat{S}p) \end{pmatrix}.$$

This is done to ensure the preconditioner is positive definite for a common class of problems, saddle points with a positive definite A_{00}: for these the Schur complement is negative definite. SHOULD THERE BE A FLAG TO SET THE SIGN?

The effectiveness of the Schur complement preconditioner depends on the availability of a good preconditioner $\hat{S}p$ for the Schur complement matrix. In general, you are responsible for supplying $\hat{S}p$ via PCFieldSplitSchurPrecondition(pc,PC_FIELDSPLIT_SCHUR_PRE_USER,Sp). In the absence of a good problem-specific $\hat{S}p$, you can use some of the built-in options.

Using `-pc_fieldsplit_schur_precondition user` on the command line activates the matrix supplied programmatically as explained above.

With `-pc_fieldsplit_schur_precondition a11` (default) $\hat{S}p = A_{11}$ is used to build a preconditioner for $\hat{S}$.

Otherwise, `-pc_fieldsplit_schur_precondition self` will set $\hat{S}p = \hat{S}$ and use the Schur complement matrix itself to build the preconditioner.

The problem with the last approach is that $\hat{S}$ is used in unassembled, matrix-free form, and many preconditioners (e.g., ILU) cannot be built out of such matrices. Instead, you can *assemble* an approximation to $\hat{S}$ by inverting A_{00}, but only approximately, so as to ensure the sparsity of $\hat{S}p$ as much as possible. Specifically, using `-pc_fieldsplit_schur_precondition selfp` will assemble $\hat{S}p = A_{11} - A_{10}inv(A_{00})A_{01}$.

By default $inv(A_{00})$ is the inverse of the diagonal of A_{00}, but using `-fieldsplit_1_mat_schur_complement_ainv_type lump` will lump A_{00} first. Option `-mat_schur_complement_ainv_type` applies to any matrix of MatSchurComplement type and here it is used with the prefix `-fieldsplit_1` of the linear system in the second split.

Finally, you can use the PCLSC preconditioner for the Schur complement with `-pc_fieldsplit_type schur -fieldsplit_1_pc_type lsc`. This uses for the preconditioner to $\hat{S}$ the operator

$$ksp(A_{10}A_{01}, A_{10}A_{01})A_{10}A_{00}A_{01}ksp(A_{10}A_{01}, A_{10}A_{01})$$

which, of course, introduces two additional inner solves for each application of the Schur complement. The options prefix for this inner KSP is `-fieldsplit_1_lsc_`. Instead of constructing the matrix $A_{10}A_{01}$ the user can provide their own matrix. This is done by attaching the matrix/matrices to the Sp matrix they provide with

```
PetscObjectCompose((PetscObject)Sp,"LSC_L",(PetscObject)L);
PetscObjectCompose((PetscObject)Sp,"LSC_Lp",(PetscObject)Lp);
```

4.6 Solving Singular Systems

Sometimes one is required to solver linear systems that are singular. That is systems with the matrix has a null space. For example, the discretization of the Laplacian operator with Neumann boundary conditions as a null space of the constant functions. PETSc has tools to help solve these systems.

First, one must know what the null space is and store it using an orthonormal basis in an array of PETSc Vecs. (The constant functions can be handled separately, since they are such a common case). Create a MatNullSpace object with the command

```
MatNullSpaceCreate(MPI_Comm,PetscBool hasconstants,int dim,Vec *basis,MatNullSpace *nsp);
```

Here `dim` is the number of vectors in `basis` and `hasconstants` indicates if the null space contains the constant functions. (If the null space contains the constant functions you do not need to include it in the `basis` vectors you provide).

One then tells the KSP object you are using what the null space is with the call

```
MatSetNullSpace(Mat Amat,MatNullSpace nsp); MatSetTransposeNullSpace(Mat Amat,MatNullSpace nsp);
```

The Amat should be the **first** matrix argument used with KSPSetOperators(), SNESSetJacobian(), or TSSetIJacobian() The PETSc solvers will now handle the null space during the solution process.

But if one chooses a direct solver (or an incomplete factorization) it may still detect a zero pivot. You can run with the additional options or `-pc_factor_shift_type NONZERO -pc_factor_shift_amount <dampingfactor>` to prevent the zero pivot. A good choice for the `dampingfactor` is 1.e-10.

4.7 Using PETSc to interface with external linear solvers

PETSc interfaces to several external linear solvers (see Acknowledgments). To use these solvers, one needs to:

1. Run ./configure with the additional options `--download-packagename`. For eg: `--download-superlu_dist --download-parmetis` (SuperLU_DIST needs ParMetis) or `--download-mumps --download-scalapack` (MUMPS requires ScaLAPACK).
2. Build the PETSc libraries.
3. Use the runtime option: `-ksp_type preonly -pc_type <pctype> -pc_factor_mat_solver_package <packagename>`. For eg: `-ksp_type preonly -pc_type lu -pc_factor_mat_solver_package superlu_dist`.

MatType	PCType	MatSolverPackage	Package (`-pc_factor_mat_solver_package`)
seqaij	lu	MATSOLVERESSL	`essl`
seqaij	lu	MATSOLVERLUSOL	`lusol`
seqaij	lu	MATSOLVERMATLAB	`matlab`
aij	lu	MATSOLVERMUMPS	`mumps`
aij	cholesky		
sbaij	cholesky		
seqaij	lu	MATSOLVERSUPERLU	`superlu`
aij	lu	MATSOLVERSUPERLU_DIST	`superlu_dist`
seqaij	lu	MATSOLVERUMFPACK	`umfpack`

Table 5: Options for External Solvers

The default and available input options for each external software can be found by specifying -help (or -h) at runtime.

As an alternative to using runtime flags to employ these external packages, procedural calls are provided for some packages. For example, following procedural calls are equivalent to runtime options `-ksp_type preonly -pc_type lu -pc_factor_mat_solver_package mumps -mat_mumps_icntl_7 2`:

```
KSPSetType(ksp,KSPPREONLY);
KSPGetPC(ksp,&pc);
PCSetType(pc,PCLU);
PCFactorSetMatSolverPackage(pc,MATSOLVERMUMPS);
```

```
PCFactorGetMatrix(pc,&F);
icntl=7; ival = 2;
MatMumpsSetIcntl(F,icntl,ival);
```

One can also create matrices with the appropriate capabilities by calling MatCreate() followed by MatSetType() specifying the desired matrix type from Table 5. These matrix types inherit capabilities from their PETSc matrix parents: seqaij, mpiaij, etc. As a result, the preallocation routines MatSeqAIJSetPreallocation(), MatMPIAIJSetPreallocation(), etc. and any other type specific routines of the base class are supported. One can also call MatConvert() inplace to convert the matrix to and from its base class without performing an expensive data copy. MatConvert() cannot be called on matrices that have already been factored.

In Table 5, the base class aij refers to the fact that inheritance is based on MATSEQAIJ when constructed with a single process communicator, and from MATMPIAIJ otherwise. The same holds for baij and sbaij. For codes that are intended to be run as both a single process or with multiple processes, depending on the mpiexec command, it is recommended that both sets of preallocation routines are called for these communicator morphing types. The call for the incorrect type will simply be ignored without any harm or message.

Chapter 5

SNES: Nonlinear Solvers

The solution of large-scale nonlinear problems pervades many facets of computational science and demands robust and flexible solution strategies. The SNES library of PETSc provides a powerful suite of data-structure-neutral numerical routines for such problems. Built on top of the linear solvers and data structures discussed in preceding chapters, SNES enables the user to easily customize the nonlinear solvers according to the application at hand. Also, the SNES interface is *identical* for the uniprocess and parallel cases; the only difference in the parallel version is that each process typically forms only its local contribution to various matrices and vectors.

The SNES class includes methods for solving systems of nonlinear equations of the form

$$\boldsymbol{F}(\boldsymbol{x}) = 0, \tag{5.1}$$

where $\boldsymbol{F} : \Re^n \to \Re^n$. Newton-like methods provide the core of the package, including both line search and trust region techniques. A suite of nonlinear Krylov methods and methods based upon problem decomposition are also included. The solvers are discussed further in Section 5.2. Following the PETSc design philosophy, the interfaces to the various solvers are all virtually identical. In addition, the SNES software is completely flexible, so that the user can at runtime change any facet of the solution process.

PETSc's default method for solving the nonlinear equation is Newton's method. The general form of the n-dimensional Newton's method for solving (5.1) is

$$\boldsymbol{x}_{k+1} = \boldsymbol{x}_k - \boldsymbol{J}(\boldsymbol{x}_k)^{-1}\boldsymbol{F}(\boldsymbol{x}_k), \quad k = 0, 1, \ldots, \tag{5.2}$$

where $\boldsymbol{x}_0$ is an initial approximation to the solution and $\boldsymbol{J}(\boldsymbol{x}_k) = \boldsymbol{F}'(\boldsymbol{x}_k)$, the Jacobian, is nonsingular at each iteration. In practice, the Newton iteration (5.2) is implemented by the following two steps:

$$1. \quad \text{(Approximately) solve} \quad \boldsymbol{J}(\boldsymbol{x}_k)\Delta\boldsymbol{x}_k = -\boldsymbol{F}(\boldsymbol{x}_k). \tag{5.3}$$

$$2. \quad \text{Update} \quad \boldsymbol{x}_{k+1} = \boldsymbol{x}_k + \Delta\boldsymbol{x}_k. \tag{5.4}$$

Other defect-correction algorithms can be implemented by using different choices for $J(\boldsymbol{x}_k)$.

5.1 Basic SNES Usage

In the simplest usage of the nonlinear solvers, the user must merely provide a C, C++, or Fortran routine to evaluate the nonlinear function of Equation (5.1). The corresponding Jacobian matrix can be approximated with finite differences. For codes that are typically more efficient and accurate, the user can provide a routine to compute the Jacobian; details regarding these application-provided routines are discussed below. To provide an overview of the use of the nonlinear solvers, we first introduce a complete and simple example in Figure 13, corresponding to `${PETSC_DIR}/src/snes/examples/tutorials/ex1.c`.

```
static char help[] = "Newton's method for a two-variable system, sequential.\n\n";

/*T
   Concepts: SNES^basic example
T*/

/*
   Include "petscsnes.h" so that we can use SNES solvers.  Note that this
   file automatically includes:
     petscsys.h       - base PETSc routines   petscvec.h - vectors
     petscmat.h - matrices
     petscis.h     - index sets            petscksp.h - Krylov subspace methods
     petscviewer.h - viewers               petscpc.h  - preconditioners
     petscksp.h   - linear solvers
*/
/*F
This examples solves either
\begin{equation}
  F\genfrac{(}{)}{0pt}{}{x_0}{x_1} = \genfrac{(}{)}{0pt}{}{x^2_0 + x_0 x_1
- 3}{x_0 x_1 + x^2_1 - 6}
\end{equation}
or if the {\tt -hard} options is given
\begin{equation}
  F\genfrac{(}{)}{0pt}{}{x_0}{x_1} = \genfrac{(}{)}{0pt}{}{\sin(3 x_0) + x_0}{x_1}
\end{equation}
F*/
#include <petscsnes.h>

/*
   User-defined routines
*/
extern PetscErrorCode FormJacobian1(SNES,Vec,Mat,Mat,void*);
extern PetscErrorCode FormFunction1(SNES,Vec,Vec,void*);
extern PetscErrorCode FormJacobian2(SNES,Vec,Mat,Mat,void*);
extern PetscErrorCode FormFunction2(SNES,Vec,Vec,void*);

#undef __FUNCT__
#define __FUNCT__ "main"
int main(int argc,char **argv)
{
  SNES           snes;         /* nonlinear solver context */
  KSP            ksp;          /* linear solver context */
  PC             pc;           /* preconditioner context */
  Vec            x,r;          /* solution, residual vectors */
  Mat            J;            /* Jacobian matrix */
  PetscErrorCode ierr;
  PetscInt       its;
  PetscMPIInt    size;
  PetscScalar    pfive = .5,*xx;
  PetscBool      flg;

  PetscInitialize(&argc,&argv,(char*)0,help);
  ierr = MPI_Comm_size(PETSC_COMM_WORLD,&size);CHKERRQ(ierr);
```

```
  if (size > 1) SETERRQ(PETSC_COMM_WORLD,PETSC_ERR_SUP,"Example is only for
sequential runs");

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Create nonlinear solver context
     - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */
  ierr = SNESCreate(PETSC_COMM_WORLD,&snes);CHKERRQ(ierr);

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Create matrix and vector data structures; set corresponding routines
     - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
*/
  /*
     Create vectors for solution and nonlinear function
  */
  ierr = VecCreate(PETSC_COMM_WORLD,&x);CHKERRQ(ierr);
  ierr = VecSetSizes(x,PETSC_DECIDE,2);CHKERRQ(ierr);
  ierr = VecSetFromOptions(x);CHKERRQ(ierr);
  ierr = VecDuplicate(x,&r);CHKERRQ(ierr);

  /*
     Create Jacobian matrix data structure
  */
  ierr = MatCreate(PETSC_COMM_WORLD,&J);CHKERRQ(ierr);
  ierr = MatSetSizes(J,PETSC_DECIDE,PETSC_DECIDE,2,2);CHKERRQ(ierr);
  ierr = MatSetFromOptions(J);CHKERRQ(ierr);
  ierr = MatSetUp(J);CHKERRQ(ierr);

  ierr = PetscOptionsHasName(NULL,NULL,"-hard",&flg);CHKERRQ(ierr);
  if (!flg) {
    /*
     Set function evaluation routine and vector.
    */
    ierr = SNESSetFunction(snes,r,FormFunction1,NULL);CHKERRQ(ierr);

    /*
     Set Jacobian matrix data structure and Jacobian evaluation routine
    */
    ierr = SNESSetJacobian(snes,J,J,FormJacobian1,NULL);CHKERRQ(ierr);
  } else {
    ierr = SNESSetFunction(snes,r,FormFunction2,NULL);CHKERRQ(ierr);
    ierr = SNESSetJacobian(snes,J,J,FormJacobian2,NULL);CHKERRQ(ierr);
  }

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Customize nonlinear solver; set runtime options
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */
  /*
     Set linear solver defaults for this problem. By extracting the
     KSP and PC contexts from the SNES context, we can then
     directly call any KSP and PC routines to set various options.
  */
  ierr = SNESGetKSP(snes,&ksp);CHKERRQ(ierr);
  ierr = KSPGetPC(ksp,&pc);CHKERRQ(ierr);
  ierr = PCSetType(pc,PCNONE);CHKERRQ(ierr);
```

```
  ierr = KSPSetTolerances(ksp,1.e-4,PETSC_DEFAULT,PETSC_DEFAULT,20);CHKERRQ(ierr);

  /*
     Set SNES/KSP/KSP/PC runtime options, e.g.,
         -snes_view -snes_monitor -ksp_type <ksp> -pc_type <pc>
     These options will override those specified above as long as
     SNESSetFromOptions() is called _after_ any other customization
     routines.
  */
  ierr = SNESSetFromOptions(snes);CHKERRQ(ierr);

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Evaluate initial guess; then solve nonlinear system
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */
  if (!flg) {
    ierr = VecSet(x,pfive);CHKERRQ(ierr);
  } else {
    ierr  = VecGetArray(x,&xx);CHKERRQ(ierr);
    xx[0] = 2.0; xx[1] = 3.0;
    ierr  = VecRestoreArray(x,&xx);CHKERRQ(ierr);
  }
  /*
     Note: The user should initialize the vector, x, with the initial guess
     for the nonlinear solver prior to calling SNESSolve().  In particular,
     to employ an initial guess of zero, the user should explicitly set
     this vector to zero by calling VecSet().
  */

  ierr = SNESSolve(snes,NULL,x);CHKERRQ(ierr);
  ierr = SNESGetIterationNumber(snes,&its);CHKERRQ(ierr);
  if (flg) {
    Vec f;
    ierr = VecView(x,PETSC_VIEWER_STDOUT_WORLD);CHKERRQ(ierr);
    ierr = SNESGetFunction(snes,&f,0,0);CHKERRQ(ierr);
    ierr = VecView(r,PETSC_VIEWER_STDOUT_WORLD);CHKERRQ(ierr);
  }

  ierr = PetscPrintf(PETSC_COMM_WORLD,"Number of SNES iterations = %D\n",its);CHKERRQ(ierr)

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Free work space.  All PETSc objects should be destroyed when they
     are no longer needed.
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

  ierr = VecDestroy(&x);CHKERRQ(ierr); ierr = VecDestroy(&r);CHKERRQ(ierr);
  ierr = MatDestroy(&J);CHKERRQ(ierr); ierr = SNESDestroy(&snes);CHKERRQ(ierr);
  ierr = PetscFinalize();
  return 0;
}
/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "FormFunction1"
/*
   FormFunction1 - Evaluates nonlinear function, F(x).
```

```
   Input Parameters:
.  snes - the SNES context
.  x    - input vector
.  ctx  - optional user-defined context

   Output Parameter:
.  f - function vector
 */
PetscErrorCode FormFunction1(SNES snes,Vec x,Vec f,void *ctx)
{
  PetscErrorCode    ierr;
  const PetscScalar *xx;
  PetscScalar       *ff;

  /*
   Get pointers to vector data.
      - For default PETSc vectors, VecGetArray() returns a pointer to
        the data array.  Otherwise, the routine is implementation dependent.
      - You MUST call VecRestoreArray() when you no longer need access to
        the array.
   */
  ierr = VecGetArrayRead(x,&xx);CHKERRQ(ierr);
  ierr = VecGetArray(f,&ff);CHKERRQ(ierr);

  /* Compute function */
  ff[0] = xx[0]*xx[0] + xx[0]*xx[1] - 3.0;
  ff[1] = xx[0]*xx[1] + xx[1]*xx[1] - 6.0;

  /* Restore vectors */
  ierr = VecRestoreArrayRead(x,&xx);CHKERRQ(ierr);
  ierr = VecRestoreArray(f,&ff);CHKERRQ(ierr);
  return 0;
}
/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "FormJacobian1"
/*
   FormJacobian1 - Evaluates Jacobian matrix.

   Input Parameters:
.  snes - the SNES context
.  x - input vector
.  dummy - optional user-defined context (not used here)

   Output Parameters:
.  jac - Jacobian matrix
.  B - optionally different preconditioning matrix
.  flag - flag indicating matrix structure
*/
PetscErrorCode FormJacobian1(SNES snes,Vec x,Mat jac,Mat B,void *dummy)
{
  const PetscScalar *xx;
  PetscScalar       A[4];
  PetscErrorCode    ierr;
  PetscInt          idx[2] = {0,1};
```

```
  /*
     Get pointer to vector data
  */
  ierr = VecGetArrayRead(x,&xx);CHKERRQ(ierr);

  /*
     Compute Jacobian entries and insert into matrix.
      - Since this is such a small problem, we set all entries for
        the matrix at once.
  */
  A[0]  = 2.0*xx[0] + xx[1]; A[1] = xx[0];
  A[2]  = xx[1]; A[3] = xx[0] + 2.0*xx[1];
  ierr  = MatSetValues(B,2,idx,2,idx,A,INSERT_VALUES);CHKERRQ(ierr);

  /*
     Restore vector
  */
  ierr = VecRestoreArrayRead(x,&xx);CHKERRQ(ierr);

  /*
     Assemble matrix
  */
  ierr = MatAssemblyBegin(B,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
  ierr = MatAssemblyEnd(B,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
  if (jac != B) {
    ierr = MatAssemblyBegin(jac,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
    ierr = MatAssemblyEnd(jac,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
  }
  return 0;
}

/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "FormFunction2"
PetscErrorCode FormFunction2(SNES snes,Vec x,Vec f,void *dummy)
{
  PetscErrorCode    ierr;
  const PetscScalar *xx;
  PetscScalar       *ff;

  /*
     Get pointers to vector data.
       - For default PETSc vectors, VecGetArray() returns a pointer to
         the data array.  Otherwise, the routine is implementation dependent.
       - You MUST call VecRestoreArray() when you no longer need access to
         the array.
  */
  ierr = VecGetArrayRead(x,&xx);CHKERRQ(ierr);
  ierr = VecGetArray(f,&ff);CHKERRQ(ierr);

  /*
     Compute function
  */
  ff[0] = PetscSinScalar(3.0*xx[0]) + xx[0];
```

```
  ff[1] = xx[1];

  /*
     Restore vectors
  */
  ierr = VecRestoreArrayRead(x,&xx);CHKERRQ(ierr);
  ierr = VecRestoreArray(f,&ff);CHKERRQ(ierr);
  return 0;
}
/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "FormJacobian2"
PetscErrorCode FormJacobian2(SNES snes,Vec x,Mat jac,Mat B,void *dummy)
{
  const PetscScalar *xx;
  PetscScalar       A[4];
  PetscErrorCode    ierr;
  PetscInt          idx[2] = {0,1};

  /*
     Get pointer to vector data
  */
  ierr = VecGetArrayRead(x,&xx);CHKERRQ(ierr);

  /*
     Compute Jacobian entries and insert into matrix.
      - Since this is such a small problem, we set all entries for
        the matrix at once.
  */
  A[0]  = 3.0*PetscCosScalar(3.0*xx[0]) + 1.0; A[1] = 0.0;
  A[2]  = 0.0;                      A[3] = 1.0;
  ierr  = MatSetValues(B,2,idx,2,idx,A,INSERT_VALUES);CHKERRQ(ierr);

  /*
     Restore vector
  */
  ierr = VecRestoreArrayRead(x,&xx);CHKERRQ(ierr);

  /*
     Assemble matrix
  */
  ierr = MatAssemblyBegin(B,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
  ierr = MatAssemblyEnd(B,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
  if (jac != B) {
    ierr = MatAssemblyBegin(jac,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
    ierr = MatAssemblyEnd(jac,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
  }
  return 0;
}
```

Figure 13: Example of Uniprocess SNES Code

To create a SNES solver, one must first call SNESCreate() as follows:

SNESCreate(MPI_Comm comm,SNES *snes);

The user must then set routines for evaluating the function of equation (5.1) and its associated Jacobian matrix, as discussed in the following sections.

To choose a nonlinear solution method, the user can either call

SNESSetType(SNES snes,SNESType method);

or use the option `-snes_type <method>`, where details regarding the available methods are presented in Section 5.2. The application code can take complete control of the linear and nonlinear techniques used in the Newton-like method by calling

SNESSetFromOptions(snes);

This routine provides an interface to the PETSc options database, so that at runtime the user can select a particular nonlinear solver, set various parameters and customized routines (e.g., specialized line search variants), prescribe the convergence tolerance, and set monitoring routines. With this routine the user can also control all linear solver options in the KSP, and PC modules, as discussed in Chapter 4.

After having set these routines and options, the user solves the problem by calling

SNESSolve(SNES snes,Vec b,Vec x);

where `x` indicates the solution vector. The user should initialize this vector to the initial guess for the nonlinear solver prior to calling SNESSolve(). In particular, to employ an initial guess of zero, the user should explicitly set this vector to zero by calling VecSet(). Finally, after solving the nonlinear system (or several systems), the user should destroy the SNES context with

SNESDestroy(SNES *snes);

5.1.1 Nonlinear Function Evaluation

When solving a system of nonlinear equations, the user must provide a vector, `f`, for storing the function of Equation (5.1), as well as a routine that evaluates this function at the vector `x`. This information should be set with the command

SNESSetFunction(SNES snes,Vec f,
PetscErrorCode (*FormFunction)(SNES snes,Vec x,Vec f,void *ctx),void *ctx);

The argument `ctx` is an optional user-defined context, which can store any private, application-specific data required by the function evaluation routine; NULL should be used if such information is not needed. In C and C++, a user-defined context is merely a structure in which various objects can be stashed; in Fortran a user context can be an integer array that contains both parameters and pointers to PETSc objects. `${PETSC_DIR}/src/snes/examples/tutorials/ex5.c` and `${PETSC_DIR}/src/snes/examples/tutorials/ex5f.F` give examples of user-defined application contexts in C and Fortran, respectively.

5.1.2 Jacobian Evaluation

The user must also specify a routine to form some approximation of the Jacobian matrix, `A`, at the current iterate, `x`, as is typically done with

SNESSetJacobian(SNES snes,Mat Amat,Mat Pmat,PetscErrorCode (*FormJacobian)(SNES snes,
Vec x,Mat A,Mat B,void *ctx),void *ctx);

The arguments of the routine `FormJacobian()` are the current iterate, `x`; the (approximate) Jacobian matrix, `Amat`; the matrix from which the preconditioner is constructed, `Pmat` (which is usually the same as `Amat`); a `flag` indicating information about the preconditioner matrix structure; and an optional user-defined Jacobian context, `ctx`, for application-specific data. The options for `flag` are identical to those for the flag of KSPSetOperators(), discussed in Section 4.1. Note that the SNES solvers are all data-structure neutral, so the full range of PETSc matrix formats (including "matrix-free" methods) can be used. Chapter 3 discusses information regarding available matrix formats and options, while Section 5.5 focuses on matrix-free methods in SNES. We briefly touch on a few details of matrix usage that are particularly important for efficient use of the nonlinear solvers.

A common usage paradigm is to assemble the problem Jacobian in the preconditioner storage `B`, rather than `A`. In the case where they are identical, as in many simulations, this makes no difference. However, it allows us to check the analytic Jacobian we construct in `FormJacobian()` by passing the `-snes_mf_operator` flag. This causes PETSc to approximate the Jacobian using finite differencing of the function evaluation (discussed in section 5.6), and the analytic Jacobian becomes merely the preconditioner. Even if the analytic Jacobian is incorrect, it is likely that the finite difference approximation will converge, and thus this is an excellent method to verify the analytic Jacobian. Moreover, if the analytic Jacobian is incomplete (some terms are missing or approximate), `-snes_mf_operator` may be used to obtain the exact solution, where the Jacobian approximation has been transferred to the preconditioner.

One such approximate Jacobian comes from "Picard linearization" which writes the nonlinear system as

$$\boldsymbol{F}(\boldsymbol{x}) = \boldsymbol{A}(\boldsymbol{x})\boldsymbol{x} - \boldsymbol{b} = 0$$

where $\boldsymbol{A}(\boldsymbol{x})$ usually contains the lower-derivative parts of the equation. For example, the nonlinear diffusion problem

$$-\nabla \cdot (\kappa(u)\nabla u) = 0$$

would be linearized as

$$A(u)v \simeq -\nabla \cdot (\kappa(u)\nabla v).$$

Usually this linearization is simpler to implement than Newton and the linear problems are somewhat easier to solve. In addition to using `-snes_mf_operator` with this approximation to the Jacobian, the Picard iterative procedure can be performed by defining $\boldsymbol{J}(\boldsymbol{x})$ to be $\boldsymbol{A}(\boldsymbol{x})$. Sometimes this iteration exhibits better global convergence than Newton linearization.

During successive calls to `FormJacobian()`, the user can either insert new matrix contexts or reuse old ones, depending on the application requirements. For many sparse matrix formats, reusing the old space (and merely changing the matrix elements) is more efficient; however, if the matrix structure completely changes, creating an entirely new matrix context may be preferable. Upon subsequent calls to the `FormJacobian()` routine, the user may wish to reinitialize the matrix entries to zero by calling MatZeroEntries(). See Section 3.4 for details on the reuse of the matrix context.

The directory `${PETSC_DIR}/src/snes/examples/tutorials` provides a variety of examples.

5.2 The Nonlinear Solvers

As summarized in Table 6, SNES includes several Newton-like nonlinear solvers based on line search techniques and trust region methods. Also provided are several nonlinear Krylov methods, as well as nonlinear methods involving decompositions of the problem.

Each solver may have associated with it a set of options, which can be set with routines and options database commands provided for this purpose. A complete list can be found by consulting the manual pages or by running a program with the `-help` option; we discuss just a few in the sections below.

Method	SNESType	Options Name	Default Line Search
Line search Newton	SNESNEWTONLS	newtonls	SNESLINESEARCHBT
Trust region Newton	SNESNEWTONTR	newtontr	–
Test Jacobian	SNESTEST	test	–
Nonlinear Richardson	SNESNRICHARDSON	nrichardson	SNESLINESEARCHL2
Nonlinear CG	SNESNCG	ncg	SNESLINESEARCHCP
Nonlinear GMRES	SNESNGMRES	ngmres	SNESLINESEARCHL2
Quasi-Newton	SNESQN	qn	see Table 8
FAS	SNESFAS	fas	–
Nonlinear ASM	SNESNASM	nasm	–
Composite	SNESCOMPOSITE	composite	–
Nonlinear Gauss-Seidel	SNESGS	ngs	–
Anderson Mixing	SNESAnderson	anderson	–
Shell	SNESSHELL	shell	–

Table 6: PETSc Nonlinear Solvers

Line Search	SNESLineSearchType	Options Name
Backtracking	SNESLINESEARCHBT	bt
(damped) step	SNESLINESEARCHBASIC	basic
L2-norm Minimization	SNESLINESEARCHL2	l2
Critical point	SNESLINESEARCHCP	cp
Shell	SNESLINESEARCHSHELL	shell

Table 7: PETSc Line Search Methods

5.2.1 Line Search Newton

The method SNESNEWTONLS (`-snes_type newtonls`) provides a line search Newton method for solving systems of nonlinear equations. By default, this technique employs cubic backtracking [8]. Alternative line search techniques are listed in Table 7.

Every SNES has a line search context of type SNESLineSearch that may be retrieved using

SNESGetLineSearch(SNES snes,SNESLineSearch *ls);.

There are several default options for the line searches. The order of polynomial approximation may be set with `-snes_linesearch_order` or

SNESLineSearchSetOrder(SNESLineSearch ls, PetscInt order);

for instance, 2 for quadratic or 3 for cubic. Sometimes, it may not be necessary to monitor the progress of the nonlinear iteration. In this case, `-snes_linesearch_norms` or

SNESLineSearchSetNorms(SNESLineSearch ls,PetscBool norms);

may be used to turn off function, step, and solution norm computation at the end of the linesearch.

The default line search for the line search Newton method, SNESLINESEARCHBT involves several parameters, which are set to defaults that are reasonable for many applications. The user can override the defaults by using the options

`-snes_linesearch_alpha <alpha>`, `-snes_linesearch_maxstep <max>`, and `-snes_linesearch_minlambda <tol>`.

Besides the backtracking linesearch, there are SNESLINESEARCHL2, which uses a polynomial secant minimization of $||F(x)||_2$, and SNESLINESEARCHCP, which minimizes $F(x) \cdot Y$ where Y is the search direction. These are both potentially iterative line searches, which may be used to find a better-fitted steplength in the case where a single secant search is not sufficient. The number of iterations may be set with `-snes_linesearch_max_it`. In addition, the convergence criteria of the iterative line searches may be set using function tolerances `-snes_linesearch_rtol` and `-snes_linesearch_atol`, and steplength tolerance `snes_linesearch_ltol`.

Custom line search types may either be defined using SNESLineSearchShell, or by creating a custom user line search type in the model of the preexisting ones and register it using

SNESLineSearchRegister(const char sname[],PetscErrorCode (*function)(SNESLineSearch));.

5.2.2 Trust Region Methods

The trust region method in SNES for solving systems of nonlinear equations, SNESNEWTONTR (`-snes_type newtontr`), is taken from the MINPACK project [21]. Several parameters can be set to control the variation of the trust region size during the solution process. In particular, the user can control the initial trust region radius, computed by

$$\Delta = \Delta_0 \|F_0\|_2,$$

by setting Δ_0 via the option `-snes_tr_delta0 <delta0>`.

5.2.3 Nonlinear Krylov Methods

A number of nonlinear Krylov methods are provided, including Nonlinear richardson, conjugate gradient, GMRES, and Anderson Mixing. These methods are described individually below. They are all instrumental to PETSc's nonlinear preconditioning.

Nonlinear Richardson. The nonlinear Richardson iteration merely takes the form of a line search-damped fixed-point iteration of the form

$$\boldsymbol{x}_{k+1} = \boldsymbol{x}_k - \lambda \boldsymbol{F}(\boldsymbol{x}_k), \quad k = 0, 1, \ldots, \tag{5.5}$$

where the default linesearch is SNESLINESEARCHL2. This simple solver is mostly useful as a nonlinear smoother, or to provide line search stabilization to an inner method.

Nonlinear Conjugate Gradients. Nonlinear CG is equivalent to linear CG, but with the steplength determined by line search (SNESLINESEARCHCP by default). Five variants (Fletcher-Reed, Hestenes-Steifel, Polak-Ribiere-Polyak, Dai-Yuan, and Conjugate Descent) are implemented in PETSc and may be chosen using

SNESNCGSetType(SNES snes, SNESNCGType btype);.

Anderson Mixing and Nonlinear GMRES Methods. Nonlinear GMRES and Anderson Mixing methods combine the last m iterates, plus a new fixed-point iteration iterate, into a residual-minimizing new iterate.

5.2.4 Quasi-Newton Methods

Quasi-Newton methods store iterative rank-one updates to the Jacobian instead of computing it directly. Three limited-memory quasi-Newton methods are provided, L-BFGS, which are described in Table 8. These all are encapsulated under `-snes_type qn` and may be changed with `snes_qn_type`. The default is L-BFGS, which provides symmetric updates to an approximate Jacobian. This iteration is similar to the line search Newton methods.

One may also control the form of the initial Jacobian approximation with

QN Method	SNESQNType	Options Name	Default Line Search
L-BFGS	SNES_QN_LBFGS	lbfgs	SNESLINESEARCHCP
"Good" Broyden	SNES_QN_BROYDEN	broyden	SNESLINESEARCHBASIC
"Bad" Broyden	SNES_QN_BADBROYEN	badbroyden	SNESLINESEARCHL2

Table 8: PETSc quasi-Newton solvers

SNESQNSetScaleType(SNES snes, SNESQNScaleType stype);

and the restart type with

SNESQNSetRestartType(SNES snes, SNESQNRestartType rtype);.

5.2.5 The Full Approximation Scheme

The Full Approximation Scheme is a nonlinear multigrid correction. At each level, there is a recursive cycle control SNES instance, and either one or two nonlinear solvers as smoothers (up and down). Problems set up using the SNES DMDA interface are automatically coarsened. FAS differs slightly from PCMG, in that the hierarchy is constructed recursively. However, much of the interface is a one-to-one map. We describe the "get" operations here, and it can be assumed that each has a corresponding "set" operation. For instance, the number of levels in the hierarchy may be retrieved using

SNESFASGetLevels(SNES snes, PetscInt *levels);

There are four SNESFAS cycle types, SNES_FAS_MULTIPLICATIVE, SNES_FAS_ADDITIVE, SNES_FAS_FULL, or SNES_FAS_KASKADE, The type may be set with

SNESFASSetType(SNES snes,SNESFASType fastype);.

and the cycle type, 1 for V, 2 for W, may be set with

SNESFASSetCycles(SNES snes, PetscInt cycles);.

Much like the interface to PCMG described in Section 4.4.9, there are interfaces to recover the various levels' cycles and smoothers. The level smoothers may be accessed with

SNESFASGetSmoother(SNES snes, PetscInt level, SNES *smooth);
SNESFASGetSmootherUp(SNES snes, PetscInt level, SNES *smooth);
SNESFASGetSmootherDown(SNES snes, PetscInt level, SNES *smooth);

and the level cycles with

SNESFASGetCycleSNES(SNES snes,PetscInt level,SNES *lsnes);.

Much like PCMG, the restriction and prolongation at a level may be acquired with

SNESFASGetInterpolation(SNES snes, PetscInt level, Mat *mat); SNESFASGetRestriction(SNES snes, PetscInt level, Mat

In addition, FAS requires special restriction for solution-like variables, called injection. This may be set with

SNESFASGetInjection(SNES snes, PetscInt level, Mat *mat);.

The coarse solve context may be acquired with

SNESFASGetCoarseSolve(SNES snes, SNES *smooth);

5.2.6 Nonlinear Additive Schwarz

Nonlinear Additive Schwarz methods (NASM) take a number of local nonlinear subproblems, solves them independently in parallel, and combines those solutions into a new approximate solution.

SNESNASMSetSubdomains(SNES snes,PetscInt n,SNES subsnes[],VecScatter iscatter[],VecScatter oscatter[],VecScatter gs

allows for the user to create these local subdomains. Problems set up using the SNES DMDA interface are automatically decomposed. To begin, the type of subdomain updates to the whole solution are limited to two types borrowed from PCASM: `PC_ASM_BASIC`, in which the overlapping updates added. `PC_ASM_RESTRICT` updates in a nonoverlapping fashion. This may be set with

SNESNASMSetType(SNES snes,PCASMType type);.

SNESASPIN is a helper SNES type that sets up a nonlinearly preconditioned Newton's method using NASM as the preconditioner.

5.3 General Options

This section discusses options and routines that apply to all SNES solvers and problem classes. In particular, we focus on convergence tests, monitoring routines, and tools for checking derivative computations.

5.3.1 Convergence Tests

Convergence of the nonlinear solvers can be detected in a variety of ways; the user can even specify a customized test, as discussed below. Most of the nonlinear solvers use SNESConvergenceTestDefault(), however, SNESNEWTONTR uses a method-specific additional convergence test as well. The convergence tests involves several parameters, which are set by default to values that should be reasonable for a wide range of problems. The user can customize the parameters to the problem at hand by using some of the following routines and options.

One method of convergence testing is to declare convergence when the norm of the change in the solution between successive iterations is less than some tolerance, `stol`. Convergence can also be determined based on the norm of the function Such a test can use either the absolute size of the norm, `atol`, or its relative decrease, `rtol`, from an initial guess. The following routine sets these parameters, which are used in many of the default SNES convergence tests:

SNESSetTolerances(SNES snes,double atol,double rtol,double stol,
int its,int fcts);

This routine also sets the maximum numbers of allowable nonlinear iterations, `its`, and function evaluations, `fcts`. The corresponding options database commands for setting these parameters are `-snes_atol <atol>`, `-snes_rtol <rtol>`, `-snes_stol <stol>`, `-snes_max_it <its>`, and `-snes_max_funcs <fcts>`. A related routine is SNESGetTolerances().

Convergence tests for trust regions methods often use an additional parameter that indicates the minimum allowable trust region radius. The user can set this parameter with the option `-snes_trtol <trtol>` or with the routine

SNESSetTrustRegionTolerance(SNES snes,double trtol);

Users can set their own customized convergence tests in SNES by using the command

SNESSetConvergenceTest(SNES snes,PetscErrorCode (*test)(SNES snes,int it,double xnorm,
double gnorm,double f,SNESConvergedReason reason,
void *cctx),void *cctx,PetscErrorCode (*destroy)(void *cctx));

The final argument of the convergence test routine, `cctx`, denotes an optional user-defined context for private data. When solving systems of nonlinear equations, the arguments `xnorm`, `gnorm`, and `f` are the current iterate norm, current step norm, and function norm, respectively. SNESConvergedReason should be set positive for convergence and negative for divergence. See `include/petscsnes.h` for a list of values for SNESConvergedReason.

5.3.2 Convergence Monitoring

By default the SNES solvers run silently without displaying information about the iterations. The user can initiate monitoring with the command

```
SNESMonitorSet(SNES snes,PetscErrorCode (*mon)(SNES,int its,double norm,void* mctx),
               void *mctx,PetscErrorCode (*monitordestroy)(void**));
```

The routine, `mon`, indicates a user-defined monitoring routine, where `its` and `mctx` respectively denote the iteration number and an optional user-defined context for private data for the monitor routine. The argument `norm` is the function norm.

The routine set by SNESMonitorSet() is called once after every successful step computation within the nonlinear solver. Hence, the user can employ this routine for any application-specific computations that should be done after the solution update. The option `-snes_monitor` activates the default SNES monitor routine, SNESMonitorDefault(), while `-snes_monitor_lg_residualnorm` draws a simple line graph of the residual norm's convergence.

Once can cancel hardwired monitoring routines for SNES at runtime with `-snes_monitor_cancel`.

As the Newton method converges so that the residual norm is small, say 10^{-10}, many of the final digits printed with the `-snes_monitor` option are meaningless. Worse, they are different on different machines; due to different round-off rules used by, say, the IBM RS6000 and the Sun SPARC. This makes testing between different machines difficult. The option `-snes_monitor_short` causes PETSc to print fewer of the digits of the residual norm as it gets smaller; thus on most of the machines it will always print the same numbers making cross process testing easier.

The routines

```
SNESGetSolution(SNES snes,Vec *x);
SNESGetFunction(SNES snes,Vec *r,void *ctx,
                int(**func)(SNES,Vec,Vec,void*));
```

return the solution vector and function vector from a SNES context. These routines are useful, for instance, if the convergence test requires some property of the solution or function other than those passed with routine arguments.

5.3.3 Checking Accuracy of Derivatives

Since hand-coding routines for Jacobian matrix evaluation can be error prone, SNES provides easy-to-use support for checking these matrices against finite difference versions. In the simplest form of comparison, users can employ the option `-snes_type test` to compare the matrices at several points. Although not exhaustive, this test will generally catch obvious problems. One can compare the elements of the two matrices by using the option `-snes_test_display`, which causes the two matrices to be printed to the screen.

Another means for verifying the correctness of a code for Jacobian computation is running the problem with either the finite difference or matrix-free variant, `-snes_fd` or `-snes_mf`. see Section 5.6 or Section 5.5). If a problem converges well with these matrix approximations but not with a user-provided routine, the problem probably lies with the hand-coded matrix.

5.4 Inexact Newton-like Methods

Since exact solution of the linear Newton systems within (5.2) at each iteration can be costly, modifications are often introduced that significantly reduce these expenses and yet retain the rapid convergence of Newton's method. Inexact or truncated Newton techniques approximately solve the linear systems using an iterative scheme. In comparison with using direct methods for solving the Newton systems, iterative methods have the virtue of requiring little space for matrix storage and potentially saving significant computational work. Within the class of inexact Newton methods, of particular interest are Newton-Krylov methods, where the subsidiary iterative technique for solving the Newton system is chosen from the class of Krylov subspace projection methods. Note that at runtime the user can set any of the linear solver options discussed in Chapter 4, such as `-ksp_type <ksp_method>` and `-pc_type <pc_method>`, to set the Krylov subspace and preconditioner methods.

Two levels of iterations occur for the inexact techniques, where during each global or outer Newton iteration a sequence of subsidiary inner iterations of a linear solver is performed. Appropriate control of the accuracy to which the subsidiary iterative method solves the Newton system at each global iteration is critical, since these inner iterations determine the asymptotic convergence rate for inexact Newton techniques. While the Newton systems must be solved well enough to retain fast local convergence of the Newton's iterates, use of excessive inner iterations, particularly when $\|\boldsymbol{x}_k - \boldsymbol{x}_*\|$ is large, is neither necessary nor economical. Thus, the number of required inner iterations typically increases as the Newton process progresses, so that the truncated iterates approach the true Newton iterates.

A sequence of nonnegative numbers $\{\eta_k\}$ can be used to indicate the variable convergence criterion. In this case, when solving a system of nonlinear equations, the update step of the Newton process remains unchanged, and direct solution of the linear system is replaced by iteration on the system until the residuals

$$\boldsymbol{r}_k^{(i)} = \boldsymbol{F}'(\boldsymbol{x}_k)\Delta\boldsymbol{x}_k + \boldsymbol{F}(\boldsymbol{x}_k)$$

satisfy

$$\frac{\|\boldsymbol{r}_k^{(i)}\|}{\|\boldsymbol{F}(\boldsymbol{x}_k)\|} \leq \eta_k \leq \eta < 1.$$

Here $\boldsymbol{x}_0$ is an initial approximation of the solution, and $\|\cdot\|$ denotes an arbitrary norm in $\Re^n$.

By default a constant relative convergence tolerance is used for solving the subsidiary linear systems within the Newton-like methods of SNES. When solving a system of nonlinear equations, one can instead employ the techniques of Eisenstat and Walker [10] to compute η_k at each step of the nonlinear solver by using the option `-snes_ksp_ew_conv` . In addition, by adding one's own KSP convergence test (see Section 4.3.2), one can easily create one's own, problem-dependent, inner convergence tests.

5.5 Matrix-Free Methods

The SNES class fully supports matrix-free methods. The matrices specified in the Jacobian evaluation routine need not be conventional matrices; instead, they can point to the data required to implement a particular matrix-free method. The matrix-free variant is allowed *only* when the linear systems are solved by an iterative method in combination with no preconditioning (PCNONE or `-pc_type none`), a user-provided preconditioner matrix, or a user-provided preconditioner shell (PCSHELL, discussed in Section 4.4); that is, obviously matrix-free methods cannot be used if a direct solver is to be employed.

The user can create a matrix-free context for use within SNES with the routine

MatCreateSNESMF(SNES snes,Mat *mat);

This routine creates the data structures needed for the matrix-vector products that arise within Krylov space iterative methods [4] by employing the matrix type MATSHELL, discussed in Section 3.3. The default SNES matrix-free approximations can also be invoked with the command `-snes_mf`. Or, one can retain the user-provided Jacobian preconditioner, but replace the user-provided Jacobian matrix with the default matrix free variant with the option `-snes_mf_operator`.

See also

MatCreateMFFD(Vec x, Mat *mat);

for users who need a matrix-free matrix but are not using SNES.

The user can set one parameter to control the Jacobian-vector product approximation with the command

MatMFFDSetFunctionError(Mat mat,double rerror);

The parameter `rerror` should be set to the square root of the relative error in the function evaluations, e_{rel}; the default is 10^{-8}, which assumes that the functions are evaluated to full double precision accuracy. This parameter can also be set from the options database with

-snes_mf_err `<err>`

In addition, SNES provides a way to register new routines to compute the differencing parameter (h); see the manual page for MatMFFDSetType() and MatMFFDRegister(). We currently provide two default routines accessible via

-snes_mf_type `<default or wp>`

For the default approach there is one "tuning" parameter, set with

MatMFFDDSSetUmin(Mat mat,PetscReal umin);

This parameter, `umin` (or u_{min}), is a bit involved; its default is 10^{-6} . The Jacobian-vector product is approximated via the formula

$$F'(u)a \approx \frac{F(u + h * a) - F(u)}{h}$$

where h is computed via

$$\begin{aligned} h &= e_{rel} * u^T a / ||a||_2^2 \quad \text{if} |u'a| > u_{min} * ||a||_1 \\ &= e_{rel} * u_{min} * sign(u^T a) * ||a||_1 / ||a||_2^2 \quad \text{otherwise.} \end{aligned}$$

This approach is taken from Brown and Saad [4]. The parameter can also be set from the options database with

-snes_mf_umin `<umin>`

The second approach, taken from Walker and Pernice, [24], computes h via

$$h = \frac{\sqrt{1 + ||u||} e_{rel}}{||a||}$$

This has no tunable parameters, but note that inside the nonlinear solve for the entire **linear** iterative process u does not change hence $\sqrt{1 + ||u||}$ need be computed only once. This information may be set with the options

MatMFFDWPSetComputeNormU(Mat mat,PetscBool);

or

-mat_mffd_compute_normu <true or false>

This information is used to eliminate the redundant computation of these parameters, therefor reducing the number of collective operations and improving the efficiency of the application code.

It is also possible to monitor the differencing parameters h that are computed via the routines

MatMFFDSetHHistory(Mat,PetscScalar *,int);
MatMFFDResetHHistory(Mat,PetscScalar *,int);
MatMFFDGetH(Mat,PetscScalar *);

We include an example in Figure 14 that explicitly uses a matrix-free approach. Note that by using the option -snes_mf one can easily convert any SNES code to use a matrix-free Newton-Krylov method without a preconditioner. As shown in this example, SNESSetFromOptions() must be called *after* SNESSetJacobian() to enable runtime switching between the user-specified Jacobian and the default SNES matrix-free form.

Table 9 summarizes the various matrix situations that SNES supports. In particular, different linear system matrices and preconditioning matrices are allowed, as well as both matrix-free and application-provided preconditioners. If ${PETSC_DIR}/src/snes/examples/tutorials/ex3.c, listed in Figure 14, is run with the options -snes_mf and -user_precond then it uses a matrix-free application of the matrix-vector multiple and a user provided matrix free Jacobian.

Matrix Use	**Conventional Matrix Formats**	**Matrix-Free Versions**
Jacobian Matrix	Create matrix with MatCreate(). * Assemble matrix with user-defined routine. †	Create matrix with MatCreateShell(). Use MatShellSetOperation() to set various matrix actions. Or use MatCreateMFFD() or MatCreateSNESMF().
Preconditioning Matrix	Create matrix with MatCreate(). * Assemble matrix with user-defined routine. †	Use SNESGetKSP() and KSPGetPC() to access the PC, then use PCSetType(pc,PCSHELL); followed by PCShellSetApply().

* Use either the generic MatCreate() or a format-specific variant such as MatCreateAIJ().
† Set user-defined matrix formation routine with SNESSetJacobian().

Table 9: Jacobian Options

```
static char help[] = "Newton methods to solve u'' + u^{2} = f in parallel.\n\
This example employs a user-defined monitoring routine and optionally a user-defined\n\
```

```
routine to check candidate iterates produced by line search routines.  This
code also\n\
demonstrates use of the macro __FUNCT__ to define routine names for use in
error handling.\n\
The command line options include:\n\
  -pre_check_iterates : activate checking of iterates\n\
  -post_check_iterates : activate checking of iterates\n\
  -check_tol <tol>: set tolerance for iterate checking\n\n";

/*T
   Concepts: SNESˆbasic parallel example
   Concepts: SNESˆsetting a user-defined monitoring routine
   Concepts: error handlingˆusing the macro __FUNCT__ to define routine names;
   Processors: n
T*/

/*
   Include "petscdraw.h" so that we can use distributed arrays (DMDAs).
   Include "petscdraw.h" so that we can use PETSc drawing routines.
   Include "petscsnes.h" so that we can use SNES solvers.  Note that this
   file automatically includes:
     petscsys.h       - base PETSc routines   petscvec.h - vectors
     petscmat.h - matrices
     petscis.h     - index sets            petscksp.h - Krylov subspace methods
     petscviewer.h - viewers               petscpc.h  - preconditioners
     petscksp.h   - linear solvers
*/

#include <petscdm.h>
#include <petscdmda.h>
#include <petscsnes.h>

/*
   User-defined routines.  Note that immediately before each routine below,
   we define the macro __FUNCT__ to be a string containing the routine name.
   If defined, this macro is used in the PETSc error handlers to provide a
   complete traceback of routine names.  All PETSc library routines use this
   macro, and users can optionally employ it as well in their application
   codes.  Note that users can get a traceback of PETSc errors regardless
of
   whether they define __FUNCT__ in application codes; this macro merely
   provides the added traceback detail of the application routine names.
*/
PetscErrorCode FormJacobian(SNES,Vec,Mat,Mat,void*);
PetscErrorCode FormFunction(SNES,Vec,Vec,void*);
PetscErrorCode FormInitialGuess(Vec);
PetscErrorCode Monitor(SNES,PetscInt,PetscReal,void*);
PetscErrorCode PreCheck(SNESLineSearch,Vec,Vec,PetscBool*,void*);
PetscErrorCode PostCheck(SNESLineSearch,Vec,Vec,Vec,PetscBool*,PetscBool*,void*);
PetscErrorCode PostSetSubKSP(SNESLineSearch,Vec,Vec,Vec,PetscBool*,PetscBool*,void*);
PetscErrorCode MatrixFreePreconditioner(PC,Vec,Vec);

/*
   User-defined application context
*/
```

```
typedef struct {
  DM          da;      /* distributed array */
  Vec         F;       /* right-hand-side of PDE */
  PetscMPIInt rank;    /* rank of processor */
  PetscMPIInt size;    /* size of communicator */
  PetscReal   h;       /* mesh spacing */
} ApplicationCtx;

/*
   User-defined context for monitoring
*/
typedef struct {
  PetscViewer viewer;
} MonitorCtx;

/*
   User-defined context for checking candidate iterates that are
   determined by line search methods
*/
typedef struct {
  Vec            last_step;  /* previous iterate */
  PetscReal      tolerance;  /* tolerance for changes between successive iterates
*/
  ApplicationCtx *user;
} StepCheckCtx;

typedef struct {
  PetscInt its0; /* num of prevous outer KSP iterations */
} SetSubKSPCtx;

#undef __FUNCT__
#define __FUNCT__ "main"
int main(int argc,char **argv)
{
  SNES           snes;                 /* SNES context */
  SNESLineSearch linesearch;          /* SNESLineSearch context */
  Mat            J;                    /* Jacobian matrix */
  ApplicationCtx ctx;                  /* user-defined context */
  Vec            x,r,U,F;              /* vectors */
  MonitorCtx     monP;                 /* monitoring context */
  StepCheckCtx   checkP;               /* step-checking context */
  SetSubKSPCtx   checkP1;
  PetscBool      pre_check,post_check,post_setsubksp; /* flag indicating whether
we're checking candidate iterates */
  PetscScalar    xp,*FF,*UU,none = -1.0;
  PetscErrorCode ierr;
  PetscInt       its,N = 5,i,maxit,maxf,xs,xm;
  PetscReal      abstol,rtol,stol,norm;
  PetscBool      flg;


  PetscInitialize(&argc,&argv,(char*)0,help);
  ierr  = MPI_Comm_rank(PETSC_COMM_WORLD,&ctx.rank);CHKERRQ(ierr);
  ierr  = MPI_Comm_size(PETSC_COMM_WORLD,&ctx.size);CHKERRQ(ierr);
  ierr  = PetscOptionsGetInt(NULL,NULL,"-n",&N,NULL);CHKERRQ(ierr);
```

```
  ctx.h = 1.0/(N-1);

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Create nonlinear solver context
     - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

  ierr = SNESCreate(PETSC_COMM_WORLD,&snes);CHKERRQ(ierr);

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Create vector data structures; set function evaluation routine
     - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
*/

  /*
     Create distributed array (DMDA) to manage parallel grid and vectors
  */
  ierr = DMDACreate1d(PETSC_COMM_WORLD,DM_BOUNDARY_NONE,N,1,1,NULL,&ctx.da);CHKERRQ(ierr);

  /*
     Extract global and local vectors from DMDA; then duplicate for remaining
     vectors that are the same types
  */
  ierr = DMCreateGlobalVector(ctx.da,&x);CHKERRQ(ierr);
  ierr = VecDuplicate(x,&r);CHKERRQ(ierr);
  ierr = VecDuplicate(x,&F);CHKERRQ(ierr); ctx.F = F;
  ierr = VecDuplicate(x,&U);CHKERRQ(ierr);

  /*
     Set function evaluation routine and vector.  Whenever the nonlinear
     solver needs to compute the nonlinear function, it will call this
     routine.
      - Note that the final routine argument is the user-defined
        context that provides application-specific data for the
        function evaluation routine.
  */
  ierr = SNESSetFunction(snes,r,FormFunction,&ctx);CHKERRQ(ierr);

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Create matrix data structure; set Jacobian evaluation routine
     - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
*/

  ierr = MatCreate(PETSC_COMM_WORLD,&J);CHKERRQ(ierr);
  ierr = MatSetSizes(J,PETSC_DECIDE,PETSC_DECIDE,N,N);CHKERRQ(ierr);
  ierr = MatSetFromOptions(J);CHKERRQ(ierr);
  ierr = MatSeqAIJSetPreallocation(J,3,NULL);CHKERRQ(ierr);
  ierr = MatMPIAIJSetPreallocation(J,3,NULL,3,NULL);CHKERRQ(ierr);

  /*
     Set Jacobian matrix data structure and default Jacobian evaluation
     routine.  Whenever the nonlinear solver needs to compute the
     Jacobian matrix, it will call this routine.
      - Note that the final routine argument is the user-defined
        context that provides application-specific data for the
        Jacobian evaluation routine.
```

```
*/
ierr = SNESSetJacobian(snes,J,J,FormJacobian,&ctx);CHKERRQ(ierr);

/*
   Optional allow user provided preconditioner
 */
ierr = PetscOptionsHasName(NULL,NULL,"-user_precond",&flg);CHKERRQ(ierr);
if (flg) {
  KSP ksp;
  PC  pc;
  ierr = SNESGetKSP(snes,&ksp);CHKERRQ(ierr);
  ierr = KSPGetPC(ksp,&pc);CHKERRQ(ierr);
  ierr = PCSetType(pc,PCSHELL);CHKERRQ(ierr);
  ierr = PCShellSetApply(pc,MatrixFreePreconditioner);CHKERRQ(ierr);
}

/* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
   Customize nonlinear solver; set runtime options
 - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

/*
   Set an optional user-defined monitoring routine
*/
ierr = PetscViewerDrawOpen(PETSC_COMM_WORLD,0,0,0,0,400,400,&monP.viewer);CHKERRQ(ierr);
ierr = SNESMonitorSet(snes,Monitor,&monP,0);CHKERRQ(ierr);

/*
   Set names for some vectors to facilitate monitoring (optional)
*/
ierr = PetscObjectSetName((PetscObject)x,"Approximate Solution");CHKERRQ(ierr);
ierr = PetscObjectSetName((PetscObject)U,"Exact Solution");CHKERRQ(ierr);

/*
   Set SNES/KSP/KSP/PC runtime options, e.g.,
       -snes_view -snes_monitor -ksp_type <ksp> -pc_type <pc>
*/
ierr = SNESSetFromOptions(snes);CHKERRQ(ierr);

/*
   Set an optional user-defined routine to check the validity of candidate
   iterates that are determined by line search methods
*/
ierr = SNESGetLineSearch(snes, &linesearch);CHKERRQ(ierr);
ierr = PetscOptionsHasName(NULL,NULL,"-post_check_iterates",&post_check);CHKERRQ(ierr);

if (post_check) {
  ierr = PetscPrintf(PETSC_COMM_WORLD,"Activating post step checking routine\n");CHKERRQ
  ierr = SNESLineSearchSetPostCheck(linesearch,PostCheck,&checkP);CHKERRQ(ierr);
  ierr = VecDuplicate(x,&(checkP.last_step));CHKERRQ(ierr);

  checkP.tolerance = 1.0;
  checkP.user      = &ctx;

  ierr = PetscOptionsGetReal(NULL,NULL,"-check_tol",&checkP.tolerance,NULL);CHKERRQ(ierr)
}
```

```
  ierr = PetscOptionsHasName(NULL,NULL,"-post_setsubksp",&post_setsubksp);CHKERRQ(ierr);
  if (post_setsubksp) {
    ierr = PetscPrintf(PETSC_COMM_WORLD,"Activating post setsubksp\n");CHKERRQ(ierr);
    ierr = SNESLineSearchSetPostCheck(linesearch,PostSetSubKSP,&checkP1);CHKERRQ(ierr);
  }

  ierr = PetscOptionsHasName(NULL,NULL,"-pre_check_iterates",&pre_check);CHKERRQ(ierr);
  if (pre_check) {
    ierr = PetscPrintf(PETSC_COMM_WORLD,"Activating pre step checking routine\n");CHKERRQ(i
    ierr = SNESLineSearchSetPreCheck(linesearch,PreCheck,&checkP);CHKERRQ(ierr);
  }

  /*
     Print parameters used for convergence testing (optional) ... just
     to demonstrate this routine; this information is also printed with
     the option -snes_view
  */
  ierr = SNESGetTolerances(snes,&abstol,&rtol,&stol,&maxit,&maxf);CHKERRQ(ierr);
  ierr = PetscPrintf(PETSC_COMM_WORLD,"atol=%g, rtol=%g, stol=%g, maxit=%D,
maxf=%D\n",(double)abstol,(double)rtol,(double)stol,maxit,maxf);CHKERRQ(ierr);

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Initialize application:
     Store right-hand-side of PDE and exact solution
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

  /*
     Get local grid boundaries (for 1-dimensional DMDA):
       xs, xm - starting grid index, width of local grid (no ghost points)
  */
  ierr = DMDAGetCorners(ctx.da,&xs,NULL,NULL,&xm,NULL,NULL);CHKERRQ(ierr);

  /*
     Get pointers to vector data
  */
  ierr = DMDAVecGetArray(ctx.da,F,&FF);CHKERRQ(ierr);
  ierr = DMDAVecGetArray(ctx.da,U,&UU);CHKERRQ(ierr);

  /*
     Compute local vector entries
  */
  xp = ctx.h*xs;
  for (i=xs; i<xs+xm; i++) {
    FF[i] = 6.0*xp + PetscPowScalar(xp+1.e-12,6.0); /* +1.e-12 is to prevent
0^6 */
    UU[i] = xp*xp*xp;
    xp   += ctx.h;
  }

  /*
     Restore vectors
  */
  ierr = DMDAVecRestoreArray(ctx.da,F,&FF);CHKERRQ(ierr);
  ierr = DMDAVecRestoreArray(ctx.da,U,&UU);CHKERRQ(ierr);
```

```
  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Evaluate initial guess; then solve nonlinear system
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

  /*
     Note: The user should initialize the vector, x, with the initial guess
     for the nonlinear solver prior to calling SNESSolve().  In particular,
     to employ an initial guess of zero, the user should explicitly set
     this vector to zero by calling VecSet().
  */
  ierr = FormInitialGuess(x);CHKERRQ(ierr);
  ierr = SNESSolve(snes,NULL,x);CHKERRQ(ierr);
  ierr = SNESGetIterationNumber(snes,&its);CHKERRQ(ierr);
  ierr = PetscPrintf(PETSC_COMM_WORLD,"Number of SNES iterations = %D\n",its);CHKERRQ(ierr)

  /* - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
     Check solution and clean up
   - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - */

  /*
     Check the error
  */
  ierr = VecAXPY(x,none,U);CHKERRQ(ierr);
  ierr = VecNorm(x,NORM_2,&norm);CHKERRQ(ierr);
  ierr = PetscPrintf(PETSC_COMM_WORLD,"Norm of error %g Iterations %D\n",(double)norm,its);

  /*
     Free work space.  All PETSc objects should be destroyed when they
     are no longer needed.
  */
  ierr = PetscViewerDestroy(&monP.viewer);CHKERRQ(ierr);
  if (post_check) {ierr = VecDestroy(&checkP.last_step);CHKERRQ(ierr);}
  ierr = VecDestroy(&x);CHKERRQ(ierr);
  ierr = VecDestroy(&r);CHKERRQ(ierr);
  ierr = VecDestroy(&U);CHKERRQ(ierr);
  ierr = VecDestroy(&F);CHKERRQ(ierr);
  ierr = MatDestroy(&J);CHKERRQ(ierr);
  ierr = SNESDestroy(&snes);CHKERRQ(ierr);
  ierr = DMDestroy(&ctx.da);CHKERRQ(ierr);
  ierr = PetscFinalize();
  PetscFunctionReturn(0);
}
/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "FormInitialGuess"
/*
   FormInitialGuess - Computes initial guess.

   Input/Output Parameter:
.  x - the solution vector
*/
PetscErrorCode FormInitialGuess(Vec x)
{
  PetscErrorCode ierr;
```

```
  PetscScalar    pfive = .50;

  PetscFunctionBeginUser;
  ierr = VecSet(x,pfive);CHKERRQ(ierr);
  PetscFunctionReturn(0);
}
/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "FormFunction"
/*
   FormFunction - Evaluates nonlinear function, F(x).

   Input Parameters:
.  snes - the SNES context
.  x - input vector
.  ctx - optional user-defined context, as set by SNESSetFunction()

   Output Parameter:
.  f - function vector

   Note:
   The user-defined context can contain any application-specific
   data needed for the function evaluation.
*/
PetscErrorCode FormFunction(SNES snes,Vec x,Vec f,void *ctx)
{
  ApplicationCtx *user = (ApplicationCtx*) ctx;
  DM             da    = user->da;
  PetscScalar    *xx,*ff,*FF,d;
  PetscErrorCode ierr;
  PetscInt       i,M,xs,xm;
  Vec            xlocal;

  PetscFunctionBeginUser;
  ierr = DMGetLocalVector(da,&xlocal);CHKERRQ(ierr);
  /*
     Scatter ghost points to local vector, using the 2-step process
        DMGlobalToLocalBegin(), DMGlobalToLocalEnd().
     By placing code between these two statements, computations can
     be done while messages are in transition.
  */
  ierr = DMGlobalToLocalBegin(da,x,INSERT_VALUES,xlocal);CHKERRQ(ierr);
  ierr = DMGlobalToLocalEnd(da,x,INSERT_VALUES,xlocal);CHKERRQ(ierr);

  /*
     Get pointers to vector data.
       - The vector xlocal includes ghost point; the vectors x and f do
         NOT include ghost points.
       - Using DMDAVecGetArray() allows accessing the values using global
ordering
  */
  ierr = DMDAVecGetArray(da,xlocal,&xx);CHKERRQ(ierr);
  ierr = DMDAVecGetArray(da,f,&ff);CHKERRQ(ierr);
  ierr = DMDAVecGetArray(da,user->F,&FF);CHKERRQ(ierr);
```

```
  /*
     Get local grid boundaries (for 1-dimensional DMDA):
       xs, xm  - starting grid index, width of local grid (no ghost points)
  */
  ierr = DMDAGetCorners(da,&xs,NULL,NULL,&xm,NULL,NULL);CHKERRQ(ierr);
  ierr = DMDAGetInfo(da,NULL,&M,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL);CH

  /*
     Set function values for boundary points; define local interior grid point
range:
        xsi - starting interior grid index
        xei - ending interior grid index
  */
  if (xs == 0) { /* left boundary */
    ff[0] = xx[0];
    xs++;xm--;
  }
  if (xs+xm == M) {  /* right boundary */
    ff[xs+xm-1] = xx[xs+xm-1] - 1.0;
    xm--;
  }

  /*
     Compute function over locally owned part of the grid (interior points
only)
  */
  d = 1.0/(user->h*user->h);
  for (i=xs; i<xs+xm; i++) ff[i] = d*(xx[i-1] - 2.0*xx[i] + xx[i+1]) + xx[i]*xx[i]
- FF[i];

  /*
     Restore vectors
  */
  ierr = DMDAVecRestoreArray(da,xlocal,&xx);CHKERRQ(ierr);
  ierr = DMDAVecRestoreArray(da,f,&ff);CHKERRQ(ierr);
  ierr = DMDAVecRestoreArray(da,user->F,&FF);CHKERRQ(ierr);
  ierr = DMRestoreLocalVector(da,&xlocal);CHKERRQ(ierr);
  PetscFunctionReturn(0);
}
/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "FormJacobian"
/*
   FormJacobian - Evaluates Jacobian matrix.

   Input Parameters:
.  snes - the SNES context
.  x - input vector
.  dummy - optional user-defined context (not used here)

   Output Parameters:
.  jac - Jacobian matrix
.  B - optionally different preconditioning matrix
.  flag - flag indicating matrix structure
*/
```

```
PetscErrorCode FormJacobian(SNES snes,Vec x,Mat jac,Mat B,void *ctx)
{
  ApplicationCtx *user = (ApplicationCtx*) ctx;
  PetscScalar    *xx,d,A[3];
  PetscErrorCode ierr;
  PetscInt       i,j[3],M,xs,xm;
  DM             da = user->da;

  PetscFunctionBeginUser;
  /*
     Get pointer to vector data
  */
  ierr = DMDAVecGetArrayRead(da,x,&xx);CHKERRQ(ierr);
  ierr = DMDAGetCorners(da,&xs,NULL,NULL,&xm,NULL,NULL);CHKERRQ(ierr);

  /*
    Get range of locally owned matrix
  */
  ierr = DMDAGetInfo(da,NULL,&M,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL,NULL);CH

  /*
     Determine starting and ending local indices for interior grid points.
     Set Jacobian entries for boundary points.
  */

  if (xs == 0) {  /* left boundary */
    i = 0; A[0] = 1.0;

    ierr = MatSetValues(jac,1,&i,1,&i,A,INSERT_VALUES);CHKERRQ(ierr);
    xs++;xm--;
  }
  if (xs+xm == M) { /* right boundary */
    i    = M-1;
    A[0] = 1.0;
    ierr = MatSetValues(jac,1,&i,1,&i,A,INSERT_VALUES);CHKERRQ(ierr);
    xm--;
  }

  /*
     Interior grid points
      - Note that in this case we set all elements for a particular
        row at once.
  */
  d = 1.0/(user->h*user->h);
  for (i=xs; i<xs+xm; i++) {
    j[0] = i - 1; j[1] = i; j[2] = i + 1;
    A[0] = A[2] = d; A[1] = -2.0*d + 2.0*xx[i];
    ierr = MatSetValues(jac,1,&i,3,j,A,INSERT_VALUES);CHKERRQ(ierr);
  }

  /*
     Assemble matrix, using the 2-step process:
       MatAssemblyBegin(), MatAssemblyEnd().
     By placing code between these two statements, computations can be
     done while messages are in transition.
```

```
     Also, restore vector.
  */

  ierr = MatAssemblyBegin(jac,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);
  ierr = DMDAVecRestoreArrayRead(da,x,&xx);CHKERRQ(ierr);
  ierr = MatAssemblyEnd(jac,MAT_FINAL_ASSEMBLY);CHKERRQ(ierr);

  PetscFunctionReturn(0);
}
/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "Monitor"
/*
   Monitor - Optional user-defined monitoring routine that views the
   current iterate with an x-window plot. Set by SNESMonitorSet().

   Input Parameters:
   snes - the SNES context
   its - iteration number
   norm - 2-norm function value (may be estimated)
   ctx - optional user-defined context for private data for the
         monitor routine, as set by SNESMonitorSet()

   Note:
   See the manpage for PetscViewerDrawOpen() for useful runtime options,
   such as -nox to deactivate all x-window output.
 */
PetscErrorCode Monitor(SNES snes,PetscInt its,PetscReal fnorm,void *ctx)
{
  PetscErrorCode ierr;
  MonitorCtx     *monP = (MonitorCtx*) ctx;
  Vec            x;

  PetscFunctionBeginUser;
  ierr = PetscPrintf(PETSC_COMM_WORLD,"iter = %D,SNES Function norm %g\n",its,(double)fnorm
  ierr = SNESGetSolution(snes,&x);CHKERRQ(ierr);
  ierr = VecView(x,monP->viewer);CHKERRQ(ierr);
  PetscFunctionReturn(0);
}

/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "PreCheck"
/*
   PreCheck - Optional user-defined routine that checks the validity of
   candidate steps of a line search method.  Set by SNESLineSearchSetPreCheck().

   Input Parameters:
   snes - the SNES context
   xcurrent - current solution
   y - search direction and length

   Output Parameters:
   y         - proposed step (search direction and length) (possibly changed)
```

```
   changed_y - tells if the step has changed or not
 */
PetscErrorCode PreCheck(SNESLineSearch linesearch,Vec xcurrent,Vec y, PetscBool
*changed_y, void * ctx)
{
  PetscFunctionBeginUser;
  *changed_y = PETSC_FALSE;
  PetscFunctionReturn(0);
}

/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "PostCheck"
/*
   PostCheck - Optional user-defined routine that checks the validity of
   candidate steps of a line search method.  Set by SNESLineSearchSetPostCheck().

   Input Parameters:
   snes - the SNES context
   ctx  - optional user-defined context for private data for the
          monitor routine, as set by SNESLineSearchSetPostCheck()
   xcurrent - current solution
   y - search direction and length
   x    - the new candidate iterate

   Output Parameters:
   y    - proposed step (search direction and length) (possibly changed)
   x    - current iterate (possibly modified)

 */
PetscErrorCode PostCheck(SNESLineSearch linesearch,Vec xcurrent,Vec y,Vec
x,PetscBool  *changed_y,PetscBool  *changed_x, void * ctx)
{
  PetscErrorCode ierr;
  PetscInt       i,iter,xs,xm;
  StepCheckCtx   *check;
  ApplicationCtx *user;
  PetscScalar    *xa,*xa_last,tmp;
  PetscReal      rdiff;
  DM             da;
  SNES           snes;

  PetscFunctionBeginUser;
  *changed_x = PETSC_FALSE;
  *changed_y = PETSC_FALSE;

  ierr  = SNESLineSearchGetSNES(linesearch, &snes);CHKERRQ(ierr);
  check = (StepCheckCtx*)ctx;
  user  = check->user;
  ierr  = SNESGetIterationNumber(snes,&iter);CHKERRQ(ierr);

  /* iteration 1 indicates we are working on the second iteration */
  if (iter > 0) {
    da   = user->da;
    ierr = PetscPrintf(PETSC_COMM_WORLD,"Checking candidate step at iteration
```

```
%D with tolerance %g\n",iter,(double)check->tolerance);CHKERRQ(ierr);

    /* Access local array data */
    ierr = DMDAVecGetArray(da,check->last_step,&xa_last);CHKERRQ(ierr);
    ierr = DMDAVecGetArray(da,x,&xa);CHKERRQ(ierr);
    ierr = DMDAGetCorners(da,&xs,NULL,NULL,&xm,NULL,NULL);CHKERRQ(ierr);

    /*
       If we fail the user-defined check for validity of the candidate iterate,
       then modify the iterate as we like.  (Note that the particular modification
       below is intended simply to demonstrate how to manipulate this data,
not
       as a meaningful or appropriate choice.)
    */
    for (i=xs; i<xs+xm; i++) {
      if (!PetscAbsScalar(xa[i])) rdiff = 2*check->tolerance;
      else rdiff = PetscAbsScalar((xa[i] - xa_last[i])/xa[i]);
      if (rdiff > check->tolerance) {
        tmp        = xa[i];
        xa[i]      = .5*(xa[i] + xa_last[i]);
        *changed_x = PETSC_TRUE;
        ierr       = PetscPrintf(PETSC_COMM_WORLD,"  Altering entry %D: x=%g,
x_last=%g, diff=%g, x_new=%g\n",
                                 i,(double)PetscAbsScalar(tmp),(double)PetscAbsScalar(xa_la
      }
    }
    ierr = DMDAVecRestoreArray(da,check->last_step,&xa_last);CHKERRQ(ierr);
    ierr = DMDAVecRestoreArray(da,x,&xa);CHKERRQ(ierr);
  }
  ierr = VecCopy(x,check->last_step);CHKERRQ(ierr);
  PetscFunctionReturn(0);
}


/* ------------------------------------------------------------------- */
#undef __FUNCT__
#define __FUNCT__ "PostSetSubKSP"
/*
   PostSetSubKSP - Optional user-defined routine that reset SubKSP options
when hierarchical bjacobi PC is used
   e.g,
     mpiexec -n 8 ./ex3 -nox -n 10000 -ksp_type fgmres -pc_type bjacobi -pc_bjacobi_blocks
4 -sub_ksp_type gmres -sub_ksp_max_it 3 -post_setsubksp -sub_ksp_rtol 1.e-16
   Set by SNESLineSearchSetPostCheck().

   Input Parameters:
   linesearch - the LineSearch context
   xcurrent - current solution
   y - search direction and length
   x    - the new candidate iterate

   Output Parameters:
   y    - proposed step (search direction and length) (possibly changed)
   x    - current iterate (possibly modified)
```

```
 */
PetscErrorCode PostSetSubKSP(SNESLineSearch linesearch,Vec xcurrent,Vec y,Vec
x,PetscBool  *changed_y,PetscBool  *changed_x, void * ctx)
{
  PetscErrorCode ierr;
  SetSubKSPCtx   *check;
  PetscInt       iter,its,sub_its,maxit;
  KSP            ksp,sub_ksp,*sub_ksps;
  PC             pc;
  PetscReal      ksp_ratio;
  SNES           snes;

  PetscFunctionBeginUser;
  ierr    = SNESLineSearchGetSNES(linesearch, &snes);CHKERRQ(ierr);
  check   = (SetSubKSPCtx*)ctx;
  ierr    = SNESGetIterationNumber(snes,&iter);CHKERRQ(ierr);
  ierr    = SNESGetKSP(snes,&ksp);CHKERRQ(ierr);
  ierr    = KSPGetPC(ksp,&pc);CHKERRQ(ierr);
  ierr    = PCBJacobiGetSubKSP(pc,NULL,NULL,&sub_ksps);CHKERRQ(ierr);
  sub_ksp = sub_ksps[0];
  ierr    = KSPGetIterationNumber(ksp,&its);CHKERRQ(ierr);      /* outer KSP
iteration number */
  ierr    = KSPGetIterationNumber(sub_ksp,&sub_its);CHKERRQ(ierr); /* inner
KSP iteration number */

  if (iter) {
    ierr      = PetscPrintf(PETSC_COMM_WORLD,"    ...PostCheck snes iteration
%D, ksp_it %d %d, subksp_it %d\n",iter,check->its0,its,sub_its);CHKERRQ(ierr);
    ksp_ratio = ((PetscReal)(its))/check->its0;
    maxit     = (PetscInt)(ksp_ratio*sub_its + 0.5);
    if (maxit < 2) maxit = 2;
    ierr = KSPSetTolerances(sub_ksp,PETSC_DEFAULT,PETSC_DEFAULT,PETSC_DEFAULT,maxit);CHKERR
    ierr = PetscPrintf(PETSC_COMM_WORLD,"    ...ksp_ratio %g, new maxit %d\n\n",ksp_ratio,m
  }
  check->its0 = its; /* save current outer KSP iteration number */
  PetscFunctionReturn(0);
}

/* -------------------------------------------------------------------- */
/*
   MatrixFreePreconditioner - This routine demonstrates the use of a
   user-provided preconditioner.  This code implements just the null
   preconditioner, which of course is not recommended for general use.

   Input Parameters:
+  pc - preconditioner
-  x - input vector

   Output Parameter:
.  y - preconditioned vector
*/
PetscErrorCode MatrixFreePreconditioner(PC pc,Vec x,Vec y)
{
  PetscErrorCode ierr;
  ierr = VecCopy(x,y);CHKERRQ(ierr);
```

```
  return 0;
}
```

Figure 14: Example of Uniprocess SNES Code - Both Conventional and Matrix-Free Jacobians

5.6 Finite Difference Jacobian Approximations

PETSc provides some tools to help approximate the Jacobian matrices efficiently via finite differences. These tools are intended for use in certain situations where one is unable to compute Jacobian matrices analytically, and matrix-free methods do not work well without a preconditioner, due to very poor conditioning. The approximation requires several steps:

- First, one colors the columns of the (not yet built) Jacobian matrix, so that columns of the same color do not share any common rows.
- Next, one creates a MatFDColoring data structure that will be used later in actually computing the Jacobian.
- Finally, one tells the nonlinear solvers of SNES to use the SNESComputeJacobianDefaultColor() routine to compute the Jacobians.

A code fragment that demonstrates this process is given below.

```
ISColoring iscoloring;
MatFDColoring fdcoloring;
MatColoring coloring;
/*
This initializes the nonzero structure of the Jacobian. This is artificial
because clearly if we had a routine to compute the Jacobian we wouldn't
need to use finite differences.
*/
FormJacobian(snes,x,&J,&J,&user);
/*
Color the matrix, i.e. determine groups of columns that share no common
rows. These columns in the Jacobian can all be computed simultaneously.
*/
MatColoringCreate(J,&coloring);
MatColoringSetType(coloring,MATCOLORINGSL);
MatColoringSetFromOptions(coloring);
MatColoringApply(coloring,&iscoloring);
MatColoringDestroy(&coloring);
/*
Create the data structure that SNESComputeJacobianDefaultColor() uses
to compute the actual Jacobians via finite differences.
```

```
*/
MatFDColoringCreate(J,iscoloring,&fdcoloring);
ISColoringDestroy(&iscoloring);
MatFDColoringSetFunction(fdcoloring,(PetscErrorCode (*)(void))FormFunction,&user);
MatFDColoringSetFromOptions(fdcoloring);
/*
Tell SNES to use the routine SNESComputeJacobianDefaultColor()
to compute Jacobians.
*/
SNESSetJacobian(snes,J,J,SNESComputeJacobianDefaultColor,fdcoloring);
```

Of course, we are cheating a bit. If we do not have an analytic formula for computing the Jacobian, then how do we know what its nonzero structure is so that it may be colored? Determining the structure is problem dependent, but fortunately, for most structured grid problems (the class of problems for which PETSc is designed) if one knows the stencil used for the nonlinear function one can usually fairly easily obtain an estimate of the location of nonzeros in the matrix. This is harder in the unstructured case, and has not yet been implemented in general.

One need not necessarily use a MatColoring object to determine a coloring. For example, if a grid can be colored directly (without using the associated matrix), then that coloring can be provided to MatFDColoringCreate(). Note that the user must always preset the nonzero structure in the matrix regardless of which coloring routine is used.

For sequential matrices PETSc provides three matrix coloring routines on from the MINPACK package [21]: smallest-last (`sl`), largest-first (`lf`), and incidence-degree (`id`). In addition, two implementations of parallel colorings are in PETSc, greedy (`greedy`) and Jones-Plassmann (`jp`). These colorings, as well as the "natural" coloring for which each column has its own unique color, may be accessed with the command line options

-mat_coloring_type <sl,id,lf,natural,greedy,jp>

Alternatively, one can set a coloring type of `MATCOLORINGGREEDY` or `MATCOLORINGJP` for parallel algorithms, or `MATCOLORINGSL`, `MATCOLORINGID`, `MATCOLORINGLF`, `MATCOLORINGNATURAL` for sequential algorithms when calling MatColoringSetType().

As for the matrix-free computation of Jacobians (see Section 5.5), two parameters affect the accuracy of the finite difference Jacobian approximation. These are set with the command

MatFDColoringSetParameters(MatFDColoring fdcoloring,double rerror,double umin);

The parameter `rerror` is the square root of the relative error in the function evaluations, e_{rel}; the default is 10^{-8}, which assumes that the functions are evaluated to full double-precision accuracy. The second parameter, `umin`, is a bit more involved; its default is $10e^{-8}$. Column i of the Jacobian matrix (denoted by $F_{:i}$) is approximated by the formula

$$F'_{:i} \approx \frac{F(u + h * dx_i) - F(u)}{h}$$

where h is computed via

$$\begin{aligned} h &= e_{rel} * u_i \quad \text{if } |u_i| > u_{min} \\ h &= e_{rel} * u_{min} * sign(u_i) \quad \text{otherwise.} \end{aligned}$$

These parameters may be set from the options database with

-mat_fd_coloring_err *err*
-mat_fd_coloring_umin *umin*

Note that MatColoring type MATCOLORINGSL, MATCOLORINGLF, and MATCOLORINGID are sequential algorithms. MATCOLORINGJP and MATCOLORINGGREEDY are parallel algorithms, although in practice they may create more colors than the sequential algorithms. If one computes the coloring `iscoloring` reasonably with a parallel algorithm or by knowledge of the discretization, the routine MatFDColoringCreate() is scalable. An example of this for 2D distributed arrays is given below that uses the utility routine DMCreateColoring().

```
DMCreateColoring(da,IS_COLORING_GHOSTED,&iscoloring);
MatFDColoringCreate(J,iscoloring,&fdcoloring);
MatFDColoringSetFromOptions(fdcoloring);
ISColoringDestroy(&iscoloring);
```

Note that the routine MatFDColoringCreate() currently is only supported for the AIJ and BAIJ matrix formats.

5.7 Variational Inequalities

SNES can also solve variational inequalities with box constraints. That is nonlinear algebraic systems with additional inequality constraints on some or all of the variables: $Lu_i \leq u_i \leq Hu_i$. Some or all of the lower bounds may be negative infinity (indicated to PETSc with SNES_VI_NINF) and some or all of the upper bounds may be infinity (indicated by SNES_VI_INF). The command

```
SNESVISetVariableBounds(SNES,Vec Lu,Vec Hu);
```

is used to indicate that one is solving a variational inequality. The option `-snes_vi_monitor` turns on extra monitoring of the active set associated with the bounds and `-snes_vi_type` allows selecting from several VI solvers, the default is preferred.

5.8 Nonlinear Preconditioning

Nonlinear preconditioning in PETSc involves the use of an inner SNES instance to define the step for an outer SNES instance. The inner instance may be extracted using

```
SNESGetNPC(SNES snes,SNES *npc);
```

and passed run-time options using the `-npc_` prefix. Nonlinear preconditioning comes in two flavors: left and right. The side may be changed using `-snes_npc_side` or SNESSetNPCSide(). Left nonlinear preconditioning redefines the nonlinear function as the action of the nonlinear preconditioner $\mathbf{M}$;

$$\boldsymbol{F}_M(x) = \mathbf{M}(\boldsymbol{x}, \mathbf{b}) - \boldsymbol{x}. \tag{5.6}$$

Right nonlinear preconditioning redefines the nonlinear function as the function on the action of the nonlinear preconditioner;

$$\boldsymbol{F}(\mathbf{M}(\boldsymbol{x}, \mathbf{b})) = \mathbf{b}, \tag{5.7}$$

which can be interpreted as putting the preconditioner into "striking distance" of the solution by outer acceleration.

In addition, basic patterns of solver composition are available with the SNESType SNESCOMPOSITE. This allows for two or more SNES instances to be combined additively or multiplicatively. By command line, a set of SNES types may be given by comma separated list argument to `-snes_composite_sneses`. There are additive (`SNES_COMPOSITE_ADDITIVE`), additive with optimal damping (`SNES_COMPOSITE_ADDITIVEOPTIMAL`), and multiplicative (`SNES_COMPOSITE_MULTIPLICATIVE`) variants which may be set with

SNESCompositeSetType(SNES,SNESCompositeType);

New subsolvers may be added to the composite solver with

SNESCompositeAddSNES(SNES,SNESType);

and accessed with

SNESCompositeGetSNES(SNES,PetscInt,SNES *);.

Chapter 6

TS: Scalable ODE and DAE Solvers

The TS library provides a framework for the scalable solution of ODEs and DAEs arising from the discretization of time-dependent PDEs.

Simple Example: Consider the PDE

$$u_t = u_{xx}$$

discretized with centered finite differences in space yielding the semi-discrete equation

$$\begin{aligned} (u_i)_t &= \frac{u_{i+1} - 2u_i + u_{i-1}}{h^2}, \\ u_t &= \tilde{A}u; \end{aligned}$$

or with piecewise linear finite elements approximation in space $u(x,t) \doteq \sum_i \xi_i(t)\phi_i(x)$ yielding the semi-discrete equation

$$B\xi'(t) = A\xi(t)$$

Now applying the backward Euler method results in

$$(B - dt^n A)u^{n+1} = Bu^n,$$

in which

$$\begin{aligned} {u^n}_i &= \xi_i(t_n) \doteq u(x_i, t_n), \\ \xi'(t_{n+1}) &\doteq \frac{{u^{n+1}}_i - {u^n}_i}{dt^n}, \end{aligned}$$

A is the stiffness matrix and B is the identity for finite differences or the mass matrix for the finite element method.

The PETSc interface for solving time dependent problems assumes the problem is written in the form

$$F(t, u, \dot{u}) = G(t, u), \quad u(t_0) = u_0.$$

In general, this is a differential algebraic equation (DAE) [1] For ODE with nontrivial mass matrices such as arise in FEM, the implicit/DAE interface significantly reduces overhead to prepare the system for algebraic solvers (SNES/KSP) by having the user assemble the correctly shifted matrix. Therefore this interface is also useful for ODE systems.

To solve an ODE or DAE one uses:

[1] If the matrix $F_{\dot{u}}(t) = \partial F/\partial \dot{u}$ is nonsingular then it is an ODE and can be transformed to the standard explicit form, although this transformation may not lead to efficient algorithms.

- Function $F(t, u, \dot{u})$

  TSSetIFunction(TS ts,Vec R,PetscErrorCode (*f)(TS,PetscReal,Vec,Vec,Vec,void*),void *funP);

  The vector `R` is an optional location to store the residual. The arguments to the function `f()` are the timestep context, current time, input state u, input time derivative $\dot{u}$, and the (optional) user-provided context `funP`. If $F(t, u, \dot{u}) = \dot{u}$ then one need not call this function.

- Function $G(t, u)$, if it is nonzero, is provided with the function

  TSSetRHSFunction(TS ts,Vec R,PetscErrorCode (*f)(TS,PetscReal,Vec,Vec,void*),void *funP);

- Jacobian $(shift)F_{\dot{u}}(t^n, u^n, \dot{u}^n) + F_u(t^n, u^n, \dot{u}^n)$
  If using a fully implicit or semi-implicit (IMEX) method one also can provide an appropriate (approximate) Jacobian matrix of $F()$.

  TSSetIJacobian(TS ts,Mat A,Mat B,
  PetscErrorCode (*fjac)(TS,PetscReal,Vec,Vec,PetscReal,Mat,Mat,void*),void *jacP);

  The arguments for the function `fjac()` are the timestep context, current time, input state u, input derivative $\dot{u}$, input $shift$, matrix A, preconditioning matrix B, and the (optional) user-provided context `jacP`.

  The Jacobian needed for the nonlinear system is, by the chain rule,

  $$\frac{dF}{du^n} = \frac{\partial F}{\partial \dot{u}}|_{u^n} \frac{\partial \dot{u}}{\partial u}|_{u^n} + \frac{\partial F}{\partial u}|_{u^n}.$$

  For any ODE integration method the approximation of $\dot{u}$ is linear in u^n hence $\frac{\partial \dot{u}}{\partial u}|_{u^n} = (shift)$, where shift depends on the ODE integrator and time step but not on the function being integrated. Thus

  $$\frac{dF}{du^n} = (shift)F_{\dot{u}}(t^n, u^n, \dot{u}^n) + F_u(t^n, u^n, \dot{u}^n).$$

  This explains why the user provide Jacobian is in the given form for all integration methods. An equivalent way to derive the formula is to note that

  $$F(t^n, u^n, \dot{u}^n) = F(t^n, u^n, w + shift * u^n)$$

  where w is some linear combination of previous time solutions of u so that

  $$\frac{dF}{du^n} = (shift)F_{\dot{u}}(t^n, u^n, \dot{u}^n) + F_u(t^n, u^n, \dot{u}^n)$$

  again by the chain rule.

  For example, consider backward Euler's method applied to the ODE $F(t, u, \dot{u}) = \dot{u} - f(t, u)$ with $\dot{u} = (u^n - u^{n-1})/\delta t$ and $\frac{\partial \dot{u}}{\partial u}|_{u^n} = 1/\delta t$ resulting in

  $$\frac{dF}{du^n} = (1/\delta t)F_{\dot{u}} + F_u(t^n, u^n, \dot{u}^n).$$

  But $F_{\dot{u}} = 1$, in this special case, resulting in the expected Jacobian $I/\delta t - f_u(t, u^n)$.

- Jacobian G_u
  If using a fully implicit method and the function $G()$ is provided, one also can provide an appropriate (approximate) Jacobian matrix of $G()$.

  TSSetRHSJacobian(TS ts,Mat A,Mat B,
    PetscErrorCode (*fjac)(TS,PetscReal,Vec,Mat,Mat,void*),void *jacP);

  The arguments for the function `fjac()` are the timestep context, current time, input state u, matrix A, preconditioning matrix B, and the (optional) user-provided context `jacP`.

Providing appropriate $F()$ and $G()$ for your problem allows for the easy runtime switching between explicit, semi-implicit (IMEX), and fully implicit method.

6.1 Basic TS Options

The user first creates a TS object with the command

int TSCreate(MPI_Comm comm,TS *ts);

int TSSetProblemType(TS ts,TSProblemType problemtype);

The TSProblemType is one of `TS_LINEAR` or `TS_NONLINEAR`.
To set up TS for solving an ODE, one must set the "initial conditions" for the ODE with

TSSetSolution(TS ts, Vec initialsolution);

One can set the solution method with the routine

TSSetType(TS ts,TSType type);

Currently supported types are TSEULER, TSRK (Runge-Kutta), TSBEULER, TSCN (Crank-Nicolson), TSTHETA, TSGL (generalized linear), TSPSEUDO, and TSSUNDIALS (only if the Sundials package is installed), or the command line option
`-ts_type euler,rk,beuler,cn,theta,gl,pseudo,sundials,eimex,arkimex,rosw`.

A list of available methods is given in Table 10.
Set the initial time and timestep with the command

TSSetInitialTimeStep(TS ts,double time,double dt);

One can change the timestep with the command

TSSetTimeStep(TS ts,double dt);

can determine the current timestep with the routine

TSGetTimeStep(TS ts,double* dt);

Here, "current" refers to the timestep being used to attempt to promote the solution form u^n to u^{n+1}.
One sets the total number of timesteps to run or the total time to run (whatever is first) with the command

TSSetDuration(TS ts,int maxsteps,double maxtime);

TSSetExactFinalTime(TS ts,TS_EXACTFINALTIME_STEPOVER,TS_EXACTFINALTIME_INTERPOLATE,TS_EXACT

Table 10: Time integration schemes.

TS Name	Reference	Class	Type	Order
euler	forward Euler	one-step	explicit	1
ssp	multistage SSP [20]	Runge-Kutta	explicit	≤ 4
rk*	multistage	Runge-Kutta	explicit	≥ 1
beuler	backward Euler	one-step	implicit	1
cn	Crank-Nicolson	one-step	implicit	2
theta*	theta-method	one-step	implicit	≤ 2
alpha	alpha-method [18]	one-step	implicit	2
gl	general linear [5]	multistep-multistage	implicit	≤ 3
eimex	extrapolated IMEX [7]	one-step	≥ 1, adaptive	
arkimex	see Tab. 12	IMEX Runge-Kutta	IMEX	$1-5$
rosw	see Tab. 13	Rosenbrock-W	linearly implicit	$1-4$

One performs the request number of time steps with

TSSolve(TS ts,Vec U);

The solve call implicitly sets up the timestep context; this can be done explicitly with

TSSetUp(TS ts);

One destroys the context with

TSDestroy(TS *ts);

and views it with

TSView(TS ts,PetscViewer viewer);

In place of TSSolve(), a single step can be taken using

TSStep(TS ts);

6.1.1 Using Implicit-Explicit (IMEX) methods for stiff problems

For problems with multiple time scales $F()$ will be treated implicitly using a method suitable for stiff problems and $G()$ will be treated explicitly when using an IMEX method like TSARKIMEX. $F()$ is typically linear or weakly nonlinear while $G()$ may have very strong nonlinearities such as arise in non-oscillatory methods for hyperbolic PDE. The user provides three pieces of information, the APIs for which have been described above.

- "Slow" part $G(t,u)$ using TSSetRHSFunction().
- "Stiff" part $F(t,u,\dot{u})$ using TSSetIFunction().
- Jacobian $F_u + (shift)F_{\dot{u}}$ using TSSetIJacobian().

The user needs to set TSSetEquationType() to TS_EQ_IMPLICIT or higher if the problem is implicit; e.g., $F(t,u,\dot{u}) = M\dot{u} - f(t,u)$, where M is not the identity matrix:

- the problem is an implicit ODE (defined implicitly through TSSetIFunction()) or

Table 11: In PETSc the DAEs and ODEs are formulated as $F(t, y, \dot{y}) = G(t, y)$, where $F()$ is meant to be integrated implicitly and $G()$ explicitly. An IMEX formulation such as $M\dot{y} = g(t, y) + f(t, y)$ requires the user to provide $M^{-1}g(t, y)$ or solve $g(t, y) - Mx = 0$ in place of $G(t, u)$. General cases such as $F(t, y, \dot{y}) = G(t, y)$ are not amenable to IMEX Runge-Kutta, but can be solved by using fully implicit methods.

$\dot{y} = g(t, y)$	nonstiff ODE	$F(t, y, \dot{y}) = \dot{y}, G(t, y) = g(t, y)$
$\dot{y} = f(t, y)$	stiff ODE	$F(t, y, \dot{y}) = \dot{y} - f(t, y), G(t, y) = 0$
$M\dot{y} = f(t, y)$	stiff ODE w/ mass matrix	$F(t, y, \dot{y}) = M\dot{y} - f(t, y), G(t, y) = 0$
$M\dot{y} = g(t, y)$	nonstiff ODE w/ mass matrix	$F(t, y, \dot{y}) = \dot{y}, G(t, y) = M^{-1}g(t, y)$
$\dot{y} = g(t, y) + f(t, y)$	stiff-nonstiff ODE	$F(t, y, \dot{y}) = \dot{y} - f(t, y), G(t, y) = g(t, y)$
$M\dot{y} = g(t, y) + f(t, y)$	mass & stiff-nonstiff ODE	$F(t, y, \dot{y}) = M\dot{y} - f(t, y)$, $G(t, y) = M^{-1}g(t, y)$
$f(t, y, \dot{y}) = 0$	implicit ODE/DAE	$F(t, y, \dot{y}) = f(t, y, \dot{y})$, $G(t, y) = 0$; the user needs to set TSSetEquationType() to TS_EQ_IMPLICIT or higher

Table 12: List of the currently available IMEX Runge-Kutta schemes. For each method we listed the -ts_arkimex_type name, the reference, the total number of stages/implicit stages, the order/stage-order, the implicit stability properties, stiff accuracy (SA), the existence of an embedded scheme, and dense output.

Name	**Reference**	**Stages (IM)**	**Order (stage)**	**IM Stab.**	**SA**	**Embed.**	**Dense Output**	**Remarks**
a2	based on CN	2(1)	2(2)	A-Stable	yes	yes(1)	yes(2)	
l2	SSP2(2,2,2)[23]	2(2)	2(1)	L-Stable	yes	yes(1)	yes(2)	SSP, SDIRK
ars122	ARS122, [1]	2(1)	3(1)	A-Stable	yes	yes(1)	yes(2)	
2c	[13]	3(2)	2(2)	L-Stable	yes	yes(1)	yes(2)	SDIRK
2d	[13]	3(2)	2(2)	L-Stable	yes	yes(1)	yes(2)	SDIRK
2e	[13]	3(2)	2(2)	L-Stable	yes	yes(1)	yes(2)	SDIRK
prssp2	PRS(3,3,2) [23]	3(3)	3(1)	L-Stable	yes	no	no	SSP, nonSDIRK
3	[19]	4(3)	3(2)	L-Stable	yes	yes(2)	yes(2)	SDIRK
bpr3	[3]	5(4)	3(2)	L-Stable	yes	no	no	SDIRK, DAE-1
ars443	[1]	5(4)	3(1)	L-Stable	yes	no	no	SDIRK
4	[19]	6(5)	4(2)	L-Stable	yes	yes(3)	yes(2,3)	SDIRK
5	[19]	8(7)	5(2)	L-Stable	yes	yes(4)	yes(3)	SDIRK

Table 13: List of the currently available Rosenbrock W-schemes. For each method we listed the reference, the total number of stages and implicit stages, the scheme order and stage-order, the implicit stability properties, stiff accuracy (SA), the existence of an embedded scheme, dense output, the capacity to use inexact Jacobian matrices (-W), and high order integration of differential algebraic equations (PDAE).

TS Name	Reference	Stages (IM)	Order (stage)	IM Stab.	SA	Embed.	Dense Output	-W	PDAE	Remarks
theta1	classical	1(1)	1(1)	L-Stable	-	-	-	-	-	-
theta2	classical	1(1)	2(2)	A-Stable	-	-	-	-	-	-
2m	Zoltan	2(2)	2(1)	L-Stable	No	Yes(1)	Yes(2)	Yes	No	SSP
2p	Zoltan	2(2)	2(1)	L-Stable	No	Yes(1)	Yes(2)	Yes	No	SSP
ra3pw	[25]	3(3)	3(1)	A-Stable	No	Yes	Yes(2)	No	Yes(3)	-
ra34pw2	[25]	4(4)	3(1)	L-Stable	Yes	Yes	Yes(3)	Yes	Yes(3)	-
rodas3	[27]	4(4)	3(1)	L-Stable	Yes	Yes	No	No	Yes	-
sandu3	[27]	3(3)	3(1)	L-Stable	Yes	Yes	Yes(2)	No	No	-
assp3p3s1c	unpublished	3(2)	3(1)	A-Stable	No	Yes	Yes(2)	Yes	No	SSP
lassp3p4s2c	unpublished	4(3)	3(1)	L-Stable	No	Yes	Yes(3)	Yes	No	SSP
lassp3p4s2c	unpublished	4(3)	3(1)	L-Stable	No	Yes	Yes(3)	Yes	No	SSP
ark3	unpublished	4(3)	3(1)	L-Stable	No	Yes	Yes(3)	Yes	No	IMEX-RK

- a DAE is being solved.

An IMEX problem representation can be made implicit by setting TSARKIMEXSetFullyImplicit().

Some use-case examples for TSARKIMEX are listed in Table 11 and a list of methods with a summary of their properties is given in Table 12.

ROSW are linearized implicit Runge-Kutta methods known as Rosenbrock W-methods. They can accommodate inexact Jacobian matrices in their formulation. A series of methods are available in PETSc listed in Table 13.

6.1.2 Using fully implicit methods

Either provide the Jacobian of $F()$ (and $G()$ if $G()$ is provided) or use a DM that provides a coloring so the Jacobian can be computed efficiently via finite differences.

Theta methods

GL methods

6.1.3 Using the Explicit Runge-Kutta timestepper with variable timesteps

The explicit Euler and Runge-Kutta methods require the ODE be in the form

$$\dot{u} = G(u, t).$$

The user can either call TSSetRHSFunction() and/or they can call TSSetIFunction() (so long as the function provided to TSSetIFunction() is equivalent to $\dot{u} + \tilde{F}(t, u)$) but the Jacobians need not be provided.[2]

The Explicit Runge-Kutta timestepper with variable timesteps is an implementation of the standard Runge-Kutta with an embedded method. The error in each timestep is calculated using the solutions from the Runge-Kutta method and its embedded method (the 2-norm of the difference is used). The default method is the 3rd-order Bogacki-Shampine method with a 2nd-order embedded method (TSRK3BS). Other available methods are the 5th-order Fehlberg RK scheme with a 4th-order embedded method (TSRK5F), and the 5th-order Dormand-Prince RK scheme with a 4th-order embedded method (TSRK5DP). Variable timesteps cannot be used with RK schemes that do not have an embedded method (TSRK1 - 1st-order, 1-stage forward Euler, TSRK2A - 2nd-order, 2-stage RK scheme, TSRK3 - 3rd-order, 3-stage RK scheme, TSRK4 - 4-th order, 4-stage RK scheme).

[2] PETSc will automatically translate the function provided to the appropriate form.

6.1.4 Special Cases

- $\dot{u} = Au$. First compute the matrix A then call

  TSSetProblemType(ts,TS_LINEAR);
  TSSetRHSFunction(ts,NULL,TSComputeRHSFunctionLinear,NULL);
  TSSetRHSJacobian(ts,A,A,TSComputeRHSJacobianConstant,NULL);

  or

  TSSetProblemType(ts,TS_LINEAR);
  TSSetIFunction(ts,NULL,TSComputeIFunctionLinear,NULL);
  TSSetIJacobian(ts,A,A,TSComputeIJacobianConstant,NULL);

- $\dot{u} = A(t)u$. Use

  TSSetProblemType(ts,TS_LINEAR);
  TSSetRHSFunction(ts,NULL,TSComputeRHSFunctionLinear,NULL);
  TSSetRHSJacobian(ts,A,A,YourComputeRHSJacobian,&appctx);

  where `YourComputeRHSJacobian()` is a function you provide that computes A as a function of time. Or use

  TSSetProblemType(ts,TS_LINEAR);
  TSSetIFunction(ts,NULL,TSComputeIFunctionLinear,NULL);
  TSSetIJacobian(ts,A,A,YourComputeIJacobian,&appctx);

6.1.5 Monitoring and visualizing solutions

- -ts_monitor - prints the time and timestep at each iteration.
- -ts_adapt_monitor - prints information about the timestep adaption calculation at each iteration.
- -ts_monitor_lg_timestep - plots the size of each timestep, TSMonitorLGTimeStep().
- -ts_monitor_lg_solution - for ODEs with only a few components (not arising from the discretization of a PDE) plots the solution as a function of time, TSMonitorLGSolution().
- -ts_monitor_lg_error - for ODEs with only a few components plots the error as a function of time, only if TSSetSolutionFunction() is provided, TSMonitorLGError().
- -ts_monitor_draw_solution - plots the solution at each iteration, TSMonitorDrawSolution().
- -ts_monitor_draw_error - plots the error at each iteration only if TSSetSolutionFunction() is provided, TSMonitorDrawSolution().
- -ts_monitor_solution binary[:filename] - saves the solution at each iteration to a binary file, TSMonitorSolution().
- -ts_monitor_solution_vtk ¡ filename-

Table 14: TSAdapt: available adaptors.

ID	Name	Notes
TSADAPTBASIC	"basic"	the default adaptor
TSADAPTNONE	"none"	no adaptivity

6.1.6 Error control via variable time-stepping

Most of the time stepping methods avaialable in PETSc have an error estimation and error control mechanism. This mechanism is implemented by changing the step size in order to maintain user specified absolute and relative tolerances. The PETSc object responsible with error control is TSAdapt. The avaialble TSAdapt types are listed in Table 14.

`TSAdapt` basic. The user typically provides a desired absolute $\mathrm{Tol_A}$ or a relative $\mathrm{Tol_R}$ error tolerance by invoking TSSetTolerances or at the command line with options `-ts_atol` and `-ts_rtol`. The error estimate is based on the local truncation error, so for every step the algorithm verifies that the estimated local truncation error satisfies the tolerances provided by the user and computes a new step size to be taken. For multistage methods, the local truncation is obtained by comparing the solution y to a lower order $\widehat{p} = p - 1$ approximation, $\widehat{y}$, where p is the order of the method and $\widehat{p}$ the order of $\widehat{y}$.

The adaptive controller at step n computes a tolerance level

$$Tol_n(i) \quad = \quad \mathrm{Tol_A}(i) + \max(y_n(i), \widehat{y}_n(i))\mathrm{Tol_R}(i)\,,$$

and forms the acceptable error level

$$\mathrm{wlte_n} \quad = \quad \frac{1}{m}\sum_{i=1}^{m}\sqrt{\frac{\|y_n(i) - \widehat{y}_n(i)\|}{Tol(i)}}\,,$$

where the errors are computed componentwise, m is the dimension of y and `-ts_adapt_wnormtype` is 2 (default). If `-ts_adapt_wnormtype` is ∞ (max norm), then

$$\mathrm{wlte_n} \quad = \quad \max_{1\ldots m}\frac{\|y_n(i) - \widehat{y}_n(i)\|}{Tol(i)}\,.$$

The error tolerances are satisfied when $\mathrm{wlte} \leq 1.0$.

The next step size is based on this error estimate, and determined by

$$\Delta t_{\mathrm{new}}(t) \quad = \quad \Delta t_{\mathrm{old}}\min(\alpha_{\max}, \max(\alpha_{\min}, \beta(1/\mathrm{wlte})^{\frac{1}{\widehat{p}+1}}))\,, \tag{6.1}$$

where $\alpha_{\min}$ =`-ts_adapt_basic_clip`[0] and $\alpha_{\max}$=`-ts_adapt_basic_clip`[1] keep the change in Δt to within a certain factor, and $\beta < 1$ is chosen through `-ts_adapt_basic_safety` so that there is some margin to which the tolerances are satisfied and so that the probability of rejection is decreased.

This adaptive controller works in the following way. After completing step k, if $\mathrm{wlte_{k+1}} \leq 1.0$, then the step is accepted and the next step is modified according to (6.1); otherwise, the step is rejected and retaken with the step length computed in (6.1).

6.1.7 Handling of discontinuities

For problems that involve discontinuous right hand sides, one can set an "event" function $g(t, u)$ for PETSc to detect and locate the times of discontinuities (zeros of $g(t, u)$). Events can be defined through the event monitoring routine

```
TSSetEventHandler(TS ts,PetscInt nevents,PetscInt *direction,PetscBool *terminate,
  PetscErrorCode (*eventhandler)(TS,PetscReal,Vec,PetscScalar*,void* eventP),
  PetscErrorCode (*postevent)(TS,PetscInt,PetscInt[],PetscReal,Vec,PetscBool,void* eventP),void *eventP);
```

Here, `nevents` denotes the number of events, `direction` sets the type of zero crossing to be detected for an event (+1 for positive zero-crossing, -1 for negative zero-crossing, and 0 for both), `terminate` conveys whether the time-stepping should continue or halt when an event is located, `eventmonitor` is a user-defined routine that specifies the event description, `postevent` is an optional user-defined routine to take specific actions following an event.

The arguments to `eventhandler()` are the timestep context, current time, input state u, array of event function value, and the (optional) user-provided context `eventP`.

The arguments to `postevent()` routine are the timestep context, number of events occured, indices of events occured, current time, input state u, a boolean flag indicating forward solve (1) or adjoint solve (0), and the (optional) user-provided context `eventP`.

The event monitoring functionality is only available with PETSc's implicit time-stepping solvers TSTHETA, TSARKIMEX, and TSROSW.

6.1.8 Debugging ODE solvers

6.1.9 Using Sundials from PETSc

Sundials is a parallel ODE solver developed by Hindmarsh et al. at LLNL. The TS library provides an interface to use the CVODE component of Sundials directly from PETSc. (To install PETSc to use Sundials, see the installation guide, `docs/installation/index.htm`.)

To use the Sundials integrators, call

```
TSSetType(TS ts,TSType TSSUNDIALS);
```

or use the command line option `-ts_type sundials`.

Sundials' CVODE solver comes with two main integrator families, Adams and BDF (backward differentiation formula). One can select these with

```
TSSundialsSetType(TS ts,TSSundialsLmmType [SUNDIALS_ADAMS,SUNDIALS_BDF]);
```

or the command line option `-ts_sundials_type <adams,bdf>`. BDF is the default.

Sundials does not use the SNES library within PETSc for its nonlinear solvers, so one cannot change the nonlinear solver options via SNES. Rather, Sundials uses the preconditioners within the PC package of PETSc, which can be accessed via

```
TSSundialsGetPC(TS ts,PC *pc);
```

The user can then directly set preconditioner options; alternatively, the usual runtime options can be employed via `-pc_xxx`.

Finally, one can set the Sundials tolerances via

```
TSSundialsSetTolerance(TS ts,double abs,double rel);
```

where `abs` denotes the absolute tolerance and `rel` the relative tolerance.

Other PETSc-Sundials options include

```
TSSundialsSetGramSchmidtType(TS ts,TSSundialsGramSchmidtType type);
```

where `type` is either `SUNDIALS_MODIFIED_GS` or `SUNDIALS_UNMODIFIED_GS`. This may be set via the options data base with `-ts_sundials_gramschmidt_type <modifed,unmodified>`.

The routine

TSSundialsSetGMRESRestart(TS ts,int restart);

sets the number of vectors in the Krylov subpspace used by GMRES. This may be set in the options database with `-ts_sundials_gmres_restart restart`.

Chapter 7

Performing sensitivity analysis in PETSc

The TS library provides a framework based on discrete adjoint models for sensitivity analysis for ODEs and DAEs. The ODE/DAE solution process (henceforth called the forward run) can be obtained by using either explicit or implicit solvers in TS, depending on the problem properties. Currently supported method types are TSRK (Runge-Kutta) explicit methods and TSTHETA implicit methods, which include TSBEULER and TSCN.

7.1 Using the discrete adjoint methods

Consider the ODE/DAE

$$F(t, y, \dot{y}, p) = 0, \quad y(t_0) = y_0(p) \quad t_0 \leq t \leq t_F$$

and the cost function(s)

$$\Psi_i(y_0, p) = \Phi_i(y_F, p) + \int_{t_0}^{t_F} r_i(y(t), p, t)dt \quad i = 1, ..., numcost.$$

The TSAdjoint routines of PETSc provide

$$\frac{\partial \Psi_i}{\partial y_0} = \lambda_i$$

and

$$\frac{\partial \Psi_i}{\partial p} = \mu_i + \lambda_i(\frac{\partial y_0}{\partial p}).$$

To perform the discrete adjoint sensitivity analysis one first sets up the TS object for a regular forward run but with one extra function call

TSSetSaveTrajectory(TS ts),

then calls TSSolve() in the usual manner.

One must create two arrays of $numcost$ vectors $lambda$ and mu (if there are no parameters p then one can use $NULL$ for the mu array.) The $lambda$ vectors are the same dimension and parallel layout as the solution vector for the ODE, the mu vectors are of dimension p; when p is small usually all its elements are on the first MPI process, while the vectors have no entries on the other processes. $lambda_i$ and mu_i should be initialized with the values $d\Phi_i/dy|_{t=t_F}$ and $d\Phi_i/dp|_{t=t_F}$ respectively. Then one calls

TSSetCostGradients(TS ts,PetscInt numcost, Vec *lambda,Vec *mu);

If $F()$ is a function of p one needs to also provide the Jacobian $-F_p$ with

TSAdjointSetRHSJacobian(TS ts,Mat Amat,PetscErrorCode (*fp)(TS,PetscReal,Vec,Mat,void*),void *ctx)

The arguments for the function `fp()` are the timestep context, current time, y, and the (optional) user-provided context.

If there is an integral term in the cost function, i.e. $r()$ is nonzero, one must also call

TSSetCostIntegrand(TS ts,PetscInt numcost,
PetscErrorCode (*rf)(TS,PetscReal,Vec,Vec,void*),
PetscErrorCode (*drdyf)(TS,PetscReal,Vec,Vec*,void*),
PetscErrorCode (*drdpf)(TS,PetscReal,Vec,Vec*,void*),void *ctx)

where $\text{drdyf} = dr/dy$, $\text{drdpf} = dr/dp$.

Lastly one starts the backward run by calling

TSAdjointSolve(TS ts).

One can get the value of the integral term by calling

TSGetCostIntegral(TS ts,Vec *q).

If TSSetCostGradients() and TSSetCostIntegrand() are called before TSSolve(), the integral term will be evaluated in the forward run. Otherwise, the computation will take place in the backward run (inside TSAdjointSolve()). Note that this allows the evaluation of the integral term in the cost function without having to run the adjoint solvers.

To provide a better understanding of the use of the adjoint solvers, we introduce a simple example, corresponding to `${PETSC_DIR}/src/ts/examples/tutorials/power_grid/ex3adj.c`. The problem is to study dynamic security of power system when there are credible contingencies such as short-circuits or loss of generators, transmission lines, or loads. The dynamic security constraints are incorporated as equality constraints in the form of discretized differential equations and inequality constraints for bounds on the trajectory. The governing ODE system is

$$\begin{aligned} \phi' &= \omega_B(\omega - \omega_S) \\ 2H/\omega_S\,\omega' &= p_m - p_{max}sin(\phi) - D(\omega - \omega_S), \quad t_0 \le t \le t_F, \end{aligned}$$

where ϕ is the phase angle and ω is the frequency.

The initial conditions at time t_0 are

$$\begin{aligned} \phi(t_0) &= \arcsin\left(p_m/p_{max}\right), \\ w(t_0) &= 1. \end{aligned}$$

p_{max} is a positive number when the system operates normally. At an event such as fault incidence/removal, p_{max} will change to 0 temporarily and back to the original value after the fault is fixed. The objective is to maximize p_m subject to the above ODE constraints and $\phi < \phi_S$ during all times. To accommodate the inequality constraint, we want to compute the sensitivity of the cost function

$$\Psi(p_m, \phi) = -p_m + c\int_{t_0}^{t_F} \left(\max(0, \phi - \phi_S)\right)^2 dt$$

with respect to the parameter p_m. $numcost$ is 1 since it is a scalar function.

For ODE solution, PETSc requires user-provided functions to evaluate the system $F(t, y, \dot{y}, p)$ (set by TSSetIFunction()) and its corresponding Jacobian $F_y + (shift)F_{\dot{y}}$ (set by TSSetIJacobian()). Note that the solution state y is $[\phi\ \omega]^T$ here. For sensitivity analysis, we need to provide a routine to compute

$f_p = [0\ 1]^T$ using TSAdjointSetRHSJacobian(), and three routines corresponding to the integrand $r = c\left(\max(0, \phi - \phi_S)\right)^2$, $r_p = [0\ 0]^T$ and $r_y = [2c\left(\max(0, \phi - \phi_S)\right)\ 0]^T$ using TSSetCostIntegrand().

In the adjoint run, λ and μ are initialized as $[0\ 0]^T$ and $[-1]$ at the final time t_F. After TSAdjointSolve(), the sensitivity of the cost function w.r.t. to initial conditions is given by the sensitivity variable λ (at time t_0) directly. And the sensitivity of the cost function w.r.t. to the parameter p_m can be computed (by users) as

$$\frac{\mathrm{d}\Psi}{\mathrm{d}p_m} = \mu(t_0) + \lambda(t_0)\frac{\mathrm{d}\left[\phi(t_0)\ \omega(t_0)\right]^T}{\mathrm{d}p_m}.$$

For explicit methods where one does not need to provide the the Jacobian F_u for the forward solve one still does need it for the backward solve and thus must call

TSSetRHSJacobian(TS ts,Mat Amat, Mat Pmat,PetscErrorCode (*f)(TS,double,Vec,Mat,Mat,
void*),void *fP);

Examples include:

- a discrete adjoint sensitivity using explicit time stepping methods `${PETSC_DIR}/src/ts/examples/tutorials/ex16adj.c`,
- a discrete adjoint sensitivity using implicit time stepping methods `${PETSC_DIR}/src/ts/examples/tutorials/ex20adj.c`,
- an optimization using the discrete adjoint models of ERK `${PETSC_DIR}/src/ts/examples/tutorials/ex16opt_ic.c` and `${PETSC_DIR}/src/ts/examples/tutorials/ex16opt_p.c`,
- an optimization using the discrete adjoint models of Theta methods for stiff DAEs `${PETSC_DIR}/src/ts/examples/tutorials/ex20opt_ic.c` and `${PETSC_DIR}/src/ts/examples/tutorials/ex20opt_p.c`,
- an ODE-constrained optimization using the discrete adjoint models of Theta methods for cost function with an integral term `${PETSC_DIR}/src/ts/examples/tutorials/power_grid/ex3opt.c`,
- a discrete adjoint sensitivity using TSCN (Crank-Nicolson) methods for DAEs with discontinuities `${PETSC_DIR}/src/ts/examples/tutorials/power_grid/stability_9bus/ex9busadj.c`,
- a DAE-constrained optimization using the discrete adjoint models of TSCN (Crank-Nicolson) methods for cost function with an integral term `${PETSC_DIR}/src/ts/examples/tutorials/power_grid/stability_9bus/ex9busopt.c`,
- a discrete adjoint sensitivity using TSCN methods for a PDE problem `${PETSC_DIR}/src/ts/examples/tutorials/advection-diffusion-reaction/ex5adj.c`.

7.2 Checkpointing

The discrete adjoint model requires the states (and stage values in the context of multistage timestepping methods) to evaluate the Jacobian matrices during the adjoint (backwarad) run. By default, PETSc stores the whole trajectory to disk as binary files, each of which contains the information for a single time step including state, time, and stage values (optional). One can also make PETSc store the trajectory to memory

with the option `-ts_trajectory_type memory`. However, there might not be sufficient memory capacity especially for large-scale problems and long-time integration.

A so-called checkpointing scheme is needed to solve this problem. The scheme stores checkpoints at selective time steps and recomputes the missing information. The `revolve` library is used by PETSc TSTrajectory to generate an optimal checkpointing schedule that minimizes the recomputations given a limited number of available checkpoints. One can specify the number of available checkpoints with the option `-ts_trajectory_max_cps_ram [maximum number of checkpoints in RAM]`. Note that one checkpoint corresponds to one time step.

The `revolve` library also provides an optimal multistage checkpointing scheme that uses both RAM and disk for storage. This scheme is automatically chosen if one uses both the option `-ts_trajectory_max_cps_ram [maximum number of checkpoints in RAM]` and the option `-ts_trajectory_max_cps_disk [maximum number of checkpoints on disk]`.

Some other useful options are listed below.

- `-ts_trajectory_view` prints the total number of recomputations,
- `-ts_monitor` and `-ts_adjoint_monitor` allow users to monitor the progress of the adjoint work flow,
- `-ts_trajectory_type visualization` may be used to save the whole trajectory for visualization. It stores the solution and the time, but no stage values. The binary files generated can be read into MATLAB via the script `${PETSC_DIR}/share/petsc/matlab/PetscReadBinaryTrajectory.m`.

Chapter 8

Solving Steady-State Problems with Pseudo-Timestepping

Simple Example: TS provides a general code for performing pseudo timestepping with a variable timestep at each physical node point. For example, instead of directly attacking the steady-state problem

$$G(u) = 0,$$

we can use pseudo-transient continuation by solving

$$u_t = G(u).$$

Using time differencing

$$u_t \doteq \frac{u^{n+1} - u^n}{dt^n}$$

with the backward Euler method, we obtain nonlinear equations at a series of pseudo-timesteps

$$\frac{1}{dt^n} B(u^{n+1} - u^n) = G(u^{n+1}).$$

For this problem the user must provide $G(u)$, the time steps dt^n and the left-hand-side matrix B (or optionally, if the timestep is position independent and B is the identity matrix, a scalar timestep), as well as optionally the Jacobian of $G(u)$.

More generally, this can be applied to implicit ODE and DAE for which the transient form is

$$F(u, \dot{u}) = 0.$$

For solving steady-state problems with pseudo-timestepping one proceeds as follows.

- Provide the function `G(u)` with the routine

  TSSetRHSFunction(TS ts,PetscErrorCode (*f)(TS,double,Vec,Vec,void*),void *fP);

  The arguments to the function `f()` are the timestep context, the current time, the input for the function, the output for the function and the (optional) user-provided context variable `fP`.

- Provide the (approximate) Jacobian matrix of `G(u)` and a function to compute it at each Newton iteration. This is done with the command

TSSetRHSJacobian(TS ts,Mat Amat, Mat Pmat,PetscErrorCode (*f)(TS,double,Vec,Mat,Mat,
void*),void *fP);

The arguments for the function `f()` are the timestep context, the current time, the location where the Jacobian is to be computed, the (approximate) Jacobian matrix, an alternative approximate Jacobian matrix used to construct the preconditioner, and the optional user-provided context, passed in as `fP`. The user must provide the Jacobian as a matrix; thus, if using a matrix-free approach, one must create a MATSHELL matrix.

In addition, the user must provide a routine that computes the pseudo-timestep. This is slightly different depending on if one is using a constant timestep over the entire grid, or it varies with location.

- For location-independent pseudo-timestepping, one uses the routine

  TSPseudoSetTimeStep(TS ts,int(*dt)(TS,double*,void*),void* dtctx);

  The function `dt` is a user-provided function that computes the next pseudo-timestep. As a default one can use TSPseudoTimeStepDefault(TS,double*,void*) for `dt`. This routine updates the pseudo-timestep with one of two strategies: the default

  $$dt^n = dt_{\text{increment}} * dt^{n-1} * \frac{||F(u^{n-1})||}{||F(u^n)||}$$

  or, the alternative,

  $$dt^n = dt_{\text{increment}} * dt^0 * \frac{||F(u^0)||}{||F(u^n)||}$$

  which can be set with the call

  TSPseudoIncrementDtFromInitialDt(TS ts);

  or the option `-ts_pseudo_increment_dt_from_initial_dt`. The value $dt_{\text{increment}}$ is by default 1.1, but can be reset with the call

  TSPseudoSetTimeStepIncrement(TS ts,double inc);

  or the option `-ts_pseudo_increment <inc>`.

- For location-dependent pseudo-timestepping, the interface function has not yet been created.

Chapter 9

High Level Support for Multigrid with KSPSetDM() and SNESSetDM()

This chapter needs to be written.

Chapter 10

Using ADIFOR with PETSc

Automatic differentiation is an incredible technique to generate code that computes Jacobians and other derivatives directly from code that only evaluates the function.

First one indicates the functions for which one needs Jacobians by adding where one lists the functions. In Fortran use

! Process adifor: FormFunctionLocal

Next one uses the call

DMDASetSNESLocal(DM dm,
PetscErrorCode (*localfunction)(DMDALocalInfo *info,void *x,void *f,void* appctx),NULL,
ad_localfunction,ad_mf_localfunction);

where the names of the last two functions are obtained by prepending in Fortran, this is done by prepending a `g_` and `m_`.

10.1 Work arrays inside the local functions

In C you can call DMDAVecGetArray() to get work arrays (this is low overhead). In Fortran you can provide a `FormFunctionLocal()` that had local arrays that have hardwired sizes that are large enough or somehow allocate space and pass it into an inner FormFunctionLocal() that is the one you differentiate; this second approach will require some hand massaging. For example,

```
subroutine TrueFormFunctionLocal(info,x,f,ctx,ierr)
double precision x(gxs:gxe,gys:gye),f(xs:xe,ys:ye)
DMDA info(DMDA_LOCAL_INFO_SIZE)
integer ctx
PetscErrorCode ierr
double precision work(gxs:gxe,gys:gye)
.... do the work ....
return
subroutine FormFunctionLocal(info,x,f,ctx,ierr)
double precision x(*),f(*)
DMDA info(DMDA_LOCAL_INFO_SIZE)
PetscErrorCode ierr
integer ctx
double precision work(10000)
```

```
call TrueFormFunctionLocal(info,x,f,work,ctx,ierr)
return
```

Chapter 11

Using MATLAB with PETSc

There are three basic ways to use MATLAB with PETSc: (1) dumping files to be read into MATLAB, (2) automatically sending data from a running PETSc program to a MATLAB process where you may interactively type MATLAB commands (or run scripts), and (3) automatically sending data back and forth between PETSc and MATLAB where MATLAB commands are issued not interactively but from a script or the PETSc program (this uses the MATLAB Engine).

11.1 Dumping Data for MATLAB

One can dump PETSc matrices and vectors to the screen (and thus save in a file via `> filename.m`) in a format that MATLAB can read in directly. This is done with the command line options `-vec_view ::ascii_matlab` or `-mat_view ::ascii_matlab`. This causes the PETSc program to print the vectors and matrices every time a VecAssemblyXXX() and MatAssemblyXXX() is called. To provide finer control over when and what vectors and matrices are dumped one can use the VecView() and MatView() functions with a viewer type of ASCII (see PetscViewerASCIIOpen(), `PETSC_VIEWER_STDOUT_WORLD`, `PETSC_VIEWER_STDOUT_SELF`, or `PETSC_VIEWER_STDOUT_(MPI_Comm)`). Before calling the viewer set the output type with, for example,

PetscViewerPushFormat(PETSC_VIEWER_STDOUT_WORLD,PETSC_VIEWER_ASCII_MATLAB);
VecView(A,PETSC_VIEWER_STDOUT_WORLD); PetscViewerPopFormat(PETSC_VIEWER_STDOUT_WORLD);

The name of each PETSc variable printed for MATLAB may be set with

PetscObjectSetName((PetscObject)A,"name");

If no name is specified, the object is given a default name using PetscObjectName.

11.2 Sending Data to Interactive Running MATLAB Session

One creates a viewer to MATLAB via

PetscViewerSocketOpen(MPI_Comm,char *machine,int port,PetscViewer *v);

(port is usually set to `PETSC_DEFAULT`, use NULL for the machine if the MATLAB interactive session is running on the same machine as the PETSc program) and then sends matrices or vectors via

VecView(Vec A,v);
MatView(Mat B,v);

One can also send arrays or integer arrays via PetscIntView(), PetscRealView() and PetscScalarView(). One may start the MATLAB program manually or use the PETSc command PetscStartMATLAB `(MPI_Comm, char *machine,char *script,FILE **fp)`; where machine and script may be NULL.

To receive the objects in MATLAB you must first make sure that `${PETSC_DIR}/${PETSC_ARCH}/lib/petsc/matlab` is in your MATLAB path. Use `p = sopen;` (or `p = sopen(portnum)` if you provided a port number in your call to PetscViewerSocketOpen()), then `a = PetscBinaryRead(p);` returns the object you have passed from PETSc. PetscBinaryRead() may be called any number of times. Each call should correspond on the PETSc side with viewing a single vector or matrix. You many call `sclose()` to close the connection from MATLAB. It is also possible to start your PETSc program from MATLAB via `launch()`.

11.3 Using the MATLAB Compute Engine

One creates access to the MATLAB engine via

PetscMatlabEngineCreate(MPI_Comm comm,char *machine,PetscMatlabEngine *e);

where `machine` is the name of the machine hosting MATLAB (NULL may be used for localhost). One can send objects to MATLAB via

PetscMatlabEnginePut(PetscMatlabEngine e,PetscObject obj);

One can get objects via

PetscMatlabEngineGet(PetscMatlabEngine e,PetscObject obj);.

Similarly one can send arrays via

PetscMatlabEnginePutArray(PetscMatlabEngine e,int m,int n,PetscScalar *array,char *name);

and get them back via

PetscMatlabEngineGetArray(PetscMatlabEngine e,int m,int n,PetscScalar *array,char *name);

One cannot use MATLAB interactively in this mode but you can send MATLAB commands via

PetscMatlabEngineEvaluate(PetscMatlabEngine,"format",...);

where `format` has the usual `printf()` format. For example,

PetscMatlabEngineEvaluate(PetscMatlabEngine,"x = %g *y + z;",avalue);

The name of each PETSc variable passed to Matlab may be set with

PetscObjectSetName((PetscObject)A,"name");

Text responses can be returned from MATLAB via

PetscMatlabEngineGetOutput(PetscMatlabEngine,char **);

or

PetscMatlabEnginedPrintOutput(PetscMatlabEngine,FILE*).

There is a short-cut to starting the MATLAB engine with `PETSC_MATLAB_ENGINE_(MPI_Comm)`.

Chapter 12

PETSc for Fortran Users

Most of the functionality of PETSc can be obtained by people who program purely in Fortran 77 or Fortran 90. The PETSc Fortran interface works with both F77 and F90 compilers.

Since Fortran77 does not provide type checking of routine input/output parameters, we find that many errors encountered within PETSc Fortran programs result from accidentally using incorrect calling sequences. Such mistakes are immediately detected during compilation when using C/C++. Thus, using a mixture of C/C++ and Fortran often works well for programmers who wish to employ Fortran for the core numerical routines within their applications. In particular, one can effectively write PETSc driver routines in C/C++, thereby preserving flexibility within the program, and still use Fortran when desired for underlying numerical computations. With Fortran 90 compilers we now can provide some type checking from Fortran.

12.1 Differences between PETSc Interfaces for C and Fortran

Only a few differences exist between the C and Fortran PETSc interfaces, all of which are due to Fortran 77 syntax limitations. Since PETSc is primarily written in C, the FORTRAN 90 dynamic allocation is not easily accessible. All Fortran routines have the same names as the corresponding C versions, and PETSc command line options are fully supported. The routine arguments follow the usual Fortran conventions; the user need not worry about passing pointers or values. The calling sequences for the Fortran version are in most cases identical to the C version, except for the error checking variable discussed in Section 12.1.2 and a few routines listed in Section 12.1.10.

12.1.1 Include Files

The Fortran include files for PETSc are located in the directory `${PETSC_DIR}/include/petsc/finclude` and should be used via statements such as the following:

#include ”petsc/finclude/includefile.h”

Since one must be very careful to include each file no more than once in a Fortran routine, application programmers must manually include each file needed for the various PETSc libraries within their program. This approach differs from the PETSc C/C++ interface, where the user need only include the highest level file, for example, `petscsnes.h`, which then automatically includes all of the required lower level files. As shown in the examples of Section 12.2, in Fortran one must explicitly list *each* of the include files. One must employ the Fortran file suffix `.F` rather than `.f`. This convention enables use of the CPP preprocessor, which allows the use of the *#include* statements that define PETSc objects and variables. (Familiarity with the CPP preprocessor is not needed for writing PETSc Fortran code; one can simply begin by copying a PETSc Fortran example and its corresponding makefile.)

For some of the Fortran 90 functionality of PETSc and type checking of PETSc function calls you can use

#include ”petsc/finclude/includefile.h #include ”petsc/finclude/includefile.h90”

See the manual page UsingFortran for how you can use PETSc Fortran module files in your code.

12.1.2 Error Checking

In the Fortran version, each PETSc routine has as its final argument an integer error variable, in contrast to the C convention of providing the error variable as the routine's return value. The error code is set to be nonzero if an error has been detected; otherwise, it is zero. For example, the Fortran and C variants of KSPSolve() are given, respectively, below, where `ierr` denotes the error variable:

```
call KSPSolve(KSP ksp,Vec b,Vec x,PetscErrorCode ierr)
KSPSolve(KSP ksp,Vec b,Vec x);
```

Fortran programmers can check these error codes with `CHKERRQ(ierr)`, which terminates all processes when an error is encountered. Likewise, one can set error codes within Fortran programs by using `SETERRQ(comm,p,' ',ierr)`, which again terminates all processes upon detection of an error. Note that complete error tracebacks with `CHKERRQ()` and `SETERRQ()`, as described in Section 1.4 for C routines, are *not* directly supported for Fortran routines; however, Fortran programmers can easily use the error codes in writing their own tracebacks. For example, one could use code such as the following:

```
call KSPSolve(ksp,b,x,ierr)
if ( ierr .ne. 0) then
print*, 'Error in routine ...'
return
endif
```

The most common reason for crashing PETSc Fortran code is forgetting the final `ierr` argument.

12.1.3 Array Arguments

Since Fortran 77 does not allow arrays to be returned in routine arguments, all PETSc routines that return arrays, such as VecGetArray(), MatDenseGetArray(), and ISGetIndices(), are defined slightly differently in Fortran than in C. Instead of returning the array itself, these routines accept as input a user-specified array of dimension one and return an integer index to the actual array used for data storage within PETSc. The Fortran interface for several routines is as follows:

```
double precision xx_v(1), aa_v(1)
PetscErrorCode ierr
integer ss_v(1), dd_v(1), nloc
PetscOffset ss_i, xx_i, aa_i, dd_i
Vec x
Mat A
IS s
DM d
call VecGetArray(x,xx_v,xx_i,ierr)
call MatDenseGetArray(A,aa_v,aa_i,ierr)
call ISGetIndices(s,ss_v,ss_i,ierr)
```

To access array elements directly, both the user-specified array and the integer index *must* then be used together. For example, the following Fortran program fragment illustrates directly setting the values of a vector array instead of using VecSetValues(). Note the (optional) use of the preprocessor `#define` statement to enable array manipulations in the conventional Fortran manner.

```
#define xx_a(ib) xx_v(xx_i + (ib))
double precision xx_v(1)
PetscOffset xx_i
PetscErrorCode ierr
integer i, n
Vec x
call VecGetArray(x,xx_v,xx_i,ierr)
call VecGetLocalSize(x,n,ierr)
do 10, i=1,n
xx_a(i) = 3*i + 1
10 continue
call VecRestoreArray(x,xx_v,xx_i,ierr)
```

Figure 16 contains an example of using VecGetArray() within a Fortran routine.

Since in this case the array is accessed directly from Fortran, indexing begins with 1, not 0 (unless the array is declared as `xx_v(0:1)`). This is different from the use of VecSetValues() where, indexing always starts with 0.

Note: If using VecGetArray(), MatDenseGetArray(), or ISGetIndices(), from Fortran, the user *must not* compile the Fortran code with options to check for "array entries out of bounds" (e.g., on the IBM RS/6000 this is done with the `-C` compiler option, so never use the `-C` option with this).

12.1.4 Calling Fortran Routines from C (and C Routines from Fortran)

Different machines have different methods of naming Fortran routines called from C (or C routines called from Fortran). Most Fortran compilers change all the capital letters in Fortran routines to small. On some machines, the Fortran compiler appends an underscore to the end of each Fortran routine name; for example, the Fortran routine `Dabsc()` would be called from C with `dabsc_()`. Other machines change all the letters in Fortran routine names to capitals.

PETSc provides two macros (defined in C/C++) to help write portable code that mixes C/C++ and Fortran. They are `PETSC_HAVE_FORTRAN_UNDERSCORE` and `PETSC_HAVE_FORTRAN_CAPS` , which are defined in the file `${PETSC_DIR}/${PETSC_ARCH}/include/petscconf.h`. The macros are used, for example, as follows:

```
#if defined(PETSC_HAVE_FORTRAN_CAPS)
#define dabsc_ DMDABSC
#elif !defined(PETSC_HAVE_FORTRAN_UNDERSCORE)
#define dabsc_ dabsc
#endif
.....
dabsc_(&n,x,y); /* call the Fortran function */
```

12.1.5 Passing Null Pointers

In several PETSc C functions, one has the option of passing a 0 (null) argument (for example, the fifth argument of MatCreateSeqAIJ()). From Fortran, users *must* pass `PETSC_NULL_XXX` to indicate a null argument (where XXX is `INTEGER`, `DOUBLE`, `CHARACTER`, or `SCALAR` depending on the type of argument

required); passing 0 from Fortran will crash the code. Note that the C convention of passing NULL (or 0) *cannot* be used. For example, when no options prefix is desired in the routine PetscOptionsGetInt(), one must use the following command in Fortran:

call PetscOptionsGetInt(PETSC_NULL_OBJECT,PETSC_NULL_CHARACTER,'-name',N,flg,ierr)

This Fortran requirement is inconsistent with C, where the user can employ NULL for all null arguments.

12.1.6 Duplicating Multiple Vectors

The Fortran interface to VecDuplicateVecs() differs slightly from the C/C++ variant because Fortran does not allow arrays to be returned in routine arguments. To create `n` vectors of the same format as an existing vector, the user must declare a vector array, `v_new` of size `n`. Then, after VecDuplicateVecs() has been called, `v_new` will contain (pointers to) the new PETSc vector objects. When finished with the vectors, the user should destroy them by calling VecDestroyVecs(). For example, the following code fragment duplicates `v_old` to form two new vectors, `v_new(1)` and `v_new(2)`.

```
Vec v_old, v_new(2)
integer ierr
PetscScalar alpha
....
call VecDuplicateVecs(v_old,2,v_new,ierr)
alpha = 4.3
call VecSet(v_new(1),alpha,ierr)
alpha = 6.0
call VecSet(v_new(2),alpha,ierr)
....
call VecDestroyVecs(2,&v_new,ierr)
```

12.1.7 Matrix, Vector and IS Indices

All matrices, vectors and IS in PETSc use zero-based indexing, regardless of whether C or Fortran is being used. The interface routines, such as MatSetValues() and VecSetValues(), always use zero indexing. See Section 3.2 for further details.

12.1.8 Setting Routines

When a function pointer is passed as an argument to a PETSc function, such as the test in KSPSetConvergenceTest(), it is assumed that this pointer references a routine written in the same language as the PETSc interface function that was called. For instance, if KSPSetConvergenceTest() is called from C, the test argument is assumed to be a C function. Likewise, if it is called from Fortran, the test is assumed to be written in Fortran.

12.1.9 Compiling and Linking Fortran Programs

Figure 22 shows a sample makefile that can be used for PETSc programs. In this makefile, one can compile and run a debugging version of the Fortran program `ex3.F` with the actions `make ex3` and `make runex3`, respectively. The compilation command is restated below:

```
ex3: ex3.o
  -${FLINKER} -o ex3 ex3.o ${PETSC_LIB}
  ${RM} ex3.o
```

12.1.10 Routines with Different Fortran Interfaces

The following Fortran routines differ slightly from their C counterparts; see the manual pages and previous discussion in this chapter for details:

PetscInitialize(char *filename,int `ierr`)
PetscError(MPI_COMM,int err,char *message,int `ierr`)
VecGetArray(), MatDenseGetArray()
ISGetIndices(),
VecDuplicateVecs(), VecDestroyVecs()
PetscOptionsGetString()

The following functions are not supported in Fortran:

PetscFClose(), PetscFOpen(), PetscFPrintf(), PetscPrintf()
PetscPopErrorHandler(), PetscPushErrorHandler()
PetscInfo()
PetscSetDebugger()
VecGetArrays(), VecRestoreArrays()
PetscViewerASCIIGetPointer(), PetscViewerBinaryGetDescriptor()
PetscViewerStringOpen(), PetscViewerStringSPrintf()
PetscOptionsGetStringArray()

12.1.11 Fortran90

PETSc includes limited support for direct use of Fortran90 pointers. Current routines include:

VecGetArrayF90(), VecRestoreArrayF90()
VecDuplicateVecsF90(), VecDestroyVecsF90()
DMDAVecGetArrayF90(), DMDAVecGetArrayReadF90(), ISLocalToGlobalMappingGetIndicesF90()
MatDenseGetArrayF90(), MatDenseRestoreArrayF90()
ISGetIndicesF90(), ISRestoreIndicesF90()

See the manual pages for details and pointers to example programs. To use the routines VecGetArrayF90(), VecRestoreArrayF90() VecDuplicateVecsF90(), and VecDestroyVecsF90(), one must use the Fortran90 vector include file,

#include ”petsc/finclude/petscvec.h90”

Analogous include files for other libraries are `petscdm.h90`, `petscmat.h90`, and `petscis.h90`.

12.2 Sample Fortran77 Programs

Sample programs that illustrate the PETSc interface for Fortran are given in Figures 15 – 18, corresponding to `${PETSC_DIR}/src/vec/vec/examples/tests/ex19f.F`, `${PETSC_DIR}/src/vec/vec/examples/tutorials/ex4f.F`, `${PETSC_DIR}/src/sys/classes/draw/examples/tests/ex5f.F`, and `${PETSC_DIR}/src/snes/examples/ex1f.F`, respectively. We also refer Fortran programmers to the C examples listed throughout the manual, since PETSc usage within the two languages differs only slightly.

```
!
!
```

```
      program main
      implicit none
#include <petsc/finclude/petscsys.h>
#include <petsc/finclude/petscvec.h>
!
!  This example demonstrates basic use of the PETSc Fortran interface
!  to vectors.
!
       PetscInt  n
       PetscErrorCode ierr
       PetscBool  flg
       PetscScalar      one,two,three,dot
       PetscReal        norm,rdot
       Vec              x,y,w
       PetscOptions     options

       n     = 20
       one   = 1.0
       two   = 2.0
       three = 3.0

       call PetscInitialize(PETSC_NULL_CHARACTER,ierr)
       call PetscOptionsCreate(options,ierr)
       call PetscOptionsGetInt(options,PETSC_NULL_CHARACTER,                  &
     &                         '-n',n,flg,ierr)
       call PetscOptionsDestroy(options,ierr)

! Create a vector, then duplicate it
       call VecCreate(PETSC_COMM_WORLD,x,ierr)
       call VecSetSizes(x,PETSC_DECIDE,n,ierr)
       call VecSetFromOptions(x,ierr)
       call VecDuplicate(x,y,ierr)
       call VecDuplicate(x,w,ierr)

       call VecSet(x,one,ierr)
       call VecSet(y,two,ierr)

       call VecDot(x,y,dot,ierr)
       rdot = PetscRealPart(dot)
       write(6,100) rdot
  100  format('Result of inner product ',f10.4)

       call VecScale(x,two,ierr)
       call VecNorm(x,NORM_2,norm,ierr)
       write(6,110) norm
  110  format('Result of scaling ',f10.4)

       call VecCopy(x,w,ierr)
       call VecNorm(w,NORM_2,norm,ierr)
       write(6,120) norm
  120  format('Result of copy ',f10.4)

       call VecAXPY(y,three,x,ierr)
       call VecNorm(y,NORM_2,norm,ierr)
       write(6,130) norm
```

```
  130  format('Result of axpy ',f10.4)

       call VecDestroy(x,ierr)
       call VecDestroy(y,ierr)
       call VecDestroy(w,ierr)
       call PetscFinalize(ierr)
       end
```

Figure 15: Sample Fortran Program: Using PETSc Vectors

```
!
!
!  Description:  Illustrates the use of VecSetValues() to set
!  multiple values at once; demonstrates VecGetArray().
!
!/*T
!   Concepts: vectors^assembling;
!   Concepts: vectors^arrays of vectors;
!   Processors: 1
!T*/
! -----------------------------------------------------------------------

      program main
      implicit none

! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!                   Include files
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!
!  The following include statements are required for Fortran programs
!  that use PETSc vectors:
!     petscsys.h       - base PETSc routines
!     petscvec.h    - vectors

#include <petsc/finclude/petscsys.h>
#include <petsc/finclude/petscvec.h>

! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!                  Macro definitions
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!
!  Macros to make clearer the process of setting values in vectors and
!  getting values from vectors.
!
!   - The element xx_a(ib) is element ib+1 in the vector x
!   - Here we add 1 to the base array index to facilitate the use of
!     conventional Fortran 1-based array indexing.
!
#define xx_a(ib)  xx_v(xx_i + (ib))
#define yy_a(ib)  yy_v(yy_i + (ib))

! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
```

```
!                Beginning of program
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -

       PetscScalar xwork(6)
       PetscScalar xx_v(1),yy_v(1)
       PetscInt     i,n,loc(6),isix
       PetscErrorCode ierr
       PetscOffset xx_i,yy_i
       Vec         x,y

       call PetscInitialize(PETSC_NULL_CHARACTER,ierr)
       n = 6
       isix = 6

!  Create initial vector and duplicate it

       call VecCreateSeq(PETSC_COMM_SELF,n,x,ierr)
       call VecDuplicate(x,y,ierr)

!  Fill work arrays with vector entries and locations.  Note that
!  the vector indices are 0-based in PETSc (for both Fortran and
!  C vectors)

       do 10 i=1,n
          loc(i) = i-1
          xwork(i) = 10.0*i
  10   continue

!  Set vector values.  Note that we set multiple entries at once.
!  Of course, usually one would create a work array that is the
!  natural size for a particular problem (not one that is as long
!  as the full vector).

       call VecSetValues(x,isix,loc,xwork,INSERT_VALUES,ierr)

!  Assemble vector

       call VecAssemblyBegin(x,ierr)
       call VecAssemblyEnd(x,ierr)

!  View vector
       call PetscObjectSetName(x, 'initial vector:',ierr)
       call VecView(x,PETSC_VIEWER_STDOUT_SELF,ierr)
       call VecCopy(x,y,ierr)

!  Get a pointer to vector data.
!    - For default PETSc vectors, VecGetArray() returns a pointer to
!      the data array.  Otherwise, the routine is implementation dependent.
!    - You MUST call VecRestoreArray() when you no longer need access to
!      the array.
!    - Note that the Fortran interface to VecGetArray() differs from the
!      C version.  See the users manual for details.

       call VecGetArray(x,xx_v,xx_i,ierr)
       call VecGetArray(y,yy_v,yy_i,ierr)
```

```
!  Modify vector data

       do 30 i=1,n
          xx_a(i) = 100.0*i
          yy_a(i) = 1000.0*i
  30   continue

!  Restore vectors

       call VecRestoreArray(x,xx_v,xx_i,ierr)
       call VecRestoreArray(y,yy_v,yy_i,ierr)

!  View vectors
       call PetscObjectSetName(x, 'new vector 1:',ierr)
       call VecView(x,PETSC_VIEWER_STDOUT_SELF,ierr)

       call PetscObjectSetName(y, 'new vector 2:',ierr)
       call VecView(y,PETSC_VIEWER_STDOUT_SELF,ierr)

!  Free work space.  All PETSc objects should be destroyed when they
!  are no longer needed.

       call VecDestroy(x,ierr)
       call VecDestroy(y,ierr)
       call PetscFinalize(ierr)
       end
```

Figure 16: Sample Fortran Program: Using VecSetValues() and VecGetArray()

```
!
!
      program main
#include <petsc/finclude/petscsys.h>
#include <petsc/finclude/petscdraw.h>
!
!  This example demonstrates basic use of the Fortran interface for
!  PetscDraw routines.
!
      PetscDraw         draw
      PetscDrawLG       lg
      PetscDrawAxis     axis
      PetscErrorCode    ierr
      PetscBool         flg
      integer           x,y,width,height
      PetscScalar       xd,yd
      PetscInt          i,n,w,h

      n      = 15
      x      = 0
      y      = 0
      w      = 400
      h      = 300
```

```
      call PetscInitialize(PETSC_NULL_CHARACTER,ierr)

!  GetInt requires a PetscInt so have to do this ugly setting
      call PetscOptionsGetInt(PETSC_NULL_OBJECT,PETSC_NULL_CHARACTER,         &
     &                        '-width',w, flg,ierr)
      width = int(w)
      call PetscOptionsGetInt(PETSC_NULL_OBJECT,PETSC_NULL_CHARACTER,         &
     &                        '-height',h,flg,ierr)
      height = int(h)
      call PetscOptionsGetInt(PETSC_NULL_OBJECT,PETSC_NULL_CHARACTER,         &
     &                        '-n',n,flg,ierr)

      call PetscDrawCreate(PETSC_COMM_WORLD,PETSC_NULL_CHARACTER,        &
     &               PETSC_NULL_CHARACTER,x,y,width,height,draw,ierr)
      call PetscDrawSetFromOptions(draw,ierr)

      call PetscDrawLGCreate(draw,1,lg,ierr)
      call PetscDrawLGGetAxis(lg,axis,ierr)
      call PetscDrawAxisSetColors(axis,PETSC_DRAW_BLACK,PETSC_DRAW_RED, &
     &     PETSC_DRAW_BLUE,ierr)
      call PetscDrawAxisSetLabels(axis,'toplabel','xlabel','ylabel',    &
     &     ierr)

      do 10, i=0,n-1
        xd = i - 5.0
        yd = xd*xd
        call PetscDrawLGAddPoint(lg,xd,yd,ierr)
 10   continue

      call PetscDrawLGSetUseMarkers(lg,PETSC_TRUE,ierr)
      call PetscDrawLGDraw(lg,ierr)

      call PetscSleep(10,ierr)

      call PetscDrawLGDestroy(lg,ierr)
      call PetscDrawDestroy(draw,ierr)
      call PetscFinalize(ierr)
      end
```

Figure 17: Sample Fortran Program: Using PETSc PetscDraw Routines

```
!
!
!  Description: Uses the Newton method to solve a two-variable system.
!
!/*T
!  Concepts: SNES^basic uniprocessor example
!  Processors: 1
!T*/
!
! -----------------------------------------------------------------------
```

```
      program main
      implicit none

! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!                   Include files
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!
!  The following include statements are generally used in SNES Fortran
!  programs:
!     petscsys.h       - base PETSc routines
!     petscvec.h    - vectors
!     petscmat.h    - matrices
!     petscksp.h    - Krylov subspace methods
!     petscpc.h     - preconditioners
!     petscsnes.h   - SNES interface
!  Other include statements may be needed if using additional PETSc
!  routines in a Fortran program, e.g.,
!     petscviewer.h - viewers
!     petscis.h     - index sets
!
#include <petsc/finclude/petscsys.h>
#include <petsc/finclude/petscvec.h>
#include <petsc/finclude/petscmat.h>
#include <petsc/finclude/petscksp.h>
#include <petsc/finclude/petscpc.h>
#include <petsc/finclude/petscsnes.h>
!
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!                   Variable declarations
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!
!  Variables:
!     snes        - nonlinear solver
!     ksp        - linear solver
!     pc          - preconditioner context
!     ksp         - Krylov subspace method context
!     x, r        - solution, residual vectors
!     J           - Jacobian matrix
!     its         - iterations for convergence
!
      SNES     snes
      PC       pc
      KSP      ksp
      Vec      x,r
      Mat      J
      SNESLineSearch linesearch
      PetscErrorCode  ierr
      PetscInt its,i2,i20
      PetscMPIInt size,rank
      PetscScalar   pfive
      PetscReal   tol
      PetscBool   setls

!  Note: Any user-defined Fortran routines (such as FormJacobian)
!  MUST be declared as external.
```

```
      external FormFunction, FormJacobian, MyLineSearch

! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
- -
!                  Macro definitions
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
- -
!
!  Macros to make clearer the process of setting values in vectors and
!  getting values from vectors.  These vectors are used in the routines
!  FormFunction() and FormJacobian().
!   - The element lx_a(ib) is element ib in the vector x
!
#define lx_a(ib) lx_v(lx_i + (ib))
#define lf_a(ib) lf_v(lf_i + (ib))
!
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!                Beginning of program
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -

      call PetscInitialize(PETSC_NULL_CHARACTER,ierr)
      call MPI_Comm_size(PETSC_COMM_WORLD,size,ierr)
      call MPI_Comm_rank(PETSC_COMM_WORLD,rank,ierr)
      if (size .ne. 1) then
         if (rank .eq. 0) then
            write(6,*) 'This is a uniprocessor example only!'
         endif
         SETERRQ(PETSC_COMM_SELF,1,' ',ierr)
      endif

      i2  = 2
      i20 = 20
! - - - - - - - - - - -- - - - - - - - - - - - - - - - - - - - - - - - - - -
!  Create nonlinear solver context
! - - - - - - - - - - -- - - - - - - - - - - - - - - - - - - - - - - - - - -

      call SNESCreate(PETSC_COMM_WORLD,snes,ierr)

! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!  Create matrix and vector data structures; set corresponding routines
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -

!  Create vectors for solution and nonlinear function

      call VecCreateSeq(PETSC_COMM_SELF,i2,x,ierr)
      call VecDuplicate(x,r,ierr)

!  Create Jacobian matrix data structure

      call MatCreate(PETSC_COMM_SELF,J,ierr)
      call MatSetSizes(J,PETSC_DECIDE,PETSC_DECIDE,i2,i2,ierr)
      call MatSetFromOptions(J,ierr)
      call MatSetUp(J,ierr)
```

```
!  Set function evaluation routine and vector

      call SNESSetFunction(snes,r,FormFunction,PETSC_NULL_OBJECT,ierr)

!  Set Jacobian matrix data structure and Jacobian evaluation routine

      call SNESSetJacobian(snes,J,J,FormJacobian,PETSC_NULL_OBJECT,     &
     &     ierr)

! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!  Customize nonlinear solver; set runtime options
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -

!  Set linear solver defaults for this problem. By extracting the
!  KSP, KSP, and PC contexts from the SNES context, we can then
!  directly call any KSP, KSP, and PC routines to set various options.

      call SNESGetKSP(snes,ksp,ierr)
      call KSPGetPC(ksp,pc,ierr)
      call PCSetType(pc,PCNONE,ierr)
      tol = 1.e-4
      call KSPSetTolerances(ksp,tol,PETSC_DEFAULT_REAL,                 &
     &                     PETSC_DEFAULT_REAL,i20,ierr)

!  Set SNES/KSP/KSP/PC runtime options, e.g.,
!      -snes_view -snes_monitor -ksp_type <ksp> -pc_type <pc>
!  These options will override those specified above as long as
!  SNESSetFromOptions() is called _after_ any other customization
!  routines.


      call SNESSetFromOptions(snes,ierr)

      call PetscOptionsHasName(PETSC_NULL_OBJECT,PETSC_NULL_CHARACTER,   &
     &                         '-setls',setls,ierr)

      if (setls) then
        call SNESGetLineSearch(snes, linesearch, ierr)
        call SNESLineSearchSetType(linesearch, 'shell', ierr)
        call SNESLineSearchShellSetUserFunc(linesearch, MyLineSearch,   &
     &PETSC_NULL_OBJECT, ierr)
      endif

! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!  Evaluate initial guess; then solve nonlinear system
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -

!  Note: The user should initialize the vector, x, with the initial guess
!  for the nonlinear solver prior to calling SNESSolve().  In particular,
!  to employ an initial guess of zero, the user should explicitly set
!  this vector to zero by calling VecSet().

      pfive = 0.5
      call VecSet(x,pfive,ierr)
      call SNESSolve(snes,PETSC_NULL_OBJECT,x,ierr)
```

```
      call SNESGetIterationNumber(snes,its,ierr);
      if (rank .eq. 0) then
         write(6,100) its
      endif
  100 format('Number of SNES iterations = ',i5)

! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -
!  Free work space.  All PETSc objects should be destroyed when they
!  are no longer needed.
! - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - - -

      call VecDestroy(x,ierr)
      call VecDestroy(r,ierr)
      call MatDestroy(J,ierr)
      call SNESDestroy(snes,ierr)
      call PetscFinalize(ierr)
      end
!
! ---------------------------------------------------------------------
!
!  FormFunction - Evaluates nonlinear function, F(x).
!
!  Input Parameters:
!  snes - the SNES context
!  x - input vector
!  dummy - optional user-defined context (not used here)
!
!  Output Parameter:
!  f - function vector
!
      subroutine FormFunction(snes,x,f,dummy,ierr)
      implicit none

#include <petsc/finclude/petscsys.h>
#include <petsc/finclude/petscvec.h>
#include <petsc/finclude/petscsnes.h>

      SNES     snes
      Vec      x,f
      PetscErrorCode ierr
      integer dummy(*)

!  Declarations for use with local arrays

      PetscScalar  lx_v(2),lf_v(2)
      PetscOffset  lx_i,lf_i

!  Get pointers to vector data.
!    - For default PETSc vectors, VecGetArray() returns a pointer to
!      the data array.  Otherwise, the routine is implementation dependent.
!    - You MUST call VecRestoreArray() when you no longer need access to
!      the array.
!    - Note that the Fortran interface to VecGetArray() differs from the
!      C version.  See the Fortran chapter of the users manual for details.
```

```
      call VecGetArrayRead(x,lx_v,lx_i,ierr)
      call VecGetArray(f,lf_v,lf_i,ierr)

!  Compute function

      lf_a(1) = lx_a(1)*lx_a(1)                                          &
     &          + lx_a(1)*lx_a(2) - 3.0
      lf_a(2) = lx_a(1)*lx_a(2)                                          &
     &          + lx_a(2)*lx_a(2) - 6.0

!  Restore vectors

      call VecRestoreArrayRead(x,lx_v,lx_i,ierr)
      call VecRestoreArray(f,lf_v,lf_i,ierr)

      return
      end

! ---------------------------------------------------------------------
!
!  FormJacobian - Evaluates Jacobian matrix.
!
!  Input Parameters:
!  snes - the SNES context
!  x - input vector
!  dummy - optional user-defined context (not used here)
!
!  Output Parameters:
!  A - Jacobian matrix
!  B - optionally different preconditioning matrix
!  flag - flag indicating matrix structure
!
      subroutine FormJacobian(snes,X,jac,B,dummy,ierr)
      implicit none

#include <petsc/finclude/petscsys.h>
#include <petsc/finclude/petscvec.h>
#include <petsc/finclude/petscmat.h>
#include <petsc/finclude/petscpc.h>
#include <petsc/finclude/petscsnes.h>

      SNES         snes
      Vec          X
      Mat          jac,B
      PetscScalar  A(4)
      PetscErrorCode ierr
      PetscInt idx(2),i2
      integer dummy(*)

!  Declarations for use with local arrays

      PetscScalar lx_v(1)
      PetscOffset lx_i

!  Get pointer to vector data
```

```
      i2 = 2
      call VecGetArrayRead(x,lx_v,lx_i,ierr)

!  Compute Jacobian entries and insert into matrix.
!   - Since this is such a small problem, we set all entries for
!     the matrix at once.
!   - Note that MatSetValues() uses 0-based row and column numbers
!     in Fortran as well as in C (as set here in the array idx).

      idx(1) = 0
      idx(2) = 1
      A(1) = 2.0*lx_a(1) + lx_a(2)
      A(2) = lx_a(1)
      A(3) = lx_a(2)
      A(4) = lx_a(1) + 2.0*lx_a(2)
      call MatSetValues(jac,i2,idx,i2,idx,A,INSERT_VALUES,ierr)

!  Restore vector

      call VecRestoreArrayRead(x,lx_v,lx_i,ierr)

!  Assemble matrix

      call MatAssemblyBegin(jac,MAT_FINAL_ASSEMBLY,ierr)
      call MatAssemblyEnd(jac,MAT_FINAL_ASSEMBLY,ierr)

      return
      end


      subroutine MyLineSearch(linesearch, lctx, ierr)
#include <petsc/finclude/petscsys.h>
#include <petsc/finclude/petscvec.h>
#include <petsc/finclude/petscmat.h>
#include <petsc/finclude/petscksp.h>
#include <petsc/finclude/petscpc.h>
#include <petsc/finclude/petscsnes.h>

      SNES              snes
      integer           lctx
      Vec               x, f,g, y, w
      PetscReal         ynorm,gnorm,xnorm
      PetscBool         flag
      PetscErrorCode    ierr

      PetscScalar       mone

      mone = -1.0
      call SNESLineSearchGetSNES(linesearch, snes, ierr)
      call SNESLineSearchGetVecs(linesearch, x, f, y, w, g, ierr)
      call VecNorm(y,NORM_2,ynorm,ierr)
      call VecAXPY(x,mone,y,ierr)
      call SNESComputeFunction(snes,x,f,ierr)
      call VecNorm(f,NORM_2,gnorm,ierr)
```

```
      call VecNorm(x,NORM_2,xnorm,ierr)
      call VecNorm(y,NORM_2,ynorm,ierr)
      call SNESLineSearchSetNorms(linesearch, xnorm, gnorm, ynorm,      &
     & ierr)
      flag = PETSC_FALSE
      return
      end
```

Figure 18: Sample Fortran Program: Using PETSc Nonlinear Solvers

Part III

Additional Information

Chapter 13

Profiling

PETSc includes a consistent, lightweight scheme to allow the profiling of application programs. The PETSc routines automatically log performance data if certain options are specified at runtime. The user can also log information about application codes for a complete picture of performance. In addition, as described in Section 13.1.1, PETSc provides a mechanism for printing informative messages about computations. Section 13.1 introduces the various profiling options in PETSc, while the remainder of the chapter focuses on details such as monitoring application codes and tips for accurate profiling.

13.1 Basic Profiling Information

If an application code and the PETSc libraries have been configured with `--with-log=1`, the default, then various kinds of profiling of code between calls to PetscInitialize() and PetscFinalize() can be activated at runtime. The profiling options include the following:

- `-log_summary` - Prints an ASCII version of performance data at program's conclusion. These statistics are comprehensive and concise and require little overhead; thus, `-log_summary` is intended as the primary means of monitoring the performance of PETSc codes.

- `-info [infofile]` - Prints verbose information about code to stdout or an optional file. This option provides details about algorithms, data structures, etc. Since the overhead of printing such output slows a code, this option should not be used when evaluating a program's performance.

- `-log_trace [logfile]` - Traces the beginning and ending of all PETSc events. This option, which can be used in conjunction with `-info`, is useful to see where a program is hanging without running in the debugger.

As discussed in Section 13.1.3, additional profiling can be done with MPE.

13.1.1 Interpreting `-log_summary` Output: The Basics

As shown in Figure 7 (in Part I), the option `-log_summary` activates printing of profile data to standard output at the conclusion of a program. Profiling data can also be printed at any time within a program by calling PetscLogView().

We print performance data for each routine, organized by PETSc libraries, followed by any user-defined events (discussed in Section 13.2). For each routine, the output data include the maximum time and floating point operation (flop) rate over all processes. Information about parallel performance is also included, as discussed in the following section.

For the purpose of PETSc floating point operation counting, we define one *flop* as one operation of any of the following types: multiplication, division, addition, or subtraction. For example, one VecAXPY() operation, which computes $y = \alpha x + y$ for vectors of length N, requires $2N$ flops (consisting of N additions and N multiplications). Bear in mind that flop rates present only a limited view of performance, since memory loads and stores are the real performance barrier.

For simplicity, the remainder of this discussion focuses on interpreting profile data for the KSP library, which provides the linear solvers at the heart of the PETSc package. Recall the hierarchical organization of the PETSc library, as shown in Figure 1. Each KSP solver is composed of a PC (preconditioner) and a KSP (Krylov subspace) part, which are in turn built on top of the Mat (matrix) and Vec (vector) modules. Thus, operations in the KSP module are composed of lower-level operations in these packages. Note also that the nonlinear solvers library, SNES, is build on top of the KSP module, and the timestepping library, TS, is in turn built on top of SNES.

We briefly discuss interpretation of the sample output in Figure 7, which was generated by solving a linear system on one process using restarted GMRES and ILU preconditioning. The linear solvers in KSP consist of two basic phases, KSPSetUp() and KSPSolve(), each of which consists of a variety of actions, depending on the particular solution technique. For the case of using the PCILU preconditioner and KSPGMRES Krylov subspace method, the breakdown of PETSc routines is listed below. As indicated by the levels of indentation, the operations in KSPSetUp() include all of the operations within PCSetUp(), which in turn include MatILUFactor(), and so on.

- KSPSetUp - Set up linear solver
  - PCSetUp - Set up preconditioner
    - MatILUFactor - Factor preconditioning matrix
      - MatILUFactorSymbolic - Symbolic factorization phase
      - MatLUFactorNumeric - Numeric factorization phase
- KSPSolve - Solve linear system
  - PCApply - Apply preconditioner
    - MatSolve - Forward/backward triangular solves
  - KSPGMRESOrthog - Orthogonalization in GMRES
    - VecDot or VecMDot - Inner products
  - MatMult - Matrix-vector product
  - MatMultAdd - Matrix-vector product + vector addition
  - VecScale, VecNorm, VecAXPY, VecCopy, ...

The summaries printed via `-log_summary` reflect this routine hierarchy. For example, the performance summaries for a particular high-level routine such as KSPSolve include all of the operations accumulated in the lower-level components that make up the routine.

Admittedly, we do not currently present the output with `-log_summary` so that the hierarchy of PETSc operations is completely clear, primarily because we have not determined a clean and uniform way to do so throughout the library. Improvements may follow. However, for a particular problem, the user should generally have an idea of the basic operations that are required for its implementation (e.g., which operations are performed when using GMRES and ILU, as described above), so that interpreting the `-log_summary` data should be relatively straightforward.

13.1.2 Interpreting `-log_summary` Output: Parallel Performance

We next discuss performance summaries for parallel programs, as shown within Figures 19 and 20, which present the combined output generated by the `-log_summary` option. The program that generated this

```
mpiexec -n 4 ./ex10 -f0 medium -f1 arco6 -ksp_gmres_classicalgramschmidt -log_summary -mat_type baij \
           -matload_block_size 3 -pc_type bjacobi -options_left

Number of iterations =  19
Residual norm = 7.7643e-05
Number of iterations =  55
Residual norm = 6.3633e-01

---------------------------------------------- PETSc Performance Summary: ----------------------------------------------

ex10 on a rs6000 named p039 with 4 processors, by mcinnes Wed Jul 24 16:30:22 1996

                         Max         Min        Avg        Total
Time (sec):           3.289e+01      1.0   3.288e+01
Objects:              1.130e+02      1.0   1.130e+02
Flops:                2.195e+08      1.0   2.187e+08   8.749e+08
Flops/sec:            6.673e+06      1.0               2.660e+07
MPI Messages:         2.205e+02      1.4   1.928e+02   7.710e+02
MPI Message Lengths:  7.862e+06      2.5   5.098e+06   2.039e+07
MPI Reductions:       1.850e+02      1.0

Summary of Stages:  ---- Time ------  ----- Flops -------  -- Messages -- -- Message-lengths -- Reductions -
                      Avg      %Total    Avg     %Total   counts   %Total    avg      %Total   counts  %Total
 0:  Load System 0: 1.191e+00    3.6% 3.980e+06   0.5%  3.80e+01    4.9%  6.102e+04   0.3%  1.80e+01   9.7%
 1:    KSPSetup 0:  6.328e-01    2.5% 1.479e+04   0.0%  0.00e+00    0.0%  0.000e+00   0.0%  0.00e+00   0.0%
 2:    KSPSolve 0:  2.269e-01    0.9% 1.340e+06   0.0%  1.52e+02   19.7%  9.405e+03   0.0%  3.90e+01  21.1%
 3:  Load System 1: 2.680e+01  107.3% 0.000e+00   0.0%  2.10e+01    2.7%  1.799e+07  88.2%  1.60e+01   8.6%
 4:    KSPSetup 1:  1.867e-01    0.7% 1.088e+08   2.3%  0.00e+00    0.0%  0.000e+00   0.0%  0.00e+00   0.0%
 5:    KSPSolve 1:  3.831e+00   15.3% 2.217e+08  97.1%  5.60e+02   72.6%  2.333e+06  11.4%  1.12e+02  60.5%

------------------------------------------------------------------------------------------------------------------------

.... [Summary of various phases, see part II below] ...

------------------------------------------------------------------------------------------------------------------------

Memory usage is given in bytes:

Object Type      Creations   Destructions   Memory  Descendants' Mem.
Viewer                 5              5        0     0
Index set             10             10   127076     0
Vector                76             76  9152040     0
Vector Scatter         2              2   106220     0
Matrix                 8              8  9611488     5.59773e+06
Krylov Solver          4              4    33960     7.5966e+06
Preconditioner         4              4       16     9.49114e+06
KSP                    4              4        0     1.71217e+07
```

Figure 19: Profiling a PETSc Program: Part I - Overall Summary

data is `${PETSC_DIR}/src/ksp/ksp/examples/ex10.c`. The code loads a matrix and right-hand-side vector from a binary file and then solves the resulting linear system; the program then repeats this process for a second linear system. This particular case was run on four processors of an IBM SP, using restarted GMRES and the block Jacobi preconditioner, where each block was solved with ILU.

Figure 19 presents an overall performance summary, including times, floating-point operations, computational rates, and message-passing activity (such as the number and size of messages sent and collective operations). Summaries for various user-defined stages of monitoring (as discussed in Section 13.3) are also given. Information about the various phases of computation then follow (as shown separately here in Figure 20). Finally, a summary of memory usage and object creation and destruction is presented.

We next focus on the summaries for the various phases of the computation, as given in the table within Figure 20. The summary for each phase presents the maximum times and flop rates over all processes, as well as the ratio of maximum to minimum times and flop rates for all processes. A ratio of approximately 1 indicates that computations within a given phase are well balanced among the processes; as the ratio increases, the balance becomes increasingly poor. Also, the total computational rate (in units of MFlops/sec) is given for each phase in the final column of the phase summary table.

$$\text{Total Mflop/sec} \;=\; 10^{-6} * (\text{sum of flops over all processors})/(\text{max time over all processors})$$

Note: Total computational rates < 1 MFlop are listed as 0 in this column of the phase summary table. Additional statistics for each phase include the total number of messages sent, the average message length, and the number of global reductions.

As discussed in the preceding section, the performance summaries for higher-level PETSc routines in-

```
mpiexec -n 4 ./ex10 -f0 medium -f1 arco6 -ksp_gmres_classicalgramschmidt -log_summary -mat_type baij \
            -matload_block_size 3 -pc_type bjacobi -options_left

---------------------------------------------- PETSc Performance Summary: ----------------------------------------------
.... [Overall summary, see part I] ...

Phase summary info:
   Count: number of times phase was executed
   Time and Flops/sec: Max - maximum over all processors
                       Ratio - ratio of maximum to minimum over all processors
   Mess: number of messages sent
   Avg. len: average message length
   Reduct: number of global reductions
   Global: entire computation
   Stage: optional user-defined stages of a computation. Set stages with PLogStagePush() and PLogStagePop().
      %T - percent time in this phase         %F - percent flops in this phase
      %M - percent messages in this phase     %L - percent message lengths in this phase
      %R - percent reductions in this phase
   Total Mflop/s: 10^6 * (sum of flops over all processors)/(max time over all processors)
------------------------------------------------------------------------------------------------------------------------
Phase               Count      Time (sec)       Flops/sec                         --- Global ---  --- Stage ---   Total
                             Max    Ratio      Max    Ratio  Mess  Avg len  Reduct %T %F %M %L %R  %T %F %M %L %R Mflop/s
------------------------------------------------------------------------------------------------------------------------
...

--- Event Stage 4: KSPSetUp 1

MatGetReordering       1  3.491e-03   1.0  0.0e+00   0.0  0.0e+00 0.0e+00 0.0e+00  0  0  0  0  0   2  0  0  0  0     0
MatILUFctrSymbol       1  6.970e-03   1.2  0.0e+00   0.0  0.0e+00 0.0e+00 0.0e+00  0  0  0  0  0   3  0  0  0  0     0
MatLUFactorNumer       1  1.829e-01   1.1  3.2e+07   1.1  0.0e+00 0.0e+00 0.0e+00  1  2  0  0  0  90 99  0  0  0   110
KSPSetUp               2  1.989e-01   1.1  2.9e+07   1.1  0.0e+00 0.0e+00 0.0e+00  1  2  0  0  0  99 99  0  0  0   102
PCSetUp                2  1.952e-01   1.1  2.9e+07   1.1  0.0e+00 0.0e+00 0.0e+00  1  2  0  0  0  97 99  0  0  0   104
PCSetUpOnBlocks        1  1.930e-01   1.1  3.0e+07   1.1  0.0e+00 0.0e+00 0.0e+00  1  2  0  0  0  96 99  0  0  0   105

--- Event Stage 5: KSPSolve 1

MatMult               56  1.199e+00   1.1  5.3e+07   1.0  1.1e+03 4.2e+03 0.0e+00  5 28 99 23  0  30 28 99 99  0   201
MatSolve              57  1.263e+00   1.0  4.7e+07   1.0  0.0e+00 0.0e+00 0.0e+00  5 27  0  0  0  33 28  0  0  0   187
VecNorm               57  1.528e-01   1.3  2.7e+07   1.3  0.0e+00 0.0e+00 2.3e+02  1  1  0  0 31   3  1  0  0 51    81
VecScale              57  3.347e-02   1.0  4.7e+07   1.0  0.0e+00 0.0e+00 0.0e+00  0  1  0  0  0   1  1  0  0  0   184
VecCopy                2  1.703e-03   1.1  0.0e+00   0.0  0.0e+00 0.0e+00 0.0e+00  0  0  0  0  0   0  0  0  0  0     0
VecSet                 3  2.098e-03   1.0  0.0e+00   0.0  0.0e+00 0.0e+00 0.0e+00  0  0  0  0  0   0  0  0  0  0     0
VecAXPY                3  3.247e-03   1.1  5.4e+07   1.1  0.0e+00 0.0e+00 0.0e+00  0  0  0  0  0   0  0  0  0  0   200
VecMDot               55  5.216e-01   1.2  9.8e+07   1.2  0.0e+00 0.0e+00 2.2e+02  2 20  0  0 30  12 20  0  0 49   327
VecMAXPY              57  6.997e-01   1.1  6.9e+07   1.1  0.0e+00 0.0e+00 0.0e+00  3 21  0  0  0  18 21  0  0  0   261
VecScatterBegin       56  4.534e-02   1.8  0.0e+00   0.0  1.1e+03 4.2e+03 0.0e+00  0  0 99 23  0   1  0 99 99  0     0
VecScatterEnd         56  2.095e-01   1.2  0.0e+00   0.0  0.0e+00 0.0e+00 0.0e+00  1  0  0  0  0   5  0  0  0  0     0
KSPSolve               1  3.832e+00   1.0  5.6e+07   1.0  1.1e+03 4.2e+03 4.5e+02 15 97 99 23 61  99 99 99 99 99   222
KSPGMRESOrthog        55  1.177e+00   1.1  7.9e+07   1.1  0.0e+00 0.0e+00 2.2e+02  4 39  0  0 30  29 40  0  0 49   290
PCSetUpOnBlocks        1  1.180e-05   1.1  0.0e+00   0.0  0.0e+00 0.0e+00 0.0e+00  0  0  0  0  0   0  0  0  0  0     0
PCApply               57  1.267e+00   1.0  4.7e+07   1.0  0.0e+00 0.0e+00 0.0e+00  5 27  0  0  0  33 28  0  0  0   186
------------------------------------------------------------------------------------------------------------------------
.... [Conclusion of overall summary, see part I] ...
```

Figure 20: Profiling a PETSc Program: Part II - Phase Summaries

clude the statistics for the lower levels of which they are made up. For example, the communication within matrix-vector products MatMult() consists of vector scatter operations, as given by the routines VecScatterBegin() and VecScatterEnd().

The final data presented are the percentages of the various statistics (time (`%T`), flops/sec (`%F`), messages(`%M`), average message length (`%L`), and reductions (`%R`)) for each event relative to the total computation and to any user-defined stages (discussed in Section 13.3). These statistics can aid in optimizing performance, since they indicate the sections of code that could benefit from various kinds of tuning. Chapter 14 gives suggestions about achieving good performance with PETSc codes.

13.1.3 Using `-log_mpe` with Jumpshot

It is also possible to use the *Jumpshot*) package [16] to visualize PETSc events. This package comes with the MPE software, which is part of the MPICH [14] implementation of MPI. The option

-log_mpe [logfile]

creates a logfile of events appropriate for viewing with *Jumpshot*. The user can either use the default logging file, `mpe.log`, or specify an optional name via `logfile`.

By default, not all PETSc events are logged with MPE. For example, since MatSetValues() may be called thousands of times in a program, by default its calls are not logged with MPE. To activate MPE logging of a particular event, one should use the command

PetscLogEventMPEActivate(int event);

To deactivate logging of an event for MPE, one should use

PetscLogEventMPEDeactivate(int event);

The `event` may be either a predefined PETSc event (as listed in the file `${PETSC_DIR}/include/petsclog.h`) or one obtained with PetscLogEventRegister() (as described in Section 13.2). These routines may be called as many times as desired in an application program, so that one could restrict MPE event logging only to certain code segments.

To see what events are logged by default, the user can view the source code; see the files `src/plot/src/plogmpe.c` and `include/petsclog.h`. A simple program and GUI interface to see the events that are predefined and their definition is being developed.

The user can also log MPI events. To do this, simply consider the PETSc application as any MPI application, and follow the MPI implementation's instructions for logging MPI calls. For example, when using MPICH, this merely required adding `-llmpich` to the library list *before* `-lmpich`.

13.2 Profiling Application Codes

PETSc automatically logs object creation, times, and floating-point counts for the library routines. Users can easily supplement this information by monitoring their application codes as well. The basic steps involved in logging a user-defined portion of code, called an *event*, are shown in the code fragment below:

```
#include ¡petsclog.h¿
int USER_EVENT;
PetscLogEventRegister("User event name",0,&USER_EVENT);
PetscLogEventBegin(USER_EVENT,0,0,0,0);
/* application code segment to monitor */
PetscLogFlops(number of flops for this code segment);
PetscLogEventEnd(USER_EVENT,0,0,0,0);
```

One must register the event by calling PetscLogEventRegister(), which assigns a unique integer to identify the event for profiling purposes:

PetscLogEventRegister(const char string[],PetscLogEvent *e);

Here `string` is a user-defined event name, and `color` is an optional user-defined event color (for use with *Jumpshot* logging); one should see the manual page for details. The argument returned in `e` should then be passed to the PetscLogEventBegin() and PetscLogEventEnd() routines.

Events are logged by using the pair

```
PetscLogEventBegin(int event,PetscObject o1,PetscObject o2,
PetscObject o3,PetscObject o4);
PetscLogEventEnd(int event,PetscObject o1,PetscObject o2,
PetscObject o3,PetscObject o4);
```

The four objects are the PETSc objects that are most closely associated with the event. For instance, in a matrix-vector product they would be the matrix and the two vectors. These objects can be omitted by specifying 0 for `o1` - `o4`. The code between these two routine calls will be automatically timed and logged as part of the specified event.

The user can log the number of floating-point operations for this segment of code by calling

PetscLogFlops(number of flops for this code segment);

between the calls to PetscLogEventBegin() and PetscLogEventEnd(). This value will automatically be added to the global flop counter for the entire program.

13.3 Profiling Multiple Sections of Code

By default, the profiling produces a single set of statistics for all code between the PetscInitialize() and PetscFinalize() calls within a program. One can independently monitor up to ten stages of code by switching among the various stages with the commands

PetscLogStagePush(PetscLogStage stage);
PetscLogStagePop();

where `stage` is an integer (0-9); see the manual pages for details. The command

PetscLogStageRegister(const char *name,PetscLogStage *stage)

allows one to associate a name with a stage; these names are printed whenever summaries are generated with `-log_summary` or PetscLogView(). The following code fragment uses three profiling stages within an program.

```
PetscInitialize(int *argc,char ***args,0,0);
/* stage 0 of code here */
PetscLogStageRegister("Stage 0 of Code",&stagenum0);
for(i=0; i¡ntimes; i++) {
   PetscLogStageRegister("Stage 1 of Code",&stagenum1);
   PetscLogStagePush(stagenum1);
   /* stage 1 of code here */
   PetscLogStagePop();
   PetscLogStageRegister("Stage 2 of Code",&stagenum2);
   PetscLogStagePush(stagenum2);
   /* stage 2 of code here */
   PetscLogStagePop();
} PetscFinalize();
```

Figures 19 and 20 show output generated by `-log_summary` for a program that employs several profiling stages. In particular, this program is subdivided into six stages: loading a matrix and right-hand-side vector from a binary file, setting up the preconditioner, and solving the linear system; this sequence is then repeated for a second linear system. For simplicity, Figure 20 contains output only for stages 4 and 5 (linear solve of the second system), which comprise the part of this computation of most interest to us in terms of performance monitoring. This code organization (solving a small linear system followed by a larger system) enables generation of more accurate profiling statistics for the second system by overcoming the often considerable overhead of paging, as discussed in Section 13.8.

13.4 Restricting Event Logging

By default, all PETSc operations are logged. To enable or disable the PETSc logging of individual events, one uses the commands

PetscLogEventActivate(int event);
PetscLogEventDeactivate(int event);

The `event` may be either a predefined PETSc event (as listed in the file `${PETSC_DIR}/include/petsclog.h`) or one obtained with PetscLogEventRegister() (as described in Section 13.2).

PETSc also provides routines that deactivate (or activate) logging for entire components of the library. Currently, the components that support such logging (de)activation are Mat (matrices), Vec (vectors), KSP (linear solvers, including KSP and PC), and SNES (nonlinear solvers):

```
PetscLogEventDeactivateClass(MAT_CLASSID);
PetscLogEventDeactivateClass(KSP_CLASSID); /* includes PC and KSP */
PetscLogEventDeactivateClass(VEC_CLASSID);
PetscLogEventDeactivateClass(SNES_CLASSID);
```

and

```
PetscLogEventActivateClass(MAT_CLASSID);
PetscLogEventActivateClass(KSP_CLASSID); /* includes PC and KSP */
PetscLogEventActivateClass(VEC_CLASSID);
PetscLogEventActivateClass(SNES_CLASSID);
```

Recall that the option `-log_all` produces extensive profile data, which can be a challenge for PETScView to handle due to the memory limitations of Tcl/Tk. Thus, one should generally use `-log_all` when running programs with a relatively small number of events or when disabling some of the events that occur many times in a code (e.g., VecSetValues(), MatSetValues()).

Section 13.1.3 gives information on the restriction of events in MPE logging.

13.5 Interpreting `-log_info` Output: Informative Messages

Users can activate the printing of verbose information about algorithms, data structures, etc. to the screen by using the option `-info` or by calling PetscInfoAllow(`PETSC_TRUE`). Such logging, which is used throughout the PETSc libraries, can aid the user in understanding algorithms and tuning program performance. For example, as discussed in Section 3.1.1, `-info` activates the printing of information about memory allocation during matrix assembly.

Application programmers can employ this logging as well, by using the routine

PetscInfo(void* obj,char *message,...)

where `obj` is the PETSc object associated most closely with the logging statement, `message`. For example, in the line search Newton methods, we use a statement such as

```
PetscInfo(snes,"Cubically determined step, lambda %g\n",lambda);
```

One can selectively turn off informative messages about any of the basic PETSc objects (e.g., Mat, SNES) with the command

PetscInfoDeactivateClass(int object_classid)

where `object_classid` is one of `MAT_CLASSID`, `SNES_CLASSID`, etc. Messages can be reactivated with the command

PetscInfoActivateClass(int object_classid)

Such deactivation can be useful when one wishes to view information about higher-level PETSc libraries (e.g., TS and SNES) without seeing all lower level data as well (e.g., Mat). One can deactivate events at runtime for matrix and linear solver libraries via `-info [no_mat, no_ksp]`.

13.6 Time

PETSc application programmers can access the wall clock time directly with the command

```
PetscLogDouble time;
PetscTime(&time);CHKERRQ(ierr);
```

which returns the current time in seconds since the epoch, and is commonly implemented with `MPI_Wtime`. A floating point number is returned in order to express fractions of a second. In addition, as discussed in Section 13.2, PETSc can automatically profile user-defined segments of code.

13.7 Saving Output to a File

All output from PETSc programs (including informative messages, profiling information, and convergence data) can be saved to a file by using the command line option `-history [filename]`. If no file name is specified, the output is stored in the file `${HOME}/.petschistory`. Note that this option only saves output printed with the PetscPrintf() and PetscFPrintf() commands, not the standard `printf()` and `fprintf()` statements.

13.8 Accurate Profiling: Overcoming the Overhead of Paging

One factor that often plays a significant role in profiling a code is paging by the operating system. Generally, when running a program only a few pages required to start it are loaded into memory rather than the entire executable. When the execution proceeds to code segments that are not in memory, a pagefault occurs, prompting the required pages to be loaded from the disk (a very slow process). This activity distorts the results significantly. (The paging effects are noticeable in the log files generated by `-log_mpe`, which is described in Section 13.1.3.)

To eliminate the effects of paging when profiling the performance of a program, we have found an effective procedure is to run the **exact same code** on a small dummy problem before running it on the actual problem of interest. We thus ensure that all code required by a solver is loaded into memory during solution of the small problem. When the code proceeds to the actual (larger) problem of interest, all required pages have already been loaded into main memory, so that the performance numbers are not distorted.

When this procedure is used in conjunction with the user-defined stages of profiling described in Section 13.3, we can focus easily on the problem of interest. For example, we used this technique in the program `${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex10.c` to generate the timings within Figures 19 and 20. In this case, the profiled code of interest (solving the linear system for the larger problem) occurs within event stages 4 and 5. Section 13.1.2 provides details about interpreting such profiling data.

In particular, the macros

PetscPreLoadBegin(PetscBool ,char* stagename),
PetscPreLoadStage(char *stagename),

and

PetscPreLoadEnd()

can be used to easily convert a regular PETSc program to one that uses preloading. The command line options `-preload true` and `-preload false` may be used to turn on and off preloading at run time for PETSc programs that use these macros.

Chapter 14

Hints for Performance Tuning

This chapter presents some tips on achieving good performance within PETSc codes. We urge users to read these hints before evaluating the performance of PETSc application codes.

14.1 Compiler Options

Code configured with `--with-debugging=0` faster than that the default debugging version, so we recommend using one of the optimized versions of code when evaluating performance.

14.2 Profiling

Users should not spend time optimizing a code until after having determined where it spends the bulk of its time on realistically sized problems. As discussed in detail in Chapter 13, the PETSc routines automatically log performance data if certain runtime options are specified. We briefly highlight usage of these features below.

- Run the code with the option `-log_summary` to print a performance summary for various phases of the code.
- Run the code with the option `-log_mpe [logfilename]`, which creates a logfile of events suitable for viewing with Jumpshot (part of MPICH).

14.3 Aggregation

Performing operations on chunks of data rather than a single element at a time can significantly enhance performance.

- Insert several (many) elements of a matrix or vector at once, rather than looping and inserting a single value at a time. In order to access elements in of vector repeatedly, employ VecGetArray() to allow direct manipulation of the vector elements.
- When possible, use VecMDot() rather than a series of calls to VecDot().

14.4 Efficient Memory Allocation

14.4.1 Sparse Matrix Assembly

Since the process of dynamic memory allocation for sparse matrices is inherently very expensive, accurate preallocation of memory is crucial for efficient sparse matrix assembly. One should use the matrix creation routines for particular data structures, such as MatCreateSeqAIJ() and MatCreateAIJ() for compressed, sparse row formats, instead of the generic MatCreate() routine. For problems with multiple degrees of freedom per node, the block, compressed, sparse row formats, created by MatCreateSeqBAIJ() and MatCreateBAIJ(), can significantly enhance performance. Section 3.1.1 includes extensive details and examples regarding preallocation.

14.4.2 Sparse Matrix Factorization

When symbolically factoring an AIJ matrix, PETSc has to guess how much fill there will be. Careful use of the fill parameter in the MatILUInfo structure when calling MatLUFactorSymbolic() or MatILUFactorSymbolic() can reduce greatly the number of mallocs and copies required, and thus greatly improve the performance of the factorization. One way to determine a good value for f is to run a program with the option `-info`. The symbolic factorization phase will then print information such as

Info:MatILUFactorSymbolic_AIJ:Realloc 12 Fill ratio:given 1 needed 2.16423

This indicates that the user should have used a fill estimate factor of about 2.17 (instead of 1) to prevent the 12 required mallocs and copies. The command line option

-pc_ilu_fill 2.17

will cause PETSc to preallocate the correct amount of space for incomplete (ILU) factorization. The corresponding option for direct (LU) factorization is `-pc_factor_fill <fill_amount\trl{>}`.

14.4.3 PetscMalloc() Calls

Users should employ a reasonable number of PetscMalloc() calls in their codes. Hundreds or thousands of memory allocations may be appropriate; however, if tens of thousands are being used, then reducing the number of PetscMalloc() calls may be warranted. For example, reusing space or allocating large chunks and dividing it into pieces can produce a significant savings in allocation overhead. Section 14.5 gives details.

14.5 Data Structure Reuse

Data structures should be reused whenever possible. For example, if a code often creates new matrices or vectors, there often may be a way to reuse some of them. Very significant performance improvements can be achieved by reusing matrix data structures with the same nonzero pattern. If a code creates thousands of matrix or vector objects, performance will be degraded. For example, when solving a nonlinear problem or timestepping, reusing the matrices and their nonzero structure for many steps when appropriate can make the code run significantly faster.

A simple technique for saving work vectors, matrices, etc. is employing a user-defined context. In C and C++ such a context is merely a structure in which various objects can be stashed; in Fortran a user context can be an integer array that contains both parameters and pointers to PETSc objects. See `${PETSC_DIR}/snes/examples/tutorials/ex5.c` and `${PETSC_DIR}/snes/examples/tutorials/ex5f.F` for examples of user-defined application contexts in C and Fortran, respectively.

14.6 Numerical Experiments

PETSc users should run a variety of tests. For example, there are a large number of options for the linear and nonlinear equation solvers in PETSc, and different choices can make a *very* big difference in convergence rates and execution times. PETSc employs defaults that are generally reasonable for a wide range of problems, but clearly these defaults cannot be best for all cases. Users should experiment with many combinations to determine what is best for a given problem and customize the solvers accordingly.

- Use the options `-snes_view`, `-ksp_view`, etc. (or the routines KSPView(), SNESView(), etc.) to view the options that have been used for a particular solver.
- Run the code with the option `-help` for a list of the available runtime commands.
- Use the option `-info` to print details about the solvers' operation.
- Use the PETSc monitoring discussed in Chapter 13 to evaluate the performance of various numerical methods.

14.7 Tips for Efficient Use of Linear Solvers

As discussed in Chapter 4, the default linear solvers are

- uniprocess: GMRES(30) with ILU(0) preconditioning
- multiprocess: GMRES(30) with block Jacobi preconditioning, where there is 1 block per process, and each block is solved with ILU(0)

One should experiment to determine alternatives that may be better for various applications. Recall that one can specify the KSP methods and preconditioners at runtime via the options:

-ksp_type <ksp_name> -pc_type <pc_name>

One can also specify a variety of runtime customizations for the solvers, as discussed throughout the manual.

In particular, note that the default restart parameter for GMRES is 30, which may be too small for some large-scale problems. One can alter this parameter with the option `-ksp_gmres_restart` <restart> or by calling KSPGMRESSetRestart(). Section 4.3 gives information on setting alternative GMRES orthogonalization routines, which may provide much better parallel performance.

14.8 Detecting Memory Allocation Problems

PETSc provides a number of tools to aid in detection of problems with memory allocation, including leaks and use of uninitialized space. We briefly describe these below.

- The PETSc memory allocation (which collects statistics and performs error checking), is employed by default for codes compiled in a debug-mode (configured with `--with-debugging=1`). PETSc memory allocation can be activated for optimized-mode (configured with `--with-debugging=0`) using the option `-malloc`. The option `-malloc=0` forces the use of conventional memory allocation when debugging is enabled. When running timing tests, one should build libraries in optimized-mode.

- When the PETSc memory allocation routines are used, the option `-malloc_dump` will print a list of unfreed memory at the conclusion of a program. If all memory has been freed, only a message stating the maximum allocated space will be printed. However, if some memory remains unfreed, this information will be printed. Note that the option `-malloc_dump` merely activates a call to PetscMallocDump() during `PetscFinalize()` the user can also call PetscMallocDump() elsewhere in a program.

- Another useful option for use with PETSc memory allocation routines is `-malloc_log`, which activates logging of all calls to malloc and reports memory usage, including all Fortran arrays. This option provides a more complete picture than `-malloc_dump` for codes that employ Fortran with hardwired arrays. The option `-malloc_log` activates logging by calling PetscMallocSetDumpLog() in `PetscInitialize()` and then prints the log by calling PetscMallocDumpLog() in PetscFinalize(). The user can also call these routines elsewhere in a program. When finer granularity is desired, the user should call PetscMallocGetCurrentUsage() and PetscMallocGetMaximumUsage() for memory allocated by PETSc, or PetscMemoryGetCurrentUsage() and PetscMemoryGetMaximumUsage() for the total memory used by the program. Note that PetscMemorySetGetMaximumUsage() must be called before PetscMemoryGetMaximumUsage() (typically at the beginning of the program).

14.9 System-Related Problems

The performance of a code can be affected by a variety of factors, including the cache behavior, other users on the machine, etc. Below we briefly describe some common problems and possibilities for overcoming them.

- **Problem too large for physical memory size**: When timing a program, one should always leave at least a ten percent margin between the total memory a process is using and the physical size of the machine's memory. One way to estimate the amount of memory used by given process is with the UNIX `getrusage` system routine. Also, the PETSc option `-log_summary` prints the amount of memory used by the basic PETSc objects, thus providing a lower bound on the memory used. Another useful option is `-malloc_log` which reports all memory, including any Fortran arrays in an application code.

- **Effects of other users**: If other users are running jobs on the same physical processor nodes on which a program is being profiled, the timing results are essentially meaningless.

- **Overhead of timing routines on certain machines**: On certain machines, even calling the system clock in order to time routines is slow; this skews all of the flop rates and timing results. The file `${PETSC_DIR}/src/benchmarks/\href{http://www.mcs.anl.gov/petsc/petsc-3.7/docs/manualpages/Sys/PetscTime.html#PetscTime}{PetscTime}\findex{PetscTime}.c` contains a simple test problem that will approximate the amount of time required to get the current time in a running program. On good systems it will on the order of 1.e-6 seconds or less.

- **Problem too large for good cache performance**: Certain machines with lower memory bandwidths (slow memory access) attempt to compensate by having a very large cache. Thus, if a significant portion of an application fits within the cache, the program will achieve very good performance; if the code is too large, the performance can degrade markedly. To analyze whether this situation affects a particular code, one can try plotting the total flop rate as a function of problem size. If the flop rate decreases rapidly at some point, then the problem may likely be too large for the cache size.

- **Inconsistent timings**: Inconsistent timings are likely due to other users on the machine, thrashing (using more virtual memory than available physical memory), or paging in of the initial executable.

Section 13.8 provides information on overcoming paging overhead when profiling a code. We have found on all systems that if you follow all the advise above your timings will be consistent within a variation of less than five percent.

Chapter 15

Other PETSc Features

15.1 PETSc on a process subset

Users who wish to employ PETSc routines on only a subset of processes within a larger parallel job, or who wish to use a "master" process to coordinate the work of "slave" PETSc processes, should specify an alternative communicator for `PETSC_COMM_WORLD` by calling

PetscSetCommWorld(MPI_Comm comm);

before calling PetscInitialize(), but, obviously, after calling `MPI_Init()`. PetscSetCommWorld() can be called at most once per process. Most users will never need to use the routine PetscSetCommWorld().

15.2 Runtime Options

Allowing the user to modify parameters and options easily at runtime is very desirable for many applications. PETSc provides a simple mechanism to enable such customization. To print a list of available options for a given program, simply specify the option `-help` (or `-h`) at runtime, e.g.,

mpiexec -n 1 ./ex1 -help

Note that all runtime options correspond to particular PETSc routines that can be explicitly called from within a program to set compile-time defaults. For many applications it is natural to use a combination of compile-time and runtime choices. For example, when solving a linear system, one could explicitly specify use of the Krylov subspace technique BiCGStab by calling

KSPSetType(ksp,KSPBCGS);

One could then override this choice at runtime with the option

-ksp_type tfqmr

to select the Transpose-Free QMR algorithm. (See Chapter 4 for details.)

The remainder of this section discusses details of runtime options.

15.2.1 The Options Database

Each PETSc process maintains a database of option names and values (stored as text strings). This database is generated with the command PetscInitialize(), which is listed below in its C/C++ and Fortran variants, respectively:

PetscInitialize(int *argc,char ***args,const char *file,const char *help);
call PetscInitialize(character file,integer ierr)

The arguments `argc` and `args` (in the C/C++ version only) are the addresses of usual command line arguments, while the `file` is a name of a file that can contain additional options. By default this file is called `.petscrc` in the user's home directory. The user can also specify options via the environmental variable `PETSC_OPTIONS`. The options are processed in the following order:

- file
- environmental variable
- command line

Thus, the command line options supersede the environmental variable options, which in turn supersede the options file.

The file format for specifying options is

-optionname possible_value
-anotheroptionname possible_value
...

All of the option names must begin with a dash (-) and have no intervening spaces. Note that the option values cannot have intervening spaces either, and tab characters cannot be used between the option names and values. The user can employ any naming convention. For uniformity throughout PETSc, we employ the format `-package_option` (for instance, `-ksp_type` and `-mat_view ::info`).

Users can specify an alias for any option name (to avoid typing the sometimes lengthy default name) by adding an alias to the `.petscrc` file in the format

alias -newname -oldname

For example,

alias -kspt -ksp_type
alias -sd -start_in_debugger

Comments can be placed in the .petscrc file by using # in the first column of a line.

15.2.2 User-Defined PetscOptions

Any subroutine in a PETSc program can add entries to the database with the command

PetscOptionsSetValue(char *name,char *value);

though this is rarely done. To locate options in the database, one should use the commands

PetscOptionsHasName(char *pre,char *name,PetscBool *flg);
PetscOptionsGetInt(PetscOptions options,char *pre,char *name,int *value,PetscBool *flg);
PetscOptionsGetReal(char *pre,char *name,double *value,PetscBool *flg);
PetscOptionsGetString(char *pre,char *name,char *value,int maxlen,PetscBool *flg);
PetscOptionsGetStringArray(char *pre,char *name,char **values,int *maxlen,PetscBool *flg);
PetscOptionsGetIntArray(char *pre,char *name,int *value,int *nmax,PetscBool *flg);
PetscOptionsGetRealArray(char *pre,char *name,double *value, int *nmax,PetscBool *flg);

All of these routines set `flg=PETSC_TRUE` if the corresponding option was found, `flg=PETSC_FALSE` if it was not found. The optional argument `pre` indicates that the true name of the option is the given name (with the dash "-" removed) prepended by the prefix `pre`. Usually `pre` should be set to NULL (or `PETSC_NULL_CHARACTER` for Fortran); its purpose is to allow someone to rename all the options in a package without knowing the names of the individual options. For example, when using block Jacobi preconditioning, the KSP and PC methods used on the individual blocks can be controlled via the options `-sub_ksp_type` and `-sub_pc_type`.

15.2.3 Keeping Track of Options

One useful means of keeping track of user-specified runtime options is use of `-options_table`, which prints to `stdout` during `PetscFinalize()` a table of all runtime options that the user has specified. A related option is `-options_left`, which prints the options table and indicates any options that have *not* been requested upon a call to PetscFinalize(). This feature is useful to check whether an option has been activated for a particular PETSc object (such as a solver or matrix format), or whether an option name may have been accidentally misspelled.

15.3 Viewers: Looking at PETSc Objects

PETSc employs a consistent scheme for examining, printing, and saving objects through commands of the form

XXXView(XXX obj,PetscViewer viewer);

Here `obj` is any PETSc object of type `XXX`, where `XXX` is Mat, Vec, SNES, etc. There are several predefined viewers:

- Passing in a zero for the viewer causes the object to be printed to the screen; this is most useful when viewing an object in a debugger.
- PETSC_VIEWER_STDOUT_SELF and PETSC_VIEWER_STDOUT_WORLD cause the object to be printed to the screen.
- PETSC_VIEWER_DRAW_SELF PETSC_VIEWER_DRAW_WORLD causes the object to be drawn in a default X window.
- Passing in a viewer obtained by PetscViewerDrawOpen() causes the object to be displayed graphically.
- To save an object to a file in ASCII format, the user creates the viewer object with the command PetscViewerASCIIOpen(`MPI_Comm comm, char* file, PetscViewer *viewer`). This object is analogous to PETSC_VIEWER_STDOUT_SELF (for a communicator of `MPI_COMM_SELF`) and PETSC_VIEWER_STDOUT_WORLD (for a parallel communicator).
- To save an object to a file in binary format, the user creates the viewer object with the command PetscViewerBinaryOpen(`MPI_Comm` comm,char* file,PetscViewerBinaryType type, PetscViewer *viewer). Details of binary I/O are discussed below.
- Vector and matrix objects can be passed to a running MATLAB process with a viewer created by

  PetscViewerSocketOpen(MPI_Comm comm,char *machine,int port,PetscViewer *viewer).

On the MATLAB side, one must first run `v = sreader(int port)` and then `A = PetscBinaryRead(v)` to obtain the matrix or vector. Once all objects have been received, the port can be closed from the MATLAB end with `close(v)`. On the PETSc side, one should destroy the viewer object with `PetscViewerDestroy()`. The corresponding MATLAB `mex` files are located in `${PETSC_DIR}/src/sys/classes/viewer/impls/socket/matlab`.

The user can control the format of ASCII printed objects with viewers created by PetscViewerASCIIOpen() by calling

PetscViewerPushFormat(PetscViewer viewer,PetscViewerFormat format);

Possible formats include `PETSC_VIEWER_DEFAULT`, `PETSC_VIEWER_ASCII_MATLAB`, and `PETSC_VIEWER_ASCII_IMPL`. The implementation-specific format, `PETSC_VIEWER_ASCII_IMPL`, displays the object in the most natural way for a particular implementation.

The routines

PetscViewerPushFormat(PetscViewer viewer,int format);
PetscViewerPopFormat(PetscViewer viewer);

allow one to temporarily change the format of a viewer.

As discussed above, one can output PETSc objects in binary format by first opening a binary viewer with PetscViewerBinaryOpen() and then using MatView(), VecView(), etc. The corresponding routines for input of a binary object have the form `XXXLoad()`. In particular, matrix and vector binary input is handled by the following routines:

MatLoad(PetscViewer viewer,MatType outtype,Mat *newmat);
VecLoad(PetscViewer viewer,VecType outtype,Vec *newvec);

These routines generate parallel matrices and vectors if the viewer's communicator has more than one process. The particular matrix and vector formats are determined from the options database; see the manual pages for details.

One can provide additional information about matrix data for matrices stored on disk by providing an optional file `matrixfilename.info`, where `matrixfilename` is the name of the file containing the matrix. The format of the optional file is the same as the `.petscrc` file and can (currently) contain the following:

-matload_block_size <bs>

The block size indicates the size of blocks to use if the matrix is read into a block oriented data structure (for example, MATMPIBAIJ). The diagonal information `s1,s2,s3,...` indicates which (block) diagonals in the matrix have nonzero values.

15.4 Debugging

PETSc programs may be debugged using one of the two options below.

- `-start_in_debugger [noxterm,dbx,xxgdb] [-display name]` - start all processes in debugger
- `-on_error_attach_debugger [noxterm,dbx,xxgdb] [-display name]` - start debugger only on encountering an error

Note that, in general, debugging MPI programs cannot be done in the usual manner of starting the programming in the debugger (because then it cannot set up the MPI communication and remote processes).

By default the GNU debugger *gdb* is used when `-start_in_debugger` or `-on_error_attach_debugger` is specified. To employ either *xxgdb* or the common UNIX debugger *dbx*, one uses command line options as indicated above. On HP-UX machines the debugger *xdb* should be used instead of *dbx*; on RS/6000 machines the *xldb* debugger is supported as well. By default, the debugger will be started in a new xterm (to enable running separate debuggers on each process), unless the option `noxterm` is used. In order to handle the MPI startup phase, the debugger command "cont" should be used to continue execution of the program within the debugger. Rerunning the program through the debugger requires terminating the first job and restarting the processor(s); the usual "run" option in the debugger will not correctly handle the MPI startup and should not be used. Not all debuggers work on all machines, so the user may have to experiment to find one that works correctly.

You can select a subset of the processes to be debugged (the rest just run without the debugger) with the option

-debugger_nodes node1,node2,...

where you simply list the nodes you want the debugger to run with.

15.5 Error Handling

Errors are handled through the routine PetscError(). This routine checks a stack of error handlers and calls the one on the top. If the stack is empty, it selects PetscTraceBackErrorHandler(), which tries to print a traceback. A new error handler can be put on the stack with

PetscPushErrorHandler(PetscErrorCode (*HandlerFunction)(int line,char *dir,char *file,
char *message,int number,void*),void *HandlerContext)

The arguments to `HandlerFunction()` are the line number where the error occurred, the file in which the error was detected, the corresponding directory, the error message, the error integer, and the `HandlerContext`. The routine

PetscPopErrorHandler()

removes the last error handler and discards it.

PETSc provides two additional error handlers besides PetscTraceBackErrorHandler():

PetscAbortErrorHandler()
PetscAttachErrorHandler()

The function PetscAbortErrorHandler() calls abort on encountering an error, while PetscAttachErrorHandler() attaches a debugger to the running process if an error is detected. At runtime, these error handlers can be set with the options `-on_error_abort` or `-on_error_attach_debugger [noxterm, dbx, xxgdb, xldb] [-display DISPLAY]`.

All PETSc calls can be traced (useful for determining where a program is hanging without running in the debugger) with the option

-log_trace [filename]

where `filename` is optional. By default the traces are printed to the screen. This can also be set with the command PetscLogTraceBegin(FILE*).

It is also possible to trap signals by using the command

PetscPushSignalHandler(PetscErrorCode (*Handler)(int,void *),void *ctx);

The default handler PetscSignalHandlerDefault() calls PetscError() and then terminates. In general, a signal in PETSc indicates a catastrophic failure. Any error handler that the user provides should try to clean up only before exiting. By default all PETSc programs use the default signal handler, although the user can turn this off at runtime with the option `-no_signal_handler`.

There is a separate signal handler for floating-point exceptions. The option `-fp_trap` turns on the floating-point trap at runtime, and the routine

PetscSetFPTrap(int flag);

can be used in-line. A `flag` of `PETSC_FP_TRAP_ON` indicates that floating-point exceptions should be trapped, while a value of `PETSC_FP_TRAP_OFF` (the default) indicates that they should be ignored. Note that on certain machines, in particular the IBM RS/6000, trapping is very expensive.

A small set of macros is used to make the error handling lightweight. These macros are used throughout the PETSc libraries and can be employed by the application programmer as well. When an error is first detected, one should set it by calling

SETERRQ(MPI_Comm comm,PetscErrorCode flag,,char *message);

The user should check the return codes for all PETSc routines (and possibly user-defined routines as well) with

ierr = PetscRoutine(...);CHKERRQ(PetscErrorCode ierr);

Likewise, all memory allocations should be checked with

ierr = PetscMalloc1(n,&ptr);CHKERRQ(ierr);

If this procedure is followed throughout all of the user's libraries and codes, any error will by default generate a clean traceback of the location of the error.

Note that the macro `__FUNCT__` is used to keep track of routine names during error tracebacks. Users need not worry about this macro in their application codes; however, users can take advantage of this feature if desired by setting this macro before each user-defined routine that may call `SETERRQ()`, `CHKERRQ()`. A simple example of usage is given below.

```
#undef __FUNCT__
#define __FUNCT__ "MyRoutine1"
int MyRoutine1() {
/* code here */
return 0;
}
```

15.6 Incremental Debugging

When developing large codes, one is often in the position of having a correctly (or at least believed to be correctly) running code; making a change to the code then changes the results for some unknown reason. Often even determining the precise point at which the old and new codes diverge is a major pain. In other cases, a code generates different results when run on different numbers of processes, although in exact arithmetic the same answer is expected. (Of course, this assumes that *exactly* the same solver and parameters are used in the two cases.)

PETSc provides some support for determining exactly where in the code the computations lead to different results. First, compile both programs with different names. Next, start running both programs as a single

MPI job. This procedure is dependent on the particular MPI implementation being used. For example, when using MPICH on workstations, *procgroup* files can be used to specify the processors on which the job is to be run. Thus, to run two programs, `old` and `new`, each on two processors, one should create the procgroup file with the following contents:

local 0
workstation1 1 /home/bsmith/old
workstation2 1 /home/bsmith/new
workstation3 1 /home/bsmith/new

(Of course, workstation1, etc. can be the same machine.) Then, one can execute the command

mpiexec -p4pg <procgroup_filemame> old -compare <tolerance> [options]

Note that the same runtime options must be used for the two programs. The first time an inner product or norm detects an inconsistency larger than `<tolerance>`, PETSc will generate an error. The usual runtime options `-start_in_debugger` and `-on_error_attach_debugger` may be used. The user can also place the commands

PetscCompareDouble()
PetscCompareScalar()
PetscCompareInt()

in portions of the application code to check for consistency between the two versions.

15.7 Complex Numbers

PETSc supports the use of complex numbers in application programs written in C, C++, and Fortran. To do so, we employ either the C99 `complex` type or the C++ versions of the PETSc libraries in which the basic "scalar" datatype, given in PETSc codes by PetscScalar, is defined as `complex` (or `complex<double>` for machines using templated complex class libraries). To work with complex numbers, the user should run ./configure with the additional option `--with-scalar-type=complex`. The file `${PETSC_DIR}/src/docs/website/documentation/installation.html` provides detailed instructions for installing PETSc. You can use `--with-clanguage=c` (the default) to use the C99 complex numbers or `--with-clanguage=c++` to use the C++ complex type.

We recommend using optimized Fortran kernels for some key numerical routines with complex numbers (such as matrix-vector products, vector norms, etc.) instead of the default C/C++ routines. This can be done with the `./configure` option `--with-fortran-kernels=generic`. This implementation exploits the maturity of Fortran compilers while retaining the identical user interface. For example, on rs6000 machines, the base single-node performance when using the Fortran kernels is 4-5 times faster than the default C++ code.

Recall that each variant of the PETSc libraries is stored in a different directory, given by

`${PETSC_DIR}/lib/${PETSC_ARCH}`,

according to the architecture.. Thus, the libraries for complex numbers are maintained separately from those for real numbers. When using any of the complex numbers versions of PETSc, *all* vector and matrix elements are treated as complex, even if their imaginary components are zero. Of course, one can elect to use only the real parts of the complex numbers when using the complex versions of the PETSc libraries; however, when working *only* with real numbers in a code, one should use a version of PETSc for real numbers for best efficiency.

The program ${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex11.c solves a linear system with a complex coefficient matrix. Its Fortran counterpart is ${PETSC_DIR}/src/ksp/ksp/examples/tutorials/ex11f.F.

15.8 Emacs Users

If users develop application codes using Emacs (which we highly recommend), the etags feature can be used to search PETSc files quickly and efficiently. To use this feature, one should first check if the file, ${PETSC_DIR}/TAGS exists. If this file is not present, it should be generated by running make alletags from the PETSc home directory. Once the file exists, from Emacs the user should issue the command

M-x visit-tags-table

where "M" denotes the Emacs Meta key, and enter the name of the TAGS file. Then the command "M-." will cause Emacs to find the file and line number where a desired PETSc function is defined. Any string in any of the PETSc files can be found with the command "M-x tags-search". To find repeated occurrences, one can simply use "M-," to find the next occurrence.

An alternative which provides reverse lookups (e.g. find all call sites for a given function) and is somewhat more amenable to managing many projects is GNU Global, available from http://www.gnu.org/s/global/. Tags for PETSc and all external packages can be generated by running the command

```
find $PETSC_DIR/{include,src,tutorials,$PETSC_ARCH/include} any/other/paths \
   -regex '.*\.\(cc\|hh\|cpp\|C\|hpp\|c\|h\|cu\)$' \
   | grep -v ftn-auto | gtags -f -
```

from your home directory or wherever you keep source code. If you are making large changes, it is useful to either set this up to run as a cron job or to make a convenient alias so that refreshing is easy. Then add the following to ~/.emacs to enable gtags and replace the plain etags bindings.

```
(when (require 'gtags)
  (global-set-key "\C-cf" 'gtags-find-file)
  (global-set-key "\M-." 'gtags-find-tag)
  (define-key gtags-mode-map (kbd "C-c r") 'gtags-find-rtag))
(add-hook 'c-mode-common-hook
          '(lambda () (gtags-mode t))) ; Or add to existing hook
```

15.9 Vi and Vim Users

If users develop application codes using Vi or Vim the tags feature can be used to search PETSc files quickly and efficiently. To use this feature, one should first check if the file, ${PETSC_DIR}/CTAGS exists. If this file is not present, it should be generated by running make alletags from the PETSc home directory. Once the file exists, from Vi/Vim the user should issue the command

:set tags=CTAGS

from the PETSC_DIR directory and enter the name of the CTAGS file. Then the command "tag functionname" will cause Vi/Vim to find the file and line number where a desired PETSc function is defined. See, for example, http://www.yolinux.com/TUTORIALS/LinuxTutorialAdvanced_vi.html for additional Vi/Vim options that allow searches etc. It is also possible to use GNU Global with Vim, see the description for Emacs above.

15.10 Eclipse Users

If you are interested in developing code that uses PETSc from Eclipse or developing PETSc in Eclipse and have knowledge of how to do indexing and build libraries in Eclipse please contact us at petsc-dev@mcs.anl.gov.

To allow an Eclipse code to compile with the PETSc include files and link with the PETSc libraries someone has suggested To do this, click on your managed C project with the right sided mouse button, select

Properties -¿ C/C++ Build -¿ Settings

Then you get a new window with on the right hand side the various setting options.

Select Includes, and add the required PETSc paths. In my case I have added

```
${PETSC_DIR}/include
${PETSC_DIR}/${PETSC_ARCH}/include
```

Then select "Libraries" under the header Linker and you should set the Library search path:

```
${PETSC_DIR}/${PETSC_ARCH}/lib
```

and then the libraries, in my case:

```
m, petsc, stdc++, mpichxx, mpich, lapack, blas, gfortran, dl, rt,gcc_s, pthread, X1
```

To make PETSc an Eclipse package

- Install the Mercurial plugin for Eclipse and then import the PETSc repository to Eclipse.
- elected New-¿Convert to C/C++ project and selected shared library. After this point you can perform searches in the code.

A PETSc user has provided the following steps to build an Eclipse index for PETSc that can be used with their own code without compiling PETSc source into their project.

- In the user project source directory, create a symlink to the petsc/src directory.
- Refresh the project explorer in Eclipse, so the new symlink is followed.
- Right-click on the project in the project explorer, and choose "Index -¿ Rebuild". The index should now be build.
- Right-click on the PETSc symlink in the project explorer, and choose "Exclude from build..." to make sure Eclipse does not try to compile PETSc with the project.

15.11 Qt Creator

Qt Creator is developed as a part of Qt SDK for cross-platform GUI programming using C++ and thus has excellent support for application developing using C/C++. This information was provided by Mohammad Mirzadeh.

1. Qt Creator supports automated and cross-platform qmake and Cmake build systems. What this means, is that, starting from a handful set of options, Qt Creator is able to generate makefiles that are used to compile and run your application package on Linux, Windows and Mac systems.
2. Qt Creator makes the task of linking PETSc library to your application package easy.
3. Qt Creator has visual debugger and supports both GNU's GDB and Microsoft's CDB (on Windows).

4. Qt Creator (as of version 2.3) has interface to memory profiling software such as valgrind.

5. Qt Creator has built-in auto-completion and function look-up. This feature of editor, makes it possible for the user to easily browse inside huge packages like PETSc and easily find relevant functions and objects.

6. Qt Creator has inline error/warning messages. This feature of the editor greatly reduces programming bugs by informing the user of potential error/warning as the code is being written and even before the compile step.

7. Qt Creator is an excellent environment for doing GUI programming using Qt framework. This means you can easily generate a nice GUI for you application package as well!

Qt Creator is a part of Qt SDK, currently being developed by Nokia Inc. It is available under GPL v3, LGPL v2 and a commercial license and may be obtained, either as a part of Qt SDK or as an stand-alone software, via http://qt.nokia.com/downloads/.

How to create a project?

Upon installation, all you really need to do to port your software package to Qt Creator is to build a `.pro` file. A `.pro` file is simply a configuration file that tells Qt Creator about all build/compile options and locations of source file. One may start with a blank `.pro` file and fill in the configuration options as needed:

```
TARGET = name_of_your_app

APP_DIR      = /home/path/to/your/app
PETSC_DIR    = /home/path/to/your/petsc
PETSC_ARCH = linux-gnu-cxx-debug
PETSC_BINS = $$PETSC_DIR/$$PETSC_ARCH/bin

INCLUDEPATH += $$APP_DIR/include \
               $$PETSC_DIR/include \
               $$PETSC_DIR/$$PETSC_ARCH/include

LIBS += -L/path/to/petsc/libs -l_all_needed_petsc_libs

QMAKE_CC   = $$PETSC_BINS/\href{http://www.mcs.anl.gov/petsc/petsc-3.7/docs/http://v
QMAKE_CXX = $$PETSC_BINS/mpicxx

QMAKE_CFLAGS     += -O3
QMAKE_CXXFLAGS += -O3
QMAKE_LFLAGS     += -O3

SOURCES += \
$$APP_DIR/src/source1.cpp \
$$APP_DIR/src/source2.cpp \
$$APP_DIR/src/main.cpp
HEADERS += \
$$APP_DIR/src/source1.h \
```

```
$$APP_DIR/src/source2.h
OTHER_FILES += \
$$APP_DIR/src/some_file.cpp \
$$APP_DIR/dir/another_file.xyz
```

In this example, there are different keywords used. These include:

- `TARGET`: This defines the name of your application executable. You can choose it however you like.
- `INCLUDEPATH`: This is used in the compile time to point to all the needed include files. Essentially it is used as a `-I $$INCLUDEPATH` flag for the compiler. This should include all your application-specific header files and those related to PETSc which may be found via `make getincludedirs`.
- `LIBS`: This defines all needed external libraries to link with your application. To get PETSc's linking libraries do `make getlinklibs`.
- `QMAKE_CC` and `QMAKE_CXX`: These define which `C/C++` compilers to choose for your application.
- `QMAKE_CFLAGS`, `QMAKE_CXXFLAGS` and `QMAKE_LFLAGS`: These set the corresponding compile and linking flags.
- `SOURCES`: This sets all the source files that need to be compiled.
- `HEADERS`: This sets all the header files that are needed for your application. Note that since header files are merely included in the source files, they are not compiled. As a result this is an optional flag and is simply included to allow users to easily access the header files in their application (or even PETSc header files).
- `OTHER_FILES`: This can be used to virtually include any other file (source, header or any other extension) just as a way to make accessing to auxiliary files easier. They could be anything from a text file to a PETSc source file. Note that none of the source files placed here are compiled.

Note that there are many more options available that one can feed into the `.pro` file. For more information, one should follow http://doc.qt.nokia.com/latest/qmake-variable-reference.html. Once the `.pro` file is generated, the user can simply open it via Qt Creator. Upon opening, generally one has the option to create two different build options, debug and release, and switch between the two easily at any time. For more information on how to use the Qt Creator interface, and other more advanced aspects of this IDE, one may refer to http://doc.qt.nokia.com/qtcreator-snapshot/index.html

15.12 Developers Studio Users

To use PETSc from MS Visual Studio - one would have to compile a PETSc example with its corresponding makefile - and then transcribe all compiler and linker options used in this build into a Visual Studio project file - in the appropriate format in Visual Studio project settings.

15.13 XCode Users (The Apple GUI Development System

15.13.1 Mac OSX

Follow the instructions in systems/Apple/OSX/bin/makeall to build the PETSc framework and documentation suitable for use in Xcode.

You can then use the PETSc framework in ${PETSC_DIR}/arch-osx/PETSc.framework in the usual manner for Apple frameworks, see the examples in systems/Apple/OSX/examples. When working in xcode things like function name completion should work for all PETSc functions as well as MPI functions. You must also link against the Apple Accelerate.framework.

15.13.2 iPhone/iPad iOS

Follow the instructions in systems/Apple/ios/bin/iosbuilder.py to build the PETSc library for use on the iPhone/iPad.

You can then use the PETSc static library in ${PETSC_DIR}/arch-osx/libPETSc.a in the usual manner for Apple libraries inside your iOS xcode projects, see the examples in systems/Apple/iOS/examples. You must also link against the Apple Accelerate.framework.

15.14 Parallel Communication

When used in a message-passing environment, all communication within PETSc is done through MPI, the message-passing interface standard [22]. Any file that includes `petscsys.h` (or any other PETSc include file), can freely use any MPI routine.

15.15 Graphics

PETSc graphics library is not intended to compete with high-quality graphics packages. Instead, it is intended to be easy to use interactively with PETSc programs. We urge users to generate their publication-quality graphics using a professional graphics package. If a user wants to hook certain packages in PETSc, he or she should send a message to petsc-maint@mcs.anl.gov, and we will see whether it is reasonable to try to provide direct interfaces.

15.15.1 Windows as PetscViewers

For drawing predefined PETSc objects such as matrices and vectors, one must first create a viewer using the command

PetscViewerDrawOpen(MPI_Comm comm,char *display,char *title,int x,
int y,int w,int h,PetscViewer *viewer);

This viewer may be passed to any of the `XXXView()` routines. To draw into the viewer, one must obtain the Draw object with the command

PetscViewerDrawGetDraw(PetscViewer viewer,PetscDraw *draw);

Then one can call any of the `PetscDrawXXX` commands on the `draw` object. If one obtains the `draw` object in this manner, one does not call the PetscDrawOpenX() command discussed below.

Predefined viewers, PETSC_VIEWER_DRAW_WORLD and PETSC_VIEWER_DRAW_SELF, may be used at any time. Their initial use will cause the appropriate window to be created.

By default, PETSc drawing tools employ a private colormap, which remedies the problem of poor color choices for contour plots due to an external program's mangling of the colormap (e.g, Netscape tends to do this). Unfortunately, this causes flashing of colors as the mouse is moved between the PETSc windows and other windows. Alternatively, a shared colormap can be used via the option `-draw_x_shared_colormap`.

15.15.2 Simple PetscDrawing

One can open a window that is not associated with a viewer directly under the X11 Window System or OpenGL with the command

PetscDrawCreate(MPI_Comm comm,char *display,char *title,int x,
int y,int w,int h,PetscDraw *win); PetscDrawSetFromOptions(win);

All drawing routines are done relative to the windows coordinate system and viewport. By default the drawing coordinates are from `(0,0)` to `(1,1)`, where `(0,0)` indicates the lower left corner of the window. The application program can change the window coordinates with the command

PetscDrawSetCoordinates(PetscDraw win,double xl,double yl,double xr,double yr);

By default, graphics will be drawn in the entire window. To restrict the drawing to a portion of the window, one may use the command

PetscDrawSetViewPort(PetscDraw win,double xl,double yl,double xr,double yr);

These arguments, which indicate the fraction of the window in which the drawing should be done, must satisfy $0 \leq$ `xl` $\leq$ `xr` ≤ 1 and $0 \leq$ `yl` $\leq$ `yr` ≤ 1.

To draw a line, one uses the command

PetscDrawLine(PetscDraw win,double xl,double yl,double xr,double yr,int cl);

The argument `cl` indicates the color (which is an integer between 0 and 255) of the line. A list of predefined colors may be found in `include/petscdraw.h` and includes `PETSC_DRAW_BLACK`, `PETSC_DRAW_RED`, `PETSC_DRAW_BLUE` etc.

To ensure that all graphics actually have been displayed, one should use the command

PetscDrawFlush(PetscDraw win);

When displaying by using double buffering, which is set with the command

PetscDrawSetDoubleBuffer(PetscDraw win);

all processes must call

PetscDrawFlush(PetscDraw win);

in order to swap the buffers. From the options database one may use `-draw_pause n`, which causes the PETSc application to pause `n` seconds at each PetscDrawPause(). A time of `-1` indicates that the application should pause until receiving mouse input from the user.

Text can be drawn with commands

PetscDrawString(PetscDraw win,PetscReal x,PetscReal y,int color,char *text);
PetscDrawStringVertical(PetscDraw win,PetscReal x,PetscReal y,int color,const char *text); PetscDrawStringCentered(Pets

The user can set the text font size or determine it with the commands

PetscDrawStringSetSize(PetscDraw win,PetscReal width,PetscReal height);
PetscDrawStringGetSize(PetscDraw win,PetscReal *width,PetscReal *height);

15.15.3 Line Graphs

PETSc includes a set of routines for manipulating simple two-dimensional graphs. These routines, which begin with PetscDrawAxisDraw(), are usually not used directly by the application programmer. Instead, the programmer employs the line graph routines to draw simple line graphs. As shown in the program, within Figure 21, line graphs are created with the command

PetscDrawLGCreate(PetscDraw win,int ncurves,PetscDrawLG *ctx);

The argument `ncurves` indicates how many curves are to be drawn. Points can be added to each of the curves with the command

PetscDrawLGAddPoint(PetscDrawLG ctx,PetscReal *x,PetscReal *y);

The arguments `x` and `y` are arrays containing the next point value for each curve. Several points for each curve may be added with

PetscDrawLGAddPoints(PetscDrawLG ctx,int n,PetscReal **x,PetscReal **y);

The line graph is drawn (or redrawn) with the command

PetscDrawLGDraw(PetscDrawLG ctx);

A line graph that is no longer needed can be destroyed with the command

PetscDrawLGDestroy(PetscDrawLG *ctx);

To plot new curves, one can reset a linegraph with the command

PetscDrawLGReset(PetscDrawLG ctx);

The line graph automatically determines the range of values to display on the two axes. The user can change these defaults with the command

PetscDrawLGSetLimits(PetscDrawLG ctx,PetscReal xmin,PetscReal xmax,PetscReal ymin,PetscReal ymax);

It is also possible to change the display of the axes and to label them. This procedure is done by first obtaining the axes context with the command

PetscDrawLGGetAxis(PetscDrawLG ctx,PetscDrawAxis *axis);

One can set the axes' colors and labels, respectively, by using the commands

PetscDrawAxisSetColors(PetscDrawAxis axis,int axis_lines,int ticks,int text);
PetscDrawAxisSetLabels(PetscDrawAxis axis,char *top,char *x,char *y);

```
static char help[] = "Plots a simple line graph.\n";

#if defined(PETSC_APPLE_FRAMEWORK)
#import <PETSc/petscsys.h>
#import <PETSc/petscdraw.h>
#else
#include <petscsys.h>
#include <petscdraw.h>
#endif
```

```
#undef __FUNCT__
#define __FUNCT__ "main"
int main(int argc,char **argv)
{
  PetscDraw          draw;
  PetscDrawLG        lg;
  PetscDrawAxis      axis;
  PetscInt           n = 15,i,x = 0,y = 0,width = 400,height = 300,nports
= 1;
  PetscBool          useports,flg;
  const char         *xlabel,*ylabel,*toplabel,*legend;
  PetscReal          xd,yd;
  PetscDrawViewPorts *ports = NULL;
  PetscErrorCode     ierr;

  toplabel = "Top Label"; xlabel = "X-axis Label"; ylabel = "Y-axis Label";
legend = "Legend";

  ierr = PetscInitialize(&argc,&argv,NULL,help);CHKERRQ(ierr);
  ierr = PetscOptionsGetInt(NULL,NULL,"-x",&x,NULL);CHKERRQ(ierr);
  ierr = PetscOptionsGetInt(NULL,NULL,"-y",&y,NULL);CHKERRQ(ierr);
  ierr = PetscOptionsGetInt(NULL,NULL,"-width",&width,NULL);CHKERRQ(ierr);
  ierr = PetscOptionsGetInt(NULL,NULL,"-height",&height,NULL);CHKERRQ(ierr);
  ierr = PetscOptionsGetInt(NULL,NULL,"-n",&n,NULL);CHKERRQ(ierr);
  ierr = PetscOptionsGetInt(NULL,NULL,"-nports",&nports,&useports);CHKERRQ(ierr);
  ierr = PetscOptionsHasName(NULL,NULL,"-nolegend",&flg);CHKERRQ(ierr);
  if (flg) legend = NULL;
  ierr = PetscOptionsHasName(NULL,NULL,"-notoplabel",&flg);CHKERRQ(ierr);
  if (flg) toplabel = NULL;
  ierr = PetscOptionsHasName(NULL,NULL,"-noxlabel",&flg);CHKERRQ(ierr);
  if (flg) xlabel = NULL;
  ierr = PetscOptionsHasName(NULL,NULL,"-noylabel",&flg);CHKERRQ(ierr);
  if (flg) ylabel = NULL;
  ierr = PetscOptionsHasName(NULL,NULL,"-nolabels",&flg);CHKERRQ(ierr);
  if (flg) {toplabel = NULL; xlabel = NULL; ylabel = NULL;}

  ierr = PetscDrawCreate(PETSC_COMM_WORLD,0,"Title",x,y,width,height,&draw);CHKERRQ(ierr);
  ierr = PetscDrawSetFromOptions(draw);CHKERRQ(ierr);
  if (useports) {
    ierr = PetscDrawViewPortsCreate(draw,nports,&ports);CHKERRQ(ierr);
    ierr = PetscDrawViewPortsSet(ports,0);CHKERRQ(ierr);
  }
  ierr = PetscDrawLGCreate(draw,1,&lg);CHKERRQ(ierr);
  ierr = PetscDrawLGSetUseMarkers(lg,PETSC_TRUE);CHKERRQ(ierr);
  ierr = PetscDrawLGGetAxis(lg,&axis);CHKERRQ(ierr);
  ierr = PetscDrawAxisSetColors(axis,PETSC_DRAW_BLACK,PETSC_DRAW_RED,PETSC_DRAW_BLUE);CHKER
  ierr = PetscDrawAxisSetLabels(axis,toplabel,xlabel,ylabel);CHKERRQ(ierr);
  ierr = PetscDrawLGSetLegend(lg,&legend);CHKERRQ(ierr);
  ierr = PetscDrawLGSetFromOptions(lg);CHKERRQ(ierr);

  for (i=0; i<=n; i++) {
    xd   = (PetscReal)(i - 5); yd = xd*xd;
    ierr = PetscDrawLGAddPoint(lg,&xd,&yd);CHKERRQ(ierr);
  }
  ierr = PetscDrawLGDraw(lg);CHKERRQ(ierr);
```

```
  ierr = PetscDrawLGSave(lg);CHKERRQ(ierr);

  ierr = PetscDrawViewPortsDestroy(ports);CHKERRQ(ierr);
  ierr = PetscDrawLGDestroy(&lg);CHKERRQ(ierr);
  ierr = PetscDrawDestroy(&draw);CHKERRQ(ierr);
  ierr = PetscFinalize();
  return 0;
}
```

Figure 21: Example of PetscDrawing Plots

It is possible to turn off all graphics with the option `-nox`. This will prevent any windows from being opened or any drawing actions to be done. This is useful for running large jobs when the graphics overhead is too large, or for timing.

15.15.4 Graphical Convergence Monitor

For both the linear and nonlinear solvers default routines allow one to graphically monitor convergence of the iterative method. These are accessed via the command line with `-ksp_monitor_lg_residualnorm` and `-snes_monitor_lg_residualnorm`. See also Sections 4.3.3 and 5.3.2.

The two functions used are KSPMonitorLGResidualNorm() and KSPMonitorLGResidualNormCreate(). These can easily be modified to serve specialized needs.

15.15.5 Disabling Graphics at Compile Time

To disable all x-window-based graphics, run ./configure with the additional option –with-x=0

Chapter 16

Makefiles

This chapter describes the design of the PETSc makefiles, which are the key to managing our code portability across a wide variety of UNIX and Windows systems.

16.1 Our Makefile System

To make a program named `ex1`, one may use the command

make PETSC_ARCH=arch ex1

which will compile the example and automatically link the appropriate libraries. The architecture, `arch`, is one of `solaris, rs6000, IRIX, hpux`, etc. Note that when using command line options with make (as illustrated above), one must *not* place spaces on either side of the "=" signs. The variable `PETSC_ARCH` can also be set as environmental variables. Although PETSc is written in C, it can be compiled with a C++ compiler. For many C++ users this may be the preferred route. To compile with the C++ compiler, one should use the ./configure option `--with-clanguage=c++`.

16.1.1 Makefile Commands

The directory `${PETSC_DIR}/conf` contains virtually all makefile commands and customizations to enable portability across different architectures. Most makefile commands for maintaining the PETSc system are defined in the file `${PETSC_DIR}/conf`. These commands, which process all appropriate files within the directory of execution, include

- `lib` - Updates the PETSc libraries based on the source code in the directory.
- `libfast` - Updates the libraries faster. Since `libfast` recompiles all source files in the directory at once, rather than individually, this command saves time when many files must be compiled.
- `clean` - Removes garbage files.

The `tree` command enables the user to execute a particular action within a directory and all of its subdirectories. The action is specified by `ACTION=[action]`, where `action` is one of the basic commands listed above. For example, if the command

make ACTION=lib tree

were executed from the directory `${PETSC_DIR}/src/ksp/ksp`, the debugging library for all Krylov subspace solvers would be built.

16.1.2 Customized Makefiles

The directory ${PETSC_DIR}/ contains a subdirectory for each architecture that contains machine-specific information, enabling the portability of our makefile system, these are ${PETSC_DIR}/${PETSC_ARCH}/conf. Each architecture directory contains two makefiles:

- petscvariables - definitions of the compilers, linkers, etc.
- petscrules - some build rules specific to this machine.

These files are generated automatically when you run ./configure.

The architecture independent makefiles, are located in ${PETSC_DIR}/lib/petsc/conf, and the machine-specific makefiles get included from here.

16.2 PETSc Flags

PETSc has several flags that determine how the source code will be compiled. The default flags for particular versions are specified by the variable PETSCFLAGS within the base files of ${PETSC_DIR}/${PETSC_ARCH}/conf, discussed in Section 16.1.2. The flags include

- PETSC_USE_DEBUG - The PETSc debugging options are activated. We recommend always using this.
- PETSC_USE_COMPLEX - The version with scalars represented as complex numbers is used.
- PETSC_USE_LOG - Various monitoring statistics on floating-point operations, and message-passing activity are kept.

16.2.1 Sample Makefiles

Maintaining portable PETSc makefiles is very simple.

The first is a "minimum" makefile for maintaining a single program that uses the PETSc libraries. The most important line in this makefile is the line starting with include:

include ${PETSC_DIR}/lib/petsc/conf/variables include ${PETSC_DIR}/lib/petsc/conf/rules

This line includes other makefiles that provide the needed definitions and rules for the particular base PETSc installation (specified by ${PETSC_DIR}) and architecture (specified by ${PETSC_ARCH}). (See 1.2 for information on setting these environmental variables.) As listed in the sample makefile, the appropriate include file is automatically completely specified; the user should *not* alter this statement within the makefile.

```
ALL: ex2
CFLAGS     =
FFLAGS     =
CPPFLAGS   =
FPPFLAGS   =
CLEANFILES = ex2

include ${PETSC_DIR}/lib/petsc/conf/variables
include ${PETSC_DIR}/lib/petsc/conf/rules

ex2: ex2.o chkopts
        ${CLINKER} -o ex2 ex2.o  ${PETSC_LIB}
        ${RM} ex2.o
```

Figure 22: Sample PETSc Makefile for a Single Program

For users who wish to manage the compile process themselves and **not** use the rules PETSc uses for compiling programs include variables instead of base. That is, use something like

```
ALL: ex2
CFLAGS   = ${PETSC_CC_INCLUDES}
FFLAGS   = ${PETSC_FC_INCLUDES}

include ${PETSC_DIR}/lib/petsc/conf/variables

ex2: ex2.o
        mylinkercommand -o ex2 ex2.o  ${PETSC_LIB}
```

Figure 23: Sample PETSc Makefile that does **not** use PETSc's rules for compiling

The variables `${PETSC_CC_INCLUDES}`, `${PETSC_FC_INCLUDES}` and `${PETSC_LIB}` are defined by the included petsc/conf/variables file.

If you do not wish to include any PETSc makefiles in your makefile, you can use the commands (run in the PETSc root directory) to get the information needed by your makefile: make getlinklibs getincludedirs getpetscflags. All the libraries listed need to be linked into your executable and the include directories and flags need to be passed to the compiler usually this is done with "CFLAGS = list of -I and other flags" and "FFLAGS = list of -I and other flags" in your makefile.

Note that the variable `${PETSC_LIB}` (as listed on the link line in the above makefile) specifies *all* of the various PETSc libraries in the appropriate order for correct linking. For users who employ only a specific PETSc library, can use alternative variables like `${PETSC_SYS_LIB}`, `${PETSC_VEC_LIB}`, `${PETSC_MAT_LIB}`, `${PETSC_DM_LIB}`, `${PETSC_KSP_LIB}`, `${PETSC_SNES_LIB}` or `${PETSC_TS_LIB}`.

The second sample makefile, given in Figure 24, controls the generation of several example programs.

```
CFLAGS   =
FFLAGS   =
CPPFLAGS =
FPPFLAGS =

include ${PETSC_DIR}/lib/petsc/conf/variables
include ${PETSC_DIR}/lib/petsc/conf/rules

ex1: ex1.o
     -${CLINKER} -o ex1 ex1.o  ${PETSC_LIB}
     ${RM} ex1.o
ex2: ex2.o
     -${CLINKER} -o ex2 ex2.o  ${PETSC_LIB}
     ${RM} ex2.o
ex3: ex3.o
     -${FLINKER} -o ex3 ex3.o  ${PETSC_LIB}
     ${RM} ex3.o
ex4: ex4.o
     -${CLINKER} -o ex4 ex4.o  ${PETSC_LIB}
     ${RM} ex4.o

runex1:
```

```
	-@${MPIEXEC} ex1
runex2:
	-@${MPIEXEC} -n 2 ./ex2 -mat_seqdense -options_left
runex3:
	-@${MPIEXEC} ex3 -v -log_summary
runex4:
	-@${MPIEXEC} -n 4 ./ex4 -trdump

RUNEXAMPLES_1 = runex1 runex2
RUNEXAMPLES_2 = runex4
RUNEXAMPLES_3 = runex3
EXAMPLESC     = ex1.c ex2.c ex4.c
EXAMPLESF     = ex3.F
EXAMPLES_1    = ex1 ex2
EXAMPLES_2    = ex4
EXAMPLES_3    = ex3

include ${PETSC_DIR}/lib/petsc/conf/test
```

Figure 24: Sample PETSc Makefile for Several Example Programs

Again, the most important line in this makefile is the `include` line that includes the files defining all of the macro variables. Some additional variables that can be used in the makefile are defined as follows:

- `CFLAGS, FFLAGS` User specified additional options for the C compiler and fortran compiler.
- `CPPFLAGS, FPPFLAGS` User specified additional flags for the C preprocessor and fortran preprocessor.
- `CLINKER, FLINKER` the C and Fortran linkers.
- `RM` the remove command for deleting files.

Note that the PETSc example programs are divided into several categories, which currently include:

- `EXAMPLES_1` basic C suite used in installation tests
- `EXAMPLES_2` additional C suite including graphics
- `EXAMPLES_3` basic Fortran .F suite
- `EXAMPLES_4` subset of 1 and 2 that runs on only a single process
- `EXAMPLES_5` examples that require complex numbers
- `EXAMPLES_6` C examples that do not work with complex numbers
- `EXAMPLES_8` Fortran .F examples that do not work with complex numbers

- `EXAMPLES_9` uniprocess version of 3

- `EXAMPLES_10` Fortran .F examples that require complex numbers

We next list in Figure 25 a makefile that maintains a PETSc library. Although most users do not need to understand or deal with such makefiles, they are also easily used.

```
ALL: lib
CFLAGS =
SOURCEC = sp1wd.c spinver.c spnd.c spqmd.c sprcm.c
SOURCEF = degree.F fnroot.F genqmd.F qmdqt.F rcm.F fn1wd.F gen1wd.F
genrcm.F qmdrch.F rootls.F fndsep.F gennd.F qmdmrg.F qmdupd.F
SOURCEH =
LIBBASE = libpetscmat
MANSEC = Mat
include $PETSC_DIR/lib/petsc/conf/variables include $PETSC_DIR/lib/petsc/conf/rules
```

Figure 25: Sample PETSc Makefile for Library Maintenance

The library's name is `libpetscmat.a`, and the source files being added to it are indicated by `SOURCEC` (for C files) and `SOURCEF` (for Fortran files).

The variable `MANSEC` indicates that any manual pages generated from this source should be included in the `Mat` section.

16.3 Limitations

This approach to portable makefiles has some minor limitations, including the following:

- Each makefile must be called "makefile".

- Each makefile can maintain at most one archive library.

Chapter 17

Unimportant and Advanced Features of Matrices and Solvers

This chapter introduces additional features of the PETSc matrices and solvers. Since most PETSc users should not need to use these features, we recommend skipping this chapter during an initial reading.

17.1 Extracting Submatrices

One can extract a (parallel) submatrix from a given (parallel) using

MatGetSubMatrix(Mat A,IS rows,IS cols,MatReuse call,Mat *B);

This extracts the `rows` and `col`umns of the matrix `A` into `B`. If call is `MAT_INITIAL_MATRIX` it will create the matrix `B`. If call is `MAT_REUSE_MATRIX` it will reuse the `B` created with a previous call.

17.2 Matrix Factorization

Normally, PETSc users will access the matrix solvers through the KSP interface, as discussed in Chapter 4, but the underlying factorization and triangular solve routines are also directly accessible to the user.

The LU and Cholesky matrix factorizations are split into two or three stages depending on the user's needs. The first stage is to calculate an ordering for the matrix. The ordering generally is done to reduce fill in a sparse factorization; it does not make much sense for a dense matrix.

MatGetOrdering(Mat matrix,MatOrderingType type,IS* rowperm,IS* colperm);

The currently available alternatives for the ordering `type` are

- `MATORDERINGNATURAL` - Natural
- `MATORDERINGND` - Nested Dissection
- `MATORDERING1WD` - One-way Dissection
- `MATORDERINGRCM` - Reverse Cuthill-McKee
- `MATORDERINGQMD` - Quotient Minimum Degree

These orderings can also be set through the options database.

Certain matrix formats may support only a subset of these; more options may be added. Check the manual pages for up-to-date information. All of these orderings are symmetric at the moment; ordering routines that are not symmetric may be added. Currently we support orderings only for sequential matrices.

Users can add their own ordering routines by providing a function with the calling sequence

```
int reorder(Mat A,MatOrderingType type,IS* rowperm,IS* colperm);
```

Here `A` is the matrix for which we wish to generate a new ordering, `type` may be ignored and `rowperm` and `colperm` are the row and column permutations generated by the ordering routine. The user registers the ordering routine with the command

```
MatOrderingRegister(MatOrderingType inname,char *path,char *sname,
                    PetscErrorCode (*reorder)(Mat,MatOrderingType,IS*,IS*)));
```

The input argument `inname` is a string of the user's choice, `iname` is either an ordering defined in `petscmat.h` or a users string, to indicate one is introducing a new ordering, while the output See the code in `src/mat/impls/order/sorder.c` and other files in that directory for examples on how the reordering routines may be written.

Once the reordering routine has been registered, it can be selected for use at runtime with the command line option `-pc_factor_mat_ordering_type sname`. If reordering directly, the user should provide the `name` as the second input argument of MatGetOrdering().

The following routines perform complete, in-place, symbolic, and numerical factorizations for symmetric and nonsymmetric matrices, respectively:

```
MatCholeskyFactor(Mat matrix,IS permutation,const MatFactorInfo *info);
MatLUFactor(Mat matrix,IS rowpermutation,IS columnpermutation,const MatFactorInfo *info);
```

The argument `info->fill > 1` is the predicted fill expected in the factored matrix, as a ratio of the original fill. For example, `info->fill=2.0` would indicate that one expects the factored matrix to have twice as many nonzeros as the original.

For sparse matrices it is very unlikely that the factorization is actually done in-place. More likely, new space is allocated for the factored matrix and the old space deallocated, but to the user it appears in-place because the factored matrix replaces the unfactored matrix.

The two factorization stages can also be performed separately, by using the out-of-place mode, first one obtains that matrix object that will hold the factor

```
MatGetFactor(Mat matrix,const MatSolverPackage package,MatFactorType ftype,Mat *factor);
```

and then performs the factorization

```
MatCholeskyFactorSymbolic(Mat factor,Mat matrix,IS perm,const MatFactorInfo *info);
MatLUFactorSymbolic(Mat factor,Mat matrix,IS rowperm,IS colperm,const MatFactorInfo *info);
MatCholeskyFactorNumeric(Mat factor,Mat matrix,const MatFactorInfo);
MatLUFactorNumeric(Mat factor,Mat matrix,const MatFactorInfo *info);
```

In this case, the contents of the matrix `result` is undefined between the symbolic and numeric factorization stages. It is possible to reuse the symbolic factorization. For the second and succeeding factorizations, one simply calls the numerical factorization with a new input `matrix` and the *same* factored `result` matrix. It is *essential* that the new input matrix have exactly the same nonzero structure as the original factored matrix. (The numerical factorization merely overwrites the numerical values in the factored matrix and does not disturb the symbolic portion, thus enabling reuse of the symbolic phase.) In general, calling `XXXFactorS`

ymbolic with a dense matrix will do nothing except allocate the new matrix; the XXXFactorNumeric routines will do all of the work.

Why provide the plain XXXfactor routines when one could simply call the two-stage routines? The answer is that if one desires in-place factorization of a sparse matrix, the intermediate stage between the symbolic and numeric phases cannot be stored in a result matrix, and it does not make sense to store the intermediate values inside the original matrix that is being transformed. We originally made the combined factor routines do either in-place or out-of-place factorization, but then decided that this approach was not needed and could easily lead to confusion.

We do not currently support sparse matrix factorization with pivoting for numerical stability. This is because trying to both reduce fill and do pivoting can become quite complicated. Instead, we provide a poor stepchild substitute. After one has obtained a reordering, with MatGetOrdering(Mat A,MatOrdering type,IS *row,IS *col) one may call

MatReorderForNonzeroDiagonal(Mat A,double tol,IS row, IS col);

which will try to reorder the columns to ensure that no values along the diagonal are smaller than tol in a absolute value. If small values are detected and corrected for, a nonsymmetric permutation of the rows and columns will result. This is not guaranteed to work, but may help if one was simply unlucky in the original ordering. When using the KSP solver interface the option -pc_factor_nonzeros_along_diagonal <tol> may be used. Here, tol is an optional tolerance to decide if a value is nonzero; by default it is $1.e-10$.

Once a matrix has been factored, it is natural to solve linear systems. The following four routines enable this process:

MatSolve(Mat A,Vec x, Vec y);
MatSolveTranspose(Mat A, Vec x, Vec y);
MatSolveAdd(Mat A,Vec x, Vec y, Vec w);
MatSolveTransposeAdd(Mat A, Vec x, Vec y, Vec w);

matrix A of these routines must have been obtained from a factorization routine; otherwise, an error will be generated. In general, the user should use the KSP solvers introduced in the next chapter rather than using these factorization and solve routines directly.

17.3 Unimportant Details of KSP

Again, virtually all users should use KSP through the KSP interface and, thus, will not need to know the details that follow.

It is possible to generate a Krylov subspace context with the command

KSPCreate(MPI_Comm comm,KSP *kps);

Before using the Krylov context, one must set the matrix-vector multiplication routine and the preconditioner with the commands

PCSetOperators(PC pc,Mat Amat,Mat Pmat);
KSPSetPC(KSP ksp,PC pc);

In addition, the KSP solver must be initialized with

KSPSetUp(KSP ksp);

Solving a linear system is done with the command

KSPSolve(KSP ksp,Vec b,Vec x);

Finally, the KSP context should be destroyed with

KSPDestroy(KSP *ksp);

It may seem strange to put the matrix in the preconditioner rather than directly in the KSP; this decision was the result of much agonizing. The reason is that for SSOR with Eisenstat's trick, and certain other preconditioners, the preconditioner has to change the matrix-vector multiply. This procedure could not be done cleanly if the matrix were stashed in the KSP context that PC cannot access.

Any preconditioner can supply not only the preconditioner, but also a routine that essentially performs a complete Richardson step. The reason for this is mainly SOR. To use SOR in the Richardson framework, that is,

$$u^{n+1} = u^n + B(f - Au^n),$$

is much more expensive than just updating the values. With this addition it is reasonable to state that *all* our iterative methods are obtained by combining a preconditioner from the `PC` package with a Krylov method from the `KSP` package. This strategy makes things much simpler conceptually, so (we hope) clean code will result. *Note*: We had this idea already implicitly in older versions of KSP, but, for instance, just doing Gauss-Seidel with Richardson in old KSP was much more expensive than it had to be. With PETSc this should not be a problem.

17.4 Unimportant Details of PC

Most users will obtain their preconditioner contexts from the KSP context with the command KSPGetPC(). It is possible to create, manipulate, and destroy PC contexts directly, although this capability should rarely be needed. To create a PC context, one uses the command

PCCreate(MPI_Comm comm,PC *pc);

The routine

PCSetType(PC pc,PCType method);

sets the preconditioner method to be used. The routine

PCSetOperators(PC pc,Mat Amat,Mat Pmat);

set the matrices that are to be used with the preconditioner. The routine

PCGetOperators(PC pc,Mat *Amat,Mat *Pmat);

returns the values set with PCSetOperators().

The preconditioners in PETSc can be used in several ways. The two most basic routines simply apply the preconditioner or its transpose and are given, respectively, by

PCApply(PC pc,Vec x,Vec y);
PCApplyTranspose(PC pc,Vec x,Vec y);

In particular, for a preconditioner matrix, `B`, that has been set via PCSetOperators(pc,Amat,Pmat), the routine PCApply(pc,x,y) computes $y = B^{-1}x$ by solving the linear system $By = x$ with the specified preconditioner method.

Additional preconditioner routines are

PCApplyBAorAB(PC pc,PCSide right,Vec x,Vec y,Vec work,int its);
PCApplyBAorABTranspose(PC pc,PCSide right,Vec x,Vec y,Vec work,int its);
PCApplyRichardson(PC pc,Vec x,Vec y,Vec work,PetscReal rtol,PetscReal atol, PetscReal dtol,PetscInt
maxits,PetscBool zeroguess,PetscInt *its,PCRichardsonConvergedReason*);

The first two routines apply the action of the matrix followed by the preconditioner or the preconditioner followed by the matrix depending on whether the `right` is `PC_LEFT` or `PC_RIGHT`. The final routine applies `its` iterations of Richardson's method. The last three routines are provided to improve efficiency for certain Krylov subspace methods.

A PC context that is no longer needed can be destroyed with the command

PCDestroy(PC *pc);

Chapter 18

Unstructured Grids in PETSc

This chapter introduces the DMPlex subclass of DM, which allows the user to handle unstructured grids using the generic DM interface for hierarchy and multi-physics. DMPlex was created to remedy a huge problem in all current PDE simulation codes, namely that the discretization was so closely tied to the data layout and solver that switching discretizations in the same code was not possible. Not only does this preclude the kind of comparison that is necessary for scientific investigation, but it makes library development (as opposed to monolithic applications) impossible.

18.1 Representing Unstructured Grids

The main advantage of DMPlex in representing topology is that it treats all the different pieces of a mesh, e.g. cells, faces, edges, vertices in exactly the same way. This allows the interface to be very small and simple, while remaining flexible and general. This also allows "dimension independent programming", which means that the same algorithm can be used unchanged for meshes of different shapes and dimensions.

All pieces of the mesh are treated as *points*, which are identified by PetscInts. A mesh is built by relating points to other points, in particular specifying a "covering" relation among the points. For example, an edge is defined by being covered by two vertices, and a triangle can be defined by begin covered by three edges (or even by three vertices). In fact, this structure has been known for a long time. It is a Hasse Diagram [31], which is a Directed Acyclic Graph (DAG) representing a cell complex using the covering relation. The graph edges represent the relation, which also encodes a poset.

For example, we can encode the doublet mesh below which can also be represented as the DAG To use the PETSc API, we first consecutively number the mesh pieces. The PETSc convention is to number

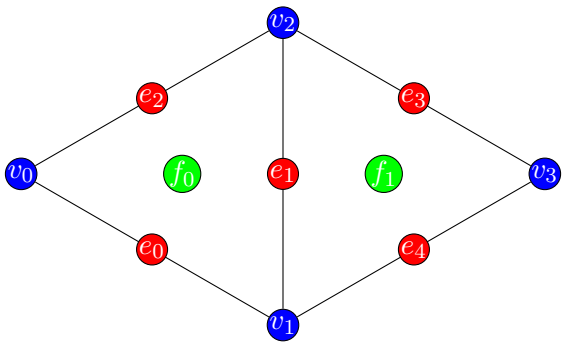

Figure 26: A 2D doublet mesh, two triangles sharing an edge.

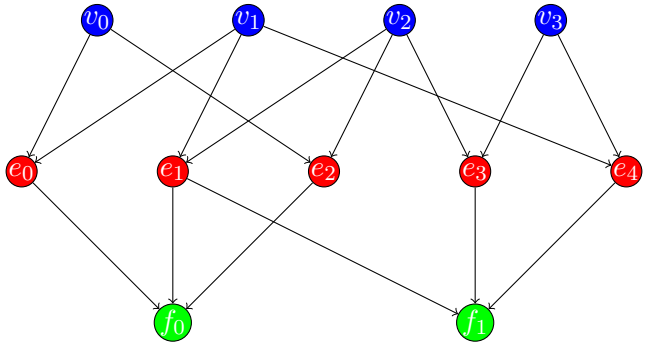

Figure 27: The Hasse diagram for our 2D doublet mesh, expressed as a DAG.

first cells, then vertices, then faces, and then edges, but the user is free to violate this convention. First, we declare the set of points present in a mesh,

```
DMPlexSetChart(dm, 0, 11);
```

We then define the covering relation, which we call the *cone*, which are also the in-edges in the DAG. In order to preallocate correctly, we first setup sizes,

```
DMPlexSetConeSize(dm, 0, 3);
DMPlexSetConeSize(dm, 1, 3);
DMPlexSetConeSize(dm, 6, 2);
DMPlexSetConeSize(dm, 7, 2);
DMPlexSetConeSize(dm, 8, 2);
DMPlexSetConeSize(dm, 9, 2);
DMPlexSetConeSize(dm, 10, 2);
DMSetUp(dm);
```

and then point values,

```
DMPlexSetCone(dm, 0, [6, 7, 8]);
DMPlexSetCone(dm, 1, [7, 9, 10]);
DMPlexSetCone(dm, 6, [2, 3]);
DMPlexSetCone(dm, 7, [3, 4]);
DMPlexSetCone(dm, 8, [4, 2]);
DMPlexSetCone(dm, 9, [4, 5]);
DMPlexSetCone(dm, 10, [5, 3]);
```

There is also an API for the dual relation, using DMPlexSetSupportSize() and DMPlexSetSupport(), but this can be calculated automatically by calling

```
DMPlexSymmetrize(dm);
```

In order to support efficient queries, we also want to construct fast search structures, indices, for the different types of points, which is done using

```
DMPlexStratify(dm);
```

18.2 Data on Unstructured Grids

The strongest links between solvers and discretizations are,

- the layout of data over the mesh,
- problem partitioning, and
- ordering of unknowns.

To enable modularity, we encode the operations above in simple data structures that can be understood by the linear algebra engine in PETSc without any reference to the mesh (topology) or discretization (analysis).

18.2.1 Data Layout

Data is associated to a mesh using the PetscSection object. A PetscSection can be thought of as a generalization of PetscLayout, in the same way that a fiber bundle is a generalization of the normal Euclidean basis used in linear algebra. With PetscLayout, we associate a unit vector (e_i) with every point in the space, and just divide up points between processes. Using PetscSection, we can associate a set of dofs, a small space $\{e_k\}$, with every point, and though our points must be contiguous like PetscLayout, they can be in any range $[\text{pStart}, \text{pEnd})$.

The sequence for setting up any PetscSection is the following:

1. Specify the chart,
2. Specify the number of dofs per point, and
3. Set up the PetscSection.

For example, using the mesh from Fig. 26, we can layout data for a continuous Galerkin P_3 finite element method,

```
PetscInt pStart, pEnd, cStart, cEnd, c, vStart, vEnd, v, eStart, eEnd, e;

DMPlexGetChart(dm, &pStart, &pEnd);
DMPlexGetHeightStratum(dm, 0, &cStart, &cEnd);
DMPlexGetHeightStratum(dm, 1, &eStart, &eEnd);
DMPlexGetDepthStratum(dm, 0, &vStart, &vEnd);
PetscSectionSetChart(s, pStart, pEnd);
for(c = cStart; c ¡ cEnd; ++c)
  PetscSectionSetDof(s, c, 1);
for(v = vStart; v ¡ vEnd; ++v)
  PetscSectionSetDof(s, v, 1);
for(e = eStart; e ¡ eEnd; ++e)
  PetscSectionSetDof(s, e, 2);
PetscSectionSetUp(s);
```

Now a PETSc local vector can be created manually using this layout,

```
PetscSectionGetStorageSize(s, &n);
VecSetSizes(localVec, n, PETSC_DETERMINE);
VecSetFromOptions(localVec);
```

however it is usually easier to use the DM directly, which also provides global vectors,

```
DMSetDefaultSection(dm, s);
DMGetLocalVector(dm, &localVec);
DMGetGlobalVector(dm, &globalVec);
```

18.2.2 Partitioning and Ordering

In exactly the same way as MatPartitioning and MatOrdering, we encode the results of a partition or order in an IS. However, the graph we are dealing with now is not the adjacency graph of the problem Jacobian, but the mesh itself. Once the mesh is partitioned and reordered, we used the data layout from a PetscSection to automatically derive a problem partitioning/ordering.

18.3 Evaluating Residuals

The evaluation of a residual, or Jacobian, for most discretizations has the following general form:

- Traverse the mesh, picking out pieces which in general overlap,
- Extract some values from the solution vector, associated with this piece,
- Calculate some values for the piece, and
- Insert these values into the residual vector

DMPlex separates these different concerns by passing sets of points, which are just integers, from mesh traversal routines to data extraction routines and back. In this way, the PetscSection which structures the data inside a Vec does not need to know anything about the mesh inside a DMPlex.

The most common mesh traversal is the transitive closure of a point, which is exactly the transitive closure of a point in the DAG using the covering relation. Note that this closure can be calculated orienting the arrows in either direction. For example, in a finite element calculation, we calculate an integral over the closure of each element, and then sum up the contributions to the basis function coefficients. The closure of the element can be expressed discretely as the transitive closure of the element point in the mesh DAG, where each point also has an orientation. Then we can retrieve the data using PetscSection methods,

```
PetscScalar *a;
PetscInt numPoints, *points = NULL, p;

VecGetArray(u, &a);
DMPlexGetTransitiveClosure(dm, cell, PETSC_TRUE, &numPoints, &points);
for(p = 0; p <= numPoints*2; p += 2) {
  PetscInt dof, off, d;

  PetscSectionGetDof(section, points[p], &dof);
  PetscSectionGetOffset(section, points[p], &off);
  for(d = 0; d <= dof; ++d) {
    myfunc(a[off+d]);
  }
}
DMPlexRestoreTransitiveClosure(dm, cell, PETSC_TRUE, &numPoints, &points);
VecRestoreArray(u, &a);
```

This operation is so common, that we have built a convenience method around it which returns the values in a contiguous array, correctly taking into account the orientations of various mesh points,

```
const PetscScalar *values;
PetscInt csize;

DMPlexVecGetClosure(dm, section, u, cell, &csize, &values);
/* Do integral in quadrature loop */
DMPlexVecRestoreClosure(dm, section, u, cell, &csize, &values);
DMPlexVecSetClosure(dm, section, residual, cell, &r, ADD_VALUES);
```

A simple example of this kind of calculation is in DMPlexComputeL2Diff(). Note that there is no restriction on the type of cell or dimension of the mesh in the code above, so it will work for polyhedral cells, hybrid meshes, and meshes of any dimension, without change. We can also reverse the covering relation, so that the code works for finite volume methods where we want the data from neighboring cells for each face

```
PetscScalar *a;
PetscInt points[2*2], numPoints, p, dofA, offA, dofB, offB;

VecGetArray(u, &a);
DMPlexGetTransitiveClosure(dm, cell, PETSC_FALSE, &numPoints, &points);
assert(numPoints == 2);
PetscSectionGetDof(section, points[0*2], &dofA);
PetscSectionGetDof(section, points[1*2], &dofB);
assert(dofA == dofB);
PetscSectionGetOffset(section, points[0*2], &offA);
PetscSectionGetOffset(section, points[1*2], &offB);
myfunc(a[offA], a[offB]);
VecRestoreArray(u, &a);
```

This kind of calculation is used in TS ex11.

18.4 Networks

Built on top of `DMPlex`, the `DMNetwork` subclass provides abstractions for representing general unstructured networks such as communication networks, power grid, computer networks, transportation networks, electrical circuits, graphs, and others.

18.4.1 Application flow

The general flow of an application code using `DMNetwork` is as follows:

1. Create a network object and a "component" library:

   ```
   DMNetworkCreate(MPI_Comm, DM*);
   ```

   creates an empty network object. A "component" is specific application data at a node/edge of the network required for its residual evaluation. For example, components could be resistor, inductor data for circuit applications, edge weights for graph problems, generator/transmission line data for power grids. Components are registered by calling

   ```
   DMNetworkRegisterComponent(DM,const char* name, PetscInt size, PetscInt* compkey);
   ```

Here, `name` is the component name,`size` is the size of component data type, and `compkey` is an integer key that can be used for setting/getting the component at a node or an edge.

2. Set network size (number of nodes, edges), edge connectivity.
3. Set the bare layout (graph) of the network

   DMNetworkLayoutSetUp(DM dm);

   DMNetworkSetSizes(DM dm, PetscInt nnodes, PetscInt nedges, PetscInt Nnodes, PetscInt Nedges);

   DMNetworkSetEdgeList(DM dm, int edgeconns[]);

4. Set components and number of variables for nodes/edges.

   DMNetworkAddComponent(DM dm, PetscInt p,PetscInt component,void* component);

   Multiple components can be added at a node/edge.

   DMNetworkSetNumVariables(DM dm,PetscInt p,PetscInt nvar);

   DMNetworkAddNumVariables(DM dm,PetscInt p,PetscInt nvar);

5. Signal network ready to be distributed.

   DMSetUp(DM dm);

6. Distribute the network (also moves components attached with nodes/edges)

   DMNetworkDistribute(DM oldDM, const char partitioner[], PetscInt overlap,DM *distDM);

7. Hook up the DM with the solver: KSPSetDM, SNESSetDM, TSSetDM

18.4.2 Utility functions

`DMNetwork` provides functions for obtaining iterators for nodes/edges, checking the "ghost" status of a node (vertex), and retrieving local/global indices of node/edge variables for inserting elements in vectors/-matrices.

DMNetworkGetEdgeRange(DM dm,PetscInt *eStart,PetscInt *eEnd);

DMNetworkGetVertexRange(DM dm,PetscInt *vStart, PetscInt *vEnd);

DMNetworkIsGhostVertex(DM dm,PetscInt p,PetscBool *isghost);

DMNetworkGetVariableOffset(DM dm,PetscInt p,PetscInt *offset);

DMNetworkGetVariableGlobalOffset(DM dm,PetscInt p,PetscInt *offsetg);

In network applications, one frequently needs to find the supporting edges for a node or the connecting nodes covering an edge. These can be obtained by the following two routines.

DMNetworkGetConnectedNodes(DM dm,PetscInt edge,const PetscInt *vertices[]);

DMNetworkGetSupportingEdges(DM dm,PetscInt vertex,PetscInt *nedges,const PetscInt *edges[]) ;

18.4.3 Retrieving components

The components set at nodes/edges are stored in a container that can be accessed by

DMNetworkGetComponentDataArray(DM dm,DMNetworkComponentGenericDataType **componentdataarray);

Using `componentdataarray`, individual components set at a node/edge can be retrieved by

DMNetworkGetComponentTypeOffset(DM dm,PetscInt p, PetscInt compnum, PetscInt *compkey, PetscInt *offset);

`compkey` is the key set by DMNetworkRegisterComponent. An example of accessing and retrieving the components at nodes is:

```
DMNetworkComponentGenericDataType *arr;
PetscInt Start, End, numcomps,offset,key,v,compnum;

DMNetworkGetComponentDataArray(dm, &arr);
DMNetworkGetVertexRange(dm,&Start,&End);
for(v=Start; v < End; v++) {
 DMNetworkGetNumComponents(dm,v,&numcomps);
 for(compnum=0; compnum < numcomps;compnum++) {
  DMNetworkGetComponentTypeOffset(dm,v,compnum,&key,&offset);
  compdata = (UserCompDataType)(arr+offset);
 }
}
```

Index

Bibliography

[1] U.M. Ascher, S.J. Ruuth, and R.J. Spiteri. Implicit-explicit Runge-Kutta methods for time-dependent partial differential equations. *Applied Numerical Mathematics*, 25:151–167, 1997.

[2] Satish Balay, William D. Gropp, Lois Curfman McInnes, and Barry F. Smith. Efficient management of parallelism in object oriented numerical software libraries. In E. Arge, A. M. Bruaset, and H. P. Langtangen, editors, *Modern Software Tools in Scientific Computing*, pages 163–202. Birkhauser Press, 1997.

[3] S. Boscarino, L. Pareschi, and G. Russo. Implicit-explicit Runge-Kutta schemes for hyperbolic systems and kinetic equations in the diffusion limit. Arxiv preprint arXiv:1110.4375, 2011.

[4] Peter N. Brown and Youcef Saad. Hybrid Krylov methods for nonlinear systems of equations. *SIAM J. Sci. Stat. Comput.*, 11:450–481, 1990.

[5] J.C. Butcher, Z. Jackiewicz, and W.M. Wright. Error propagation of general linear methods for ordinary differential equations. *Journal of Complexity*, 23(4-6):560–580, 2007.

[6] X.-C. Cai and M. Sarkis. A restricted additive Schwarz preconditioner for general sparse linear systems. Technical Report CU-CS 843-97, Computer Science Department, University of Colorado-Boulder, 1997. (accepted by SIAM J. of Scientific Computing).

[7] E.M. Constantinescu and A. Sandu. Extrapolated implicit-explicit time stepping. *SIAM Journal on Scientific Computing*, 31(6):4452–4477, 2010.

[8] J. E. Dennis, Jr. and Robert B. Schnabel. *Numerical Methods for Unconstrained Optimization and Nonlinear Equations*. Prentice-Hall, Englewood Cliffs, NJ, 1983.

[9] S. Eisenstat. Efficient implementation of a class of CG methods. *SIAM J. Sci. Stat. Comput.*, 2:1–4, 1981.

[10] S. C. Eisenstat and H. F. Walker. Choosing the forcing terms in an inexact Newton method. *SIAM J. Scientific Computing*, 17:16–32, 1996.

[11] R. Freund, G. H. Golub, and N. Nachtigal. *Iterative Solution of Linear Systems*, pages 57–100. Acta Numerica. Cambridge University Press, 1992.

[12] Roland W. Freund. A transpose-free quasi-minimal residual algorithm for non-Hermitian linear systems. *SIAM J. Sci. Stat. Comput.*, 14:470–482, 1993.

[13] F.X. Giraldo, J.F. Kelly, and E.M. Constantinescu. Implicit-explicit formulations of a three-dimensional nonhydrostatic unified model of the atmosphere (NUMA). *SIAM Journal on Scientific Computing*, 35(5):B1162–B1194, 2013.

[14] William Gropp and et. al. MPICH Web page. http://www.mpich.org.

[15] William Gropp, Ewing Lusk, and Anthony Skjellum. *Using MPI: Portable Parallel Programming with the Message Passing Interface*. MIT Press, 1994.

[16] Virginia Herrarte and Ewing Lusk. Studying parallel program behavior with Upshot. Technical Report ANL-91/15, Argonne National Laboratory, August 1991.

[17] Magnus R. Hestenes and Eduard Steifel. Methods of conjugate gradients for solving linear systems. *J. Research of the National Bureau of Standards*, 49:409–436, 1952.

[18] K.E. Jansen, C.H. Whiting, and G.M. Hulbert. A generalized-α method for integrating the filtered Navier–Stokes equations with a stabilized finite element method. *Computer Methods in Applied Mechanics and Engineering*, 190(3):305–319, 2000.

[19] C.A. Kennedy and M.H. Carpenter. Additive Runge-Kutta schemes for convection-diffusion-reaction equations. *Appl. Numer. Math.*, 44(1-2):139–181, 2003.

[20] D.I. Ketcheson. Highly efficient strong stability-preserving Runge–Kutta methods with low-storage implementations. *SIAM Journal on Scientific Computing*, 30(4):2113–2136, 2008.

[21] Jorge J. Moré, Danny C. Sorenson, Burton S. Garbow, and Kenneth E. Hillstrom. The MINPACK project. In Wayne R. Cowell, editor, *Sources and Development of Mathematical Software*, pages 88–111, 1984.

[22] MPI: A message-passing interface standard. *International J. Supercomputing Applications*, 8(3/4), 1994.

[23] L. Pareschi and G. Russo. Implicit-explicit Runge-Kutta schemes and applications to hyperbolic systems with relaxation. *Journal of Scientific Computing*, 25(1):129–155, 2005.

[24] M. Pernice and H. F. Walker. NITSOL: A Newton iterative solver for nonlinear systems. *SIAM J. Sci. Stat. Comput.*, 19:302–318, 1998.

[25] J. Rang and L. Angermann. New Rosenbrock W-methods of order 3 for partial differential algebraic equations of index 1. *BIT Numerical Mathematics*, 45(4):761–787, 2005.

[26] Youcef Saad and Martin H. Schultz. GMRES: A generalized minimal residual algorithm for solving nonsymmetric linear systems. *SIAM J. Sci. Stat. Comput.*, 7:856–869, 1986.

[27] A. Sandu, J.G. Verwer, J.G. Blom, E.J. Spee, G.R. Carmichael, and F.A. Potra. Benchmarking stiff ode solvers for atmospheric chemistry problems II: Rosenbrock solvers. *Atmospheric Environment*, 31(20):3459–3472, 1997.

[28] Barry F. Smith, Petter Bjørstad, and William D. Gropp. *Domain Decomposition: Parallel Multilevel Methods for Elliptic Partial Differential Equations*. Cambridge University Press, 1996.

[29] Peter Sonneveld. CGS, a fast Lanczos-type solver for nonsymmetric linear systems. *SIAM J. Sci. Stat. Comput.*, 10:36–52, 1989.

[30] H. A. van der Vorst. BiCGSTAB: A fast and smoothly converging variant of BiCG for the solution of nonsymmetric linear systems. *SIAM J. Sci. Stat. Comput.*, 13:631–644, 1992.

[31] Wikipedia. Hasse diagram, 2015. http://en.wikipedia.org/wiki/Hasse_diagram.

Mathematics and Computer Science Division
Argonne National Laboratory
9700 South Cass Avenue, Bldg. 240
Argonne, IL 60439

www.anl.gov