Enable students to develop new simulations with PETSc.

- Serial and Parallel
Enable students to develop new simulations with PETSc.

- Serial and Parallel
- Linear and Nonlinear
Enable students to develop new simulations with PETSc.

- Serial and Parallel
- Linear and Nonlinear
- Finite Difference and Finite Element
Enable students to develop new simulations with PETSc.

- Serial and Parallel
- Linear and Nonlinear
- Finite Difference and Finite Element
- Structured and Unstructured
Enable students to develop new simulations with PETSc.

- Serial and Parallel
- Linear and Nonlinear
- Finite Difference and Finite Element
- Structured and Unstructured
- Triangles and Hexes
Enable students to develop new simulations with PETSc.

- Serial and Parallel
- Linear and Nonlinear
- Finite Difference and Finite Element
- Structured and Unstructured
- Triangles and Hexes
- Optimal Solvers
Enable students to develop new simulations with PETSc.

- Serial and Parallel
- Linear and Nonlinear
- Finite Difference and Finite Element
- Structured and Unstructured
- Triangles and Hexes
- Optimal Solvers

Items in red not finished for tutorial
Outline

1. Creating a PETSc Application
2. Creating a Simple Mesh
3. Defining a Function
4. Discretization
5. Defining an Operator
6. Solving Systems of Equations
7. Optimal Solvers
8. The Undiscovered Country
Creating a PETSc Application
- What is PETSc?
- Who uses and develops PETSc?
- How can I get PETSc?
- How do I Configure PETSc?
- How do I Build PETSc?
- How do I run an example?
- How do I get more help?
- Minimal PETSc application

Creating a Simple Mesh

Defining a Function

Discretization
PETSc was developed as a Platform for Experimentation

We want to experiment with different

- Models
- Discretizations
- Solvers
- Algorithms (which blur these boundaries)
What I Need From You

- Tell me if you do not understand
- Tell me if an example does not work
- Suggest better wording or figures
- Followup problems at petsc-maint@mcs.anl.gov
How Can We Help?

- Provide documentation
- Answer email at petsc-maint@mcs.anl.gov
How Can We Help?

- Provide documentation
- Quickly answer questions

- Answer email at petsc-maint@mcs.anl.gov
How Can We Help?

- Provide documentation
- Quickly answer questions
- Help install

Answer email at petsc-maint@mcs.anl.gov
How Can We Help?

- Provide documentation
- Quickly answer questions
- Help install
- Guide large scale flexible code development
- Answer email at petsc-maint@mcs.anl.gov
Developing parallel, nontrivial PDE solvers that deliver high performance is still difficult and requires months (or even years) of concentrated effort.

PETSc is a toolkit that can ease these difficulties and reduce the development time, but it is not a black-box PDE solver, nor a silver bullet.

— Barry Smith
A freely available and supported research code
- Download from http://www.mcs.anl.gov/petsc
- Free for everyone, including industrial users
- Hyperlinked manual, examples, and manual pages for all routines
- Hundreds of tutorial-style examples
- Support via email: petsc-maint@mcs.anl.gov
- Usable from C, C++, Fortran 77/90, and Python
What is PETSc?

- Portable to any parallel system supporting MPI, including:
  - Tightly coupled systems
    - Cray T3E, SGI Origin, IBM SP, HP 9000, Sub Enterprise
  - Loosely coupled systems, such as networks of workstations
    - Compaq, HP, IBM, SGI, Sun, PCs running Linux or Windows

- PETSc History
  - Begun September 1991
  - Over 20,000 downloads since 1995 (version 2), currently 300 per month

- PETSc Funding and Support
  - Department of Energy
    - SciDAC, MICS Program, INL Reactor Program
  - National Science Foundation
    - CIG, CISE, Multidisciplinary Challenge Program
What Can We Handle?

- PETSc has run problems with over 500 million unknowns
PETSc has run problems with over 500 million unknowns
PETSc has run on over 6,000 processors efficiently
What Can We Handle?

- PETSc has run problems with over 500 million unknowns
- PETSc has run on over 6,000 processors efficiently
- PETSc applications have run at 2 Teraflops
  - LANL PFLOTRAN code
Who Uses PETSc?

- **Computational Scientists**
  - PyLith (TECTON), Underworld, Columbia group

- **Algorithm Developers**
  - Iterative methods and Preconditioning researchers

- **Package Developers**
  - SLEPc, TAO, MagPar, StGermain, DealII
The PETSc Team

- Bill Gropp
- Barry Smith
- Satish Balay
- Dinesh Kaushik
- Kris Buschelman
- Matt Knepley
- Hong Zhang
- Victor Eijkhout
- Lois McInnes
Downloading PETSc

- The latest tarball is on the PETSc site
  - We no longer distribute patches (everything is in the distribution)
- There is a Debian package
- There is a FreeBSD Port
- There is a Mercurial development repository
Cloning PETSc

The full development repository is open to the public
  - http://petsc.cs.iit.edu/petsc/petsc-dev
  - http://petsc.cs.iit.edu/petsc/BuildSystem

Why is this better?
  - You can clone to any release (or any specific ChangeSet)
  - You can easily rollback changes (or releases)
  - You can get fixes from us the same day

We also make release repositories available
  - http://petsc.cs.iit.edu/petsc/petsc-release-2.3.2
Unpacking PETSc

- Just clone development repository
  - `hg clone http://petsc.cs.iit.edu/petsc/petsc-dev petsc-dev`
  - `hg clone -rRelease-2.3.2 petsc-dev petsc-2.3.2`
  
  or

- Unpack the tarball
  - `tar xzf petsc.tar.gz`
You will need the Developer copy of PETSc:

- **Using Mercurial**
  
  `hg clone http://petsc.cs.iit.edu/petsc/petsc-dev`
  
  `cd petsc-dev/python`
  
  `hg clone http://petsc.cs.iit.edu/petsc/BuildSystem`

- **Manual download**
  

and the tutorial source code:

- **Using Mercurial**
  
  `hg clone http://petsc.cs.iit.edu/petsc/ParCFD07TutorialCode`

- **Manual download**
  
Configuring PETSc

- Set $PETSC_DIR$ to the installation root directory
- Run the configuration utility
  - $PETSC_DIR$/config/configure.py
  - $PETSC_DIR$/config/configure.py --help
  - $PETSC_DIR$/config/configure.py --download-mpich
- There are many examples on the installation page
- Configuration files are placed in $PETSC_DIR$/bmake/$PETSC_ARCH
  - $PETSC_ARCH$ has a default if not specified
You can easily reconfigure with the same options
   - ./bmake/$PETSC_ARCH/configure.py

Can maintain several different configurations
   - ./config/configure.py -PETSC_ARCH=linux-fast --with-debugging=0

All configuration information is in configure.log
   - ALWAYS send this file with bug reports
Starting in 2.2.1, some packages are automatically
  Downloaded
  Configured and Built (in $PETSC_DIR/externalpackages)
  Installed in PETSc

Currently works for
  PETSc documentation utilities (Sowing, lgrind, c2html)
  BLAS, LAPACK, BLACS, ScaLAPACK, PLAPACK
  MPICH, MPE, LAM
  ParMetis, Chaco, Jostle, Party, Scotch, Zoltan
  MUMPS, Spooles, SuperLU, SuperLU_Dist, UMFPack, pARMS
  BLOPEX, FFTW, SPRNG
  Prometheus, HYPRE, ML, SPAl
  Sundials
  Triangle, TetGen
  FIAT, FFC
Creating a PETSc Application  How do I Build PETSc?

Building PETSc

- Uses recursive make starting in cd $PETSC_DIR
  - make
  - Check build when done with make test
- Complete log for each build in make_log_$PETSC_ARCH
  - ALWAYS send this with bug reports
- Can build multiple configurations
  - PETSC_ARCH=linux-fast make
  - Libraries are in $PETSC_DIR/lib/$PETSC_ARCH/
- Can also build a subtree
  - cd src/snes; make
  - cd src/snes; make ACTION=libfast tree
Try running PETSc examples first
  - cd $PETSC_DIR/src/snes/examples/tutorials

Build examples using make targets
  - make ex5

Run examples using the make target
  - make runex5

Can also run using MPI directly
  - mpirun ./ex5 -snes_max_it 5
  - mpiexec ./ex5 -snes_monitor
Using MPI

- **Message Passing Interface** is:
  - a library for parallel communication
  - a system for launching parallel jobs (mpirun/mpiexec)
  - a community standard

- Launching jobs is easy
  - `mpiexec -np 4 ./ex5`

- You should never have to make MPI calls when using PETSc
  - Almost never
Communicator

- A context (or scope) for parallel communication ("Who can I talk to")
- There are two defaults:
  - yourself (PETSC_COMM_SELF),
  - and everyone launched (PETSC_COMM_WORLD)
- Can create new communicators by splitting existing ones
- Every PETSc object has a communicator

Point-to-point communication

- Happens between two processes (like in MatMult())

Reduction or scan operations

- Happens among all processes (like in VecDot())
Alternative Memory Models

- Single process (address space) model
  - OpenMP and threads in general
  - Fortran 90/95 and compiler-discovered parallelism
  - System manages memory and (usually) thread scheduling
  - Named variables refer to the same storage

- Single name space model
  - HPF, UPC
  - Global Arrays
  - Titanium
  - Named variables refer to the coherent values (distribution is automatic)

- Distributed memory (shared nothing)
  - Message passing
  - Names variables in different processes are unrelated
Common Viewing Options

- Gives a text representation
  - `-vec_view`
- Generally views subobjects too
  - `-snes_view`
- Can visualize some objects
  - `-mat_view_draw`
- Alternative formats
  - `-vec_view_binary`, `-vec_view_matlab`, `-vec_view_socket`
- Sometimes provides extra information
  - `-mat_view_info`, `-mat_view_info_detailed`
Common Monitoring Options

- Display the residual
  - `-ksp_monitor`, graphically `-ksp_xmonitor`
- Can disable dynamically
  - `-ksp_cancelmonitors`
- Does not display subsolvers
  - `-snes_monitor`
- Can use the true residual
  - `-ksp_truemonitor`
- Can display different subobjects
  - `-snes_vecmonitor`, `-snes_vecmonitor_update`, `-snes_vecmonitor_residual`
  - `-ksp_gmres_krylov_monitor`
- Can display the spectrum
  - `-ksp_singmonitor`
Getting More Help

- http://www.mcs.anl.gov/petsc
- Hyperlinked documentation
  - Manual
  - Manual pages for every method
  - HTML of all example code (linked to manual pages)
- FAQ
- Full support at petsc-maint@mcs.anl.gov
- High profile users
  - Lorena Barba
  - David Keyes
  - Xiao-Chuan Cai
  - Richard Katz
Following the Tutorial

Update to each new checkpoint (r0):

- `hg clone -r0 ParCFD07TutorialCode code-test`
- `hg update 0`

Build the executable with `make`, and then run:

- `make runbratu`
- `make debugbratu`
- `make valbratu`
- `make NP=2 runbratu`
- `make EXTRA_ARGS="-pc_type jacobi" runbratu`
Code Update

Update to Revision 0
Initialization

- Call PetscInitialize()
  - Setup static data and services
  - Setup MPI if it is not already
- Call PetscFinalize()
  - Calculates logging summary
  - Shutdown and release resources
- Checks compile and link
Command Line Processing

- Check for an option
  - PetscOptionsHasName()

- Retrieve a value
  - PetscOptionsGetInt(), PetscOptionsGetIntArray()

- Set a value
  - PetscOptionsSetValue()

- Check for unused options
  - -options_left

- Clear, alias, reject, etc.
Profiling

- Use `-log_summary` for a performance profile
  - Event timing
  - Event flops
  - Memory usage
  - MPI messages
- Call `PetscLogStagePush()` and `PetscLogStagePop()`
  - User can add new stages
- Call `PetscLogEventBegin()` and `PetscLogEventEnd()`
  - User can add new events
Outline

1. Creating a PETSc Application

2. Creating a Simple Mesh
   - Structured Meshes
   - Common PETSc Usage
   - Unstructured Meshes
   - 3D Meshes

3. Defining a Function

4. Discretization

5. Defining an Operator

6. Solving Systems of Equations
Higher Level Abstractions

The PETSc DA class is a topology and discretization interface.

- Structured grid interface
  - Fixed simple topology
- Supports stencils, communication, reordering
  - Limited idea of operators
- Nice for simple finite differences

The PETSc Mesh class is a topology interface.

- Unstructured grid interface
  - Arbitrary topology and element shape
- Supports partitioning, distribution, and global orders
Creating a Simple Mesh

Structured Meshes

Higher Level Abstractions

The PETSc DM class is a hierarchy interface.

- Supports multigrid
  - DMMG combines it with the MG preconditioner
- Abstracts the logic of multilevel methods

The PETSc Section class is a function interface.

- Functions over unstructured grids
  - Arbitrary layout of degrees of freedom
- Support distribution and assembly
Update to Revision 1
Creating a Simple Mesh

Structured Meshes

Creating a DA

DACreate2d(comm, wrap, type, M, N, m, n, dof, s, lm[], ln[], DA *da)

wrap: Specifies periodicity
- DA_NONPERIODIC, DA_XPERIODIC, DA_YPERIODIC, or DA_XYPERIODIC

type: Specifies stencil
- DA_STENCIL_BOX or DA_STENCIL_STAR

M/N: Number of grid points in x/y-direction

m/n: Number of processes in x/y-direction

dof: Degrees of freedom per node

s: The stencil width

lm/n: Alternative array of local sizes
- Use PETSC_NULL for the default
Ghost Values

To evaluate a local function $f(x)$, each process requires

- its local portion of the vector $x$
- its **ghost values**, bordering portions of $x$ owned by neighboring processes
### DA Global Numberings

<table>
<thead>
<tr>
<th>Proc 2</th>
<th>Proc 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>26</td>
</tr>
<tr>
<td>20</td>
<td>21</td>
</tr>
<tr>
<td>15</td>
<td>16</td>
</tr>
<tr>
<td>10</td>
<td>11</td>
</tr>
<tr>
<td>5</td>
<td>6</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

**Natural numbering**

<table>
<thead>
<tr>
<th>Proc 2</th>
<th>Proc 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>22</td>
</tr>
<tr>
<td>18</td>
<td>19</td>
</tr>
<tr>
<td>15</td>
<td>16</td>
</tr>
<tr>
<td>6</td>
<td>7</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

**PETSc numbering**
### DA Global vs. Local Numbering

- **Global**: Each vertex belongs to a unique process and has a unique id.
- **Local**: Numbering includes *ghost* vertices from neighboring processes.

**Local Numbering**

<table>
<thead>
<tr>
<th>Proc 2</th>
<th>Proc 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>12</td>
<td>13</td>
</tr>
<tr>
<td>8</td>
<td>9</td>
</tr>
<tr>
<td>4</td>
<td>5</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Proc 2</th>
<th>Proc 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Global Numbering**

<table>
<thead>
<tr>
<th>Proc 2</th>
<th>Proc 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>21</td>
<td>22</td>
</tr>
<tr>
<td>18</td>
<td>19</td>
</tr>
<tr>
<td>15</td>
<td>16</td>
</tr>
<tr>
<td>6</td>
<td>7</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Proc 2</th>
<th>Proc 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>28</td>
<td>29</td>
</tr>
<tr>
<td>26</td>
<td>27</td>
</tr>
<tr>
<td>24</td>
<td>25</td>
</tr>
<tr>
<td>13</td>
<td>14</td>
</tr>
<tr>
<td>11</td>
<td>12</td>
</tr>
<tr>
<td>9</td>
<td>10</td>
</tr>
</tbody>
</table>

**M. Knepley (ANL)**

Tutorial

SCAT '07 42 / 137
Creating a Simple Mesh
Structured Meshes

Viewing the DA

- make NP=1 EXTRA_ARGS="-da_view_draw -draw_pause -1" runbratu

- make NP=1 EXTRA_ARGS="-da_grid_x 10 -da_grid_y 10 -da_view_draw -draw_pause -1" runbratu

- make NP=4 EXTRA_ARGS="-da_grid_x 10 -da_grid_y 10 -da_view_draw -draw_pause -1" runbratu
Correctness Debugging

- Automatic generation of tracebacks
- Detecting memory corruption and leaks
- Optional user-defined error handlers
Interacting with the Debugger

- Launch the debugger
  - `--start_in_debugger [gdb, dbx, noxterm]`
  - `--on_error_attach_debugger [gdb, dbx, noxterm]`
- Attach the debugger only to some parallel processes
  - `--debugger_nodes 0,1`
- Set the display (often necessary on a cluster)
  - `--display khan.mcs.anl.gov:0.0`
Debugging Tips

- Putting a breakpoint in PetscError() can catch errors as they occur.
- PETSc tracks memory overwrites at the beginning and end of arrays.
  - The CHKMEMQ macro causes a check of all allocated memory.
  - Track memory overwrites by bracketing them with CHKMEMQ.
- PETSc checks for leaked memory.
  - Use PetscMalloc() and PetscFree() for all allocation.
  - Option -malloc_dump will print unfreed memory on PetscFinalize().
Memory Debugging

We can check for unfreed memory using:

```
make EXTRA_ARGS="-malloc_dump" runbratu
There is a leak!
```

All options can be seen using:

```
make EXTRA_ARGS="-help" runbratu
```
Update to Revision 2
Command Line Processing

- Check for an option
  - PetscOptionsHasName()
- Retrieve a value
  - PetscOptionsGetInt(), PetscOptionsGetIntArray()
- Set a value
  - PetscOptionsSetValue()
- Check for unused options
  - -options_left
- Clear, alias, reject, etc.
Update to Revision 3
Creating the Mesh

- Generic object
  - MeshCreate()
  - MeshSetMesh()

- File input
  - MeshCreatePCICE()
  - MeshCreatePyLith()

- Generation
  - MeshGenerate()
  - MeshRefine()
  - ALE::MeshBuilder::createSquareBoundary()

- Representation
  - ALE::SieveBuilder::buildTopology()
  - ALE::SieveBuilder::buildCoordinates()

- Partitioning and Distribution
  - MeshDistribute()
  - MeshDistributeByFace()
Update to Revision 4
Creating a Simple Mesh

Unstructured Meshes

Viewing the Mesh

- make NP=1 EXTRA_ARGS="-structured 0 -mesh_view.vtk" runbratu

- mayavi -d bratu.vtk -m SurfaceMap&

- make NP=4 EXTRA_ARGS="-structured 0 -mesh_view.vtk" runbratu

- Viewable using Mayavi
Creating a Simple Mesh

Unstructured Meshes

Refining the Mesh

- make NP=1 EXTRA_ARGS="-structured 0 -generate -mesh_view_vtk" runbratu
- make NP=1 EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.0625 -mesh_view_vtk" runbratu
- make NP=4 EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.0625 -mesh_view_vtk" runbratu
Parallel Sieves

- Sieves use *names*, not numberings
  - Numberings can be constructed on demand
- Overlaps relate names on different processes
  - An Overlap can be encoded by a Sieve
- Distribution of a Section pushes forward along the Overlap
  - Sieves are distributed as “cone” sections
The send overlap is above the receive overlap

Green points are remote process ranks

Arrow labels indicate remote process names
Update to Revision 5
viewing the 3d mesh

- make NP=1 EXTRA_ARGS="-dim 3 -da_view_draw -draw_pause -1" runbratu

- make NP=4 EXTRA_ARGS="-da_grid_x 5 -da_grid_y 5 -da_grid_z 5 -da_view_draw -draw_pause -1" runbratu

- make NP=1 EXTRA_ARGS="-dim 3 -structured 0 -generate -mesh_view_vtk" runbratu

- mayavi -d bratu.vtk -m SurfaceMap -f UserDefined:vtkExtractEdges

- make NP=4 EXTRA_ARGS="-dim 3 -structured 0 -generate -mesh_view_vtk -refinement_limit 0.01" runbratu
Outline

1. Creating a PETSc Application
2. Creating a Simple Mesh
3. Defining a Function
   - Vectors
   - Sections
4. Discretization
5. Defining an Operator
6. Solving Systems of Equations
7. Optimal Solvers
A DA is more than a Mesh

A DA contains **topology**, **geometry**, and an implicit Q1 **discretization**.

It is used as a template to create

- Vectors (functions)
- Matrices (linear operators)
The DA object contains only layout (topology) information
- All field data is contained in PETSc Vecs

Global vectors are parallel
- Each process stores a unique local portion
- DACreateGlobalVector(DA da, Vec *gvec)

Local vectors are sequential (and usually temporary)
- Each process stores its local portion plus ghost values
- DACreateLocalVector(DA da, Vec *lvec)
- includes ghost values!
Two-step process enables overlapping computation and communication

- **DAGlobalToLocalBegin(da, gvec, mode, lvec)**
  - gvec provides the data
  - mode is either INSERT_VALUES or ADD_VALUES
  - lvec holds the local and ghost values

- **DAGlobalToLocalEnd(da, gvec, mode, lvec)**
  - Finishes the communication

The process can be reversed with **DALocalToGlobal()**.
Defining a Function

Vectors

DA Local Function

The user provided function which calculates the nonlinear residual in 2D has signature

\[
\text{PetscErrorCode } (*\text{lfunc})(\text{DALocalInfo } *\text{info, PetscScalar } **\text{x, }
\text{PetscScalar } **\text{r, void } *\text{ctx})
\]

\textbf{info}: All layout and numbering information
\textbf{x}: The current solution
\>
\begin{itemize}
\item Notice that it is a multidimensional array
\end{itemize}
\textbf{r}: The residual
\textbf{ctx}: The user context passed to \text{DASetLocalFunction}()

The local DA function is activated by calling

\[
\text{SNESSetFunction(snes, r, SNESDADAF ormFunction, ctx)}
\]
Both the box stencil and star stencil are available.
PETSc provides

\[
\text{MatSetValuesStencil}(\text{Mat } A, m, \text{MatStencil } idxm[], \text{ n, MatStencil } idxn[], \text{ values }[], \text{ mode })
\]

- Each row or column is actually a MatStencil
  - This specifies grid coordinates and a component if necessary
  - Can imagine for unstructured grids, they are vertices
- The values are the same logically dense block in rows and columns
Update to Revision 6
Structured Functions

- Functions take values at the DA vertices
- Used as approximations to functions on the continuous domain
  - Values are really coefficients of linear basis
- User only constructs the local portion

make NP=2 EXTRAARGS="-run test -vec_view_draw -draw_pause -1" runbratu
**Sections** associate data to submeshes

- Name comes from section of a fiber bundle
  - Generalizes linear algebra paradigm
- Define `restrict()``, `update()`
- Define `complete()`
- Assembly routines take a Sieve and several Sections
  - This is called a Bundle
Section Types

Section can contain arbitrary values

- C++ interface is templated over value type
- C interface has two value types
  - SectionReal
  - SectionInt

Section can have arbitrary layout

- C++ interface can place unknowns on any Mesh entity (Sieve point)
  - Mesh::setupField() parametrized by Discretization and BoundaryCondition
- C interface has default layouts
  - MeshGetVertexSectionReal()
  - MeshGetCellSectionReal()
Update to Revision 7
Viewing the Section

- `make EXTRA_ARGS="-run test -structured 0 -vec_view_vtk" runbratu`
  - Produces `linear.vtk` and `cos.vtk`
- Viewable with MayaVi, exactly as with the mesh.
- `make NP=2 EXTRA_ARGS="-run test -structured 0 -vec_view_vtk -generate -refinement_limit 0.003125" runbratu`
  - Use `mayavi -d cos.vtk -m SurfaceMap -f WarpScalar`
Outline

1. Creating a PETSc Application
2. Creating a Simple Mesh
3. Defining a Function
4. Discretization
   - Finite Elements
   - Finite Differences
   - Evaluating the Error
5. Defining an Operator
6. Solving Systems of Equations
A *weak form* is the pairing of a function with an element of the *dual space*.

- Produces a number (by definition of the dual)
- Can be viewed as a “function” of the dual vector
- Used to define finite element solutions
- Require a dual space and integration rules

For example, for $f \in V$, we have the weak form

$$\int_{\Omega} \phi(x)f(x) dx \quad \phi \in V^*$$
Finite Element Integrator and Tabulator by Rob Kirby

http://www.fenics.org/fiat

FIAT understands

- Reference element shapes (line, triangle, tetrahedron)
- Quadrature rules
- Polynomial spaces
- Functionals over polynomials (dual spaces)
- Derivatives

User can build arbitrary elements by specifying the Ciarlet triple \((K, P, P')\)

FIAT is part of the FEniCS project, as is the PETSc Sieve module
Update to Revision 8
FIAT Integration

The quadrature.fiat file contains:
- An element (usually a family and degree) defined by FIAT
- A quadrature rule

It is run
- automatically by make, or
- independently by the user

It can take arguments
- --element_family and --element_order, or
- make takes variables ELEMENT and ORDER

Then make produces quadrature.h with:
- Quadrature points and weights
- Basis function and derivative evaluations at the quadrature points
- Integration against dual basis functions over the cell
- Local dofs for Section allocation
Boundary Conditions

For this problem, we allow two types of conditions

- **Dirichlet**
  \[ u|_\Gamma = g \]
  - Implemented by constraints on dofs in a Section

- **Neumann**
  \[ \nabla u \cdot \hat{n}|_\Gamma = h \]
  - Implemented by integration along the boundary (future)
Dirichlet Conditions (Essential BC)

- Explicit limitation of the approximation space
- Idea:
  - Maintain the same FEM interface (restrict(), update())
  - Allow direct access to reduced problem (contiguous storage)
- Implementation
  - Ignored by size() and update(), but restrict() works normally
  - Use updateBC() to define the boundary values
  - Use updateAll() to define both boundary and regular values
  - Points have a negative fiber dimension or
  - Dof are specified as constrained
Topological boundary is marked during generation

Cells bordering boundary are marked using `markBoundaryCells()`

To set values:
1. Loop over boundary cells
2. Loop over the element closure
3. For each boundary point, apply the corresponding functional $N_i$ to the function $g$

Functional application uses `BoundaryCondition::integrateDual()`

The functionals are generated and set using `BoundaryCondition::setDualIntegrator()`

`Mesh::setupField()` applies Dirichlet conditions automatically
Maps

We are interested in nonlinear maps $F : \mathbb{R}^n \rightarrow \mathbb{R}^n$.

- Can contain the action of differential operators
- Encapsulated in $\text{Rhs}_*(\cdot)$ methods
- Will later be used to form the residual of our system
Update to Revision 9
Section Assembly

First we do **local** operations:

- Loop over cells
- Compute cell geometry
- Integrate each basis function to produce an element vector
- Call `SectionUpdateAdd()`
  - Note that this updates the *closure* of the cell

Then we do **global** operations:

- `SectionComplete()` exchanges data across overlap
  - C just adds nonlocal values (C++ is flexible)
- C++ also allows completion over arbitrary overlaps
We use finite elements and a *Galerkin* formulation
- We calculate the residual $F(u) = -\Delta u - f$
- Correct basis/derivatives table chosen by setupQuadrature()
- Could substitute exact integrals for quadrature

```make
make NP=2 EXTRA_ARGS="-run test -structured 0 -vec_view_vtk -generate -refinement_limit 0.003125" runbratu
```

```make
make EXTRA_ARGS="-run test -dim 3 -structured 0 -generate -vec_view_vtk"
runbratu
```
Local (analytical)

- Discretization/Approximation
  - FEM integrals
  - FV fluxes
- Boundary conditions
Global and Local

Local (analytical)
- Discretization/Approximation
  - FEM integrals
  - FV fluxes
- Boundary conditions

Largely dim dependent
(e.g. quadrature)
Global and Local

Local (analytical)
- Discretization/Approximation
  - FEM integrals
  - FV fluxes
- Boundary conditions

Largely dim dependent
(e.g. quadrature)

Global (topological)
- Data management
  - Sections (local pieces)
  - Completions (assembly)
- Boundary definition
- Multiple meshes
  - Mesh hierarchies
Global and Local

Local (analytical)
- Discretization/Approximation
  - FEM integrals
  - FV fluxes
- Boundary conditions
Largely dim dependent (e.g. quadrature)

Global (topological)
- Data management
  - Sections (local pieces)
  - Completions (assembly)
- Boundary definition
- Multiple meshes
  - Mesh hierarchies
Largely dim independent (e.g. mesh traversal)
Difference Approximations

With finite differences, we approximate differential operators with difference quotients,

\[
\frac{\partial u(x)}{\partial x} \approx \frac{u(x+h) - u(x-h)}{2h}
\]

\[
\frac{\partial^2 u(x)}{\partial x^2} \approx \frac{u(x+h) - 2u(x) + u(x-h)}{h^2}
\]

The important property for the approximation is consistency, meaning

\[
\lim_{h \to 0} \frac{\partial u(x)}{\partial x} - \frac{u(x+h) - u(x-h)}{2h} = 0
\]

and in fact,

\[
\frac{\partial^2 u(x)}{\partial x^2} - \frac{u(x+h) - 2u(x) + u(x-h)}{h^2} \in O(h^2)
\]
Update to Revision 10
We cannot currently visualize the 3D results,

- make EXTRA_ARGS="-run test -vec_view_draw -draw_pause -1" runbratu
- make EXTRA_ARGS="-run test -da_grid_x 10 -da_grid_y 10 -vec_view_draw -draw_pause -1" runbratu
- make EXTRA_ARGS="-run test -dim 3 -vec_view" runbratu

but can check the ASCII output if necessary.
On two processes, I get a SEGV!

So we try running with:

- make NP=2 EXTRA_ARGS="-run test -vec_view_draw -draw_pause -1" debugbratu
On two processes, I get a SEGV!

So we try running with:

- `make NP=2 EXTRA_ARGS="-run test -vec_view_draw -draw_pause -1" debugbratu`
- Spawns one debugger window per process
On two processes, I get a **SEGV**!

So we try running with:

- make NP=2 EXTRA_ARGS="-run test -vec_view_draw -draw_pause -1" debugbratu
- Spawns one debugger window per process
- SEGV on access to ghost coordinates
On two processes, I get a SEGV!

So we try running with:

- `make NP=2 EXTRA_ARGS="-run test -vec_view_draw -draw_pause -1" debugbratu`
- Spawns one debugger window per process
- SEGV on access to ghost coordinates
- Fix by using a local ghosted vector
  - Update to Revision 11
On two processes, I get a **SEGV**!

So we try running with:

- `make NP=2 EXTRA_ARGS="-run test -vec_view_draw -draw_pause -1" debugbratu`
- Spawns one debugger window per process
- **SEGV** on access to ghost coordinates
- Fix by using a local ghosted vector
  - Update to Revision 11
- Notice
  - we already use ghosted assembly (completion) for FEM
  - FD does not need ghosted assembly
Representations of the Error

- A single number, the norm itself
- A number per element, the element-wise norm
- Injection into the finite element space

\[ e = \sum_{i} e_i \phi_i(x) \]  

- We calculate \( e_i \) by least-squares projection into \( P \)
Comparing solutions on different meshes can be problematic.

- Picture our solutions as functions defined over the entire domain
  - For FEM, \( \hat{u}(x) = \sum_i u_i \phi_i(x) \)
- After interpolation, the interpolant might not be the same function
- We often want to preserve thermodynamic bulk properties
  - Energy, stress energy, incompressibility, . . .
- Can constrain interpolation to preserve desirable quantities
  - Usually produces a saddlepoint system
Calculating the $L_2$ Error

We begin with a continuum field $u(x)$ and a finite element approximation

$$\hat{u}(x) = \sum_i \hat{u}_i \phi_i(x) \quad (2)$$

The FE theory predicts a convergence rate for the quantity

$$||u - \hat{u}||_2^2 = \sum_T \int_T dA (u - \hat{u})^2$$

$$= \sum_T \sum_q w_q |J| \left( u(q) - \sum_j \hat{u}_j \phi_j(q) \right)^2 \quad (4)$$

The estimate for linear elements is

$$||u - \hat{u}_h|| < Ch ||u|| \quad (6)$$
Update to Revision 12
Calculating the Error

- **Added CreateProblem()**
  - Define the global section
  - Setup exact solution and boundary conditions
- **Added CreateExactSolution()** to project the solution function
- **Added CheckError()** to form the error norm
  - Finite differences calculates a pointwise error
  - Finite elements calculates a normwise error
- **Added CheckResidual()** which uses our previous functionality
Checking the Error

- `make NP=2 EXTRA_ARGS="-run full -da_grid_x 10 -da_grid_y 10" runbratu`
- `make EXTRA_ARGS="-run full -dim 3" runbratu`
- `make EXTRA_ARGS="-run full -structured 0 -generate" runbratu`
- `make NP=2 EXTRA_ARGS="-run full -structured 0 -generate" runbratu`
- `make EXTRA_ARGS="-run full -structured 0 -generate -refinement_limit 0.03125" runbratu`
- `make EXTRA_ARGS="-run full -dim 3 -structured 0 -generate -refinement_limit 0.01" runbratu`

Notice that the FE error does not always vanish, since we are using information across the entire element. We can enrich our FE space:

- `rm quadrature.h; make ORDER=2`
- `make EXTRA_ARGS="-run full -structured 0 -generate -refinement_limit 0.03125" runbratu`
- `make EXTRA_ARGS="-run full -dim 3 -structured 0 -generate -refinement_limit 0.01" runbratu`
Defining an Operator

Outline

1. Creating a PETSc Application
2. Creating a Simple Mesh
3. Defining a Function
4. Discretization
5. Defining an Operator
6. Solving Systems of Equations
7. Optimal Solvers
8. The Undiscovered Country
Defining an Operator

DA Local Jacobian

The user provided function which calculates the Jacobian in 2D has signature

\[ \text{PetscErrorCode} \ (\text{*lfunc})(\text{DALocalInfo} \ (*info, \ \text{PetscScalar} \ (**x, \ \text{Mat} \ J, \ void \ *ctx)} \]

**info**: All layout and numbering information

**x**: The current solution

**J**: The Jacobian

**ctx**: The user context passed to `DASetLocalFunction()`

The local DA function is activated by calling

\[ \text{SNESSetJacobian(snes, J, J, SNESDADComputeJacobian, ctx)} \]
Update to Revision 13
- Evaluate only the local portion
  - No nice local array form without copies
- Use `MatSetValuesStencil()` to convert \((i,j,k)\) to indices
- `make NP=2 EXTRA_ARGS="-run test -da_grid_x 10 -da_grid_y 10 -mat_view_draw -draw_pause -1" runbratu`
- `make NP=2 EXTRA_ARGS="-run test -dim 3 -da_grid_x 5 -da_grid_y 5 -da_grid_z 5 -mat_view_draw -draw_pause -1" runbratu`
We evaluate the local portion just as with functions
Notice we use $J^{-1}$ to convert derivatives
Currently `updateOperator()` uses `MatSetValues()`
  - We need to call `MatAssemblyBegin/End()`
  - We should properly have `OperatorComplete()`
  - Also requires a Section, for layout, and a global variable order for PETSc index conversion

make `EXTRA_ARGS="-run test -structured 0 -mat_view_draw -draw_pause -1 -generate"` runbratu

make `NP=2 EXTRA_ARGS="-run test -structured 0 -mat_view_draw -draw_pause -1 -generate -refinement_limit 0.03125"` runbratu

make `EXTRA_ARGS="-run test -dim 3 -structured 0 -mat_view_draw -draw_pause -1 -generate"` runbratu
Update to Revision 14
Operator Assembly

- **Full assembly**
  - Aggregate along all element interfaces
  - Global sparse matrix

- **Stored assembly**
  - No aggregation along element interfaces
  - Store element matrices
  - Use a MATSHELL in the solve

- **Calculated assembly**
  - No aggregation along element interfaces
  - Calculate element matrices on the fly
  - Use a MATSHELL in the solve

- **Other alternatives...**
  - Aggregation along some element interfaces (local?)
  - Custom overlaps
  - Partial calculation
Solving Systems of Equations

Outline

1. Creating a PETSc Application
2. Creating a Simple Mesh
3. Defining a Function
4. Discretization
5. Defining an Operator
6. Solving Systems of Equations
   - Linear Equations
   - Nonlinear Equations
7. Optimal Solvers
Flow Control for a PETSc Application

Main Routine

- Timestepping Solvers (TS)
- Nonlinear Solvers (SNES)
- Linear Solvers (KSP)
- Preconditioners (PC)

Application Initialization
Function Evaluation
Jacobian Evaluation
Postprocessing

PETSc
The SNES interface is based upon callback functions

- SNESSetFunction()
- SNESetJacobian()

When PETSc needs to evaluate the nonlinear residual $F(x)$, the solver calls the user's function inside the application.

The user function get application state through the ctx variable. PETSc never sees application data.
SNES Function

The user provided function which calculates the nonlinear residual has signature

```c
PetscErrorCode (*func)(SNES snes, Vec x, Vec r, void *ctx)
```

- **x**: The current solution
- **r**: The residual
- **ctx**: The user context passed to `SNESSetFunction()`
  - Use this to pass application information, e.g. physical constants
SNES Jacobian

The user provided function which calculates the Jacobian has signature

\[
\text{PetscErrorCode} \ (\text{*func})(\text{SNES} \ \text{snes}, \ \text{Vec} \ \text{x}, \ \text{Mat} \ \text{*J}, \ \text{Mat} \ \text{*M}, \ \text{MatStructure} \ \text{*flag}, \ \text{void} \ \text{*ctx})
\]

\text{x}: \ The \ current \ solution
\text{J}: \ The \ Jacobian
\text{M}: \ The \ Jacobian \ preconditioning \ matrix \ (possibly \ J \ itself)
\text{ctx}: \ The \ user \ context \ passed \ to \ \text{SNESSetFunction}()

- Use this to pass application information, e.g. physical constants
- Possible MatrStructure values are:
  - SAME_NONZERO_PATTERN, DIFFERENT_NONZERO_PATTERN,
  - ...

Alternatively, you can use

- a builtin sparse finite difference approximation
- automatic differentiation
  - AD support via ADIC/ADIFOR (P. Hovland and B. Norris from ANL)
SNES Variants

- Line search strategies
- Trust region approaches
- Pseudo-transient continuation
- Matrix-free variants
PETSc can compute and explicitly store a Jacobian via 1st-order FD

- **Dense**
  - Activated by `-snes_fd`
  - Computed by `SNESDefaultComputeJacobian()`

- **Sparse via colorings**
  - Coloring is created by `MatFDColoringCreate()`
  - Computed by `SNESDefaultComputeJacobianColor()`

Can also use Matrix-free Newton-Krylov via 1st-order FD

- Activated by `-snes_mf` without preconditioning
- Activated by `-snes_mf_operator` with user-defined preconditioning
  - Uses preconditioning matrix from `SNESSetJacobian()`
Update to Revision 15
DMMG Integration with SNES

- DMMG supplies global residual and Jacobian to SNES
  - User supplies local version to DMMG
  - The `Rhs_*(\cdot)` and `Jac_*(\cdot)` functions in the example
- Allows automatic parallelism
- Allows grid hierarchy
  - Enables multigrid once interpolation/restriction is defined
- Paradigm is developed in unstructured work
  - Notice we have to scatter into contiguous global vectors (initial guess)
- Handle Neumann BC using `DMMGSetNullSpace()`
Solving the Dirichlet Problem: $P_1$

- make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -vec_view_vtk" runbratu
- make EXTRA_ARGS="-dim 3 -structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -vec_view_vtk" runbratu

The linear basis cannot represent the quadratic solution exactly
- make EXTRA_ARGS="-structured 0 -generate -ksp_monitor -snes_monitor -vec_view_vtk -refinement_limit 0.00125 -ksp_rtol 1.0e-9" runbratu

The error decreases with $h$
- make NP=2 EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.00125 -ksp_monitor -snes_monitor -vec_view_vtk -ksp_rtol 1.0e-9" runbratu
- make EXTRA_ARGS="-dim 3 -structured 0 -generate -refinement_limit 0.00125 -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -vec_view_vtk" runbratu

Notice that the preconditioner is weaker in parallel
Solving the Dirichlet Problem: $P_1$
Solving the Dirichlet Problem: $P_2$

- rm quadrature.h; make ORDER=2
- make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9" runbratu
- make EXTRA_ARGS="-dim 3 -structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9" runbratu
- Here we get the exact solution
- make EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.00125 -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9" runbratu
- Notice that the solution is only as accurate as the KSP tolerance
- make NP=2 EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.00125 -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9" runbratu
- make EXTRA_ARGS="-dim 3 -structured 0 -generate -refinement_limit 0.00125 -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9" runbratu
- Again the preconditioner is weaker in parallel
- Currently we have no system for visualizing higher order solutions
Alternative Assembly

- make EXTRA_ARGS="-structured 0 -generate -ksp_monitor -snes_monitor -ksp_rtol 1.0e-9 -assembly_type full -pc_type none" runbratu
- Since we cannot precondition without a matrix, we turn it off for comparison
- make EXTRA_ARGS="-structured 0 -generate -ksp_monitor -snes_monitor -ksp_rtol 1.0e-9 -assembly_type stored -pc_type none" runbratu
- Here we store all the element matrices
- make EXTRA_ARGS="-structured 0 -generate -ksp_monitor -snes_monitor -ksp_rtol 1.0e-9 -assembly_type calculated -pc_type none" runbratu
- This reduces storage, but increases computation
Solving the Dirichlet Problem: FD

- make EXTRA_ARGS="-snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -vec view draw -draw pause -1" runbratu

- Notice that we converge at the vertices, despite the quadratic solution

- make EXTRA_ARGS="-snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -da grid x 40 -da grid y 40 -vec view draw -draw pause -1" runbratu

- make NP=2 EXTRA_ARGS="-snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -da grid x 40 -da grid y 40 -vec view draw -draw pause -1" runbratu

- Again the preconditioner is weaker in parallel

- make NP=2 EXTRA_ARGS="-dim 3 -snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -da grid x 10 -da grid y 10 -da grid z 10" runbratu
Solving the Neumann Problem: $P_1$

- make EXTRAARGV="-structured 0 -generate -bc_type neumann -snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -vec_view_vtk" runbratu
- make EXTRAARGV="-dim 3 -structured 0 -generate -bc_type neumann -snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -vec_view_vtk" runbratu
- make EXTRAARGV="-structured 0 -generate -refinement_limit 0.00125 -bc_type neumann -snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -vec_view_vtk" runbratu

The error decreases with $h$

- make NP=2 EXTRAARGV="-structured 0 -generate -refinement_limit 0.00125 -bc_type neumann -snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -vec_view_vtk" runbratu
Solving the Neumann Problem: $P_3$

- `rm quadrature.h; make ORDER=3`
- `make EXTRA_ARGS="-structured 0 -generate -bc_type neumann -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9" runbratu`
- Here we get the exact solution
- `make EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.00125 -bc_type neumann -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9" runbratu`
- `make NP=2 EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.00125 -bc_type neumann -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9" runbratu`
The Louisville-Bratu-Gelfand Problem

\[ -\Delta u - \lambda e^u = f \]  

- Simplification of the Solid-Fuel Ignition Problem
- Also a nonlinear eigenproblem
- Exhibits a bifurcation at \( \lambda \approx 6.8 \)
- We will use Dirichlet conditions
We will have to alter

- The residual calculation, \( \text{Rhs}_*() \)
- The Jacobian calculation, \( \text{Jac}_*() \)
- The forcing function to match our chosen solution, \( \text{CreateProblem}() \)
Update to Revision 19
Solving the Bratu Problem: FD

- make EXTRAARGS="-snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -vec_view_draw -draw_pause -1 -lambda 0.4" runbratu

Notice that we converge at the vertices, despite the quadratic solution

- make NP=2 EXTRAARGS="-da_grid_x 40 -da_grid_y 40 -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -vec_view_draw -draw_pause -1 -lambda 6.8" runbratu

Notice the problem is more nonlinear near the bifurcation

- make NP=2 EXTRAARGS="-dim 3 -da_grid_x 10 -da_grid_y 10 -da_grid_z 10 -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 6.8" runbratu
Finding Problems

We switch to quadratic elements so that our FE solution will be exact

- `rm quadrature.h; make ORDER=2`
- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4" runbratu`

We do not converge!

Residual is zero, so the Jacobian could be wrong (try FD)

It works!

Entries are too big, we forgot to initialize the matrix

M. Knepley (ANL)
Tutorial
SCAT '07 121 / 137
Finding Problems

We switch to quadratic elements so that our FE solution will be exact

- `rm quadrature.h; make ORDER=2`
- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4" runbratu`

We do not converge!
Finding Problems

We switch to quadratic elements so that our FE solution will be exact

- `rm quadrature.h; make ORDER=2`
- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4" runbratu`

We do not converge!

- Residual is zero, so the Jacobian could be wrong (try FD)
- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4 -snes_fd" runbratu`
Finding Problems

We switch to quadratic elements so that our FE solution will be exact

- `rm quadrature.h; make ORDER=2`
- `make EXTRAARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4" runbratu`

**We do not converge!**

- Residual is zero, so the Jacobian could be wrong (try FD)
- `make EXTRAARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4 -snes_fd" runbratu`

**It works!**
Finding Problems

We switch to quadratic elements so that our FE solution will be exact

- `rm quadrature.h; make ORDER=2`
- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4" runbratu`

**We do not converge!**

- Residual is zero, so the Jacobian could be wrong (try FD)
- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4 -snes_fd" runbratu`

**It works!**

- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4 -snes_max_it 3 -mat_view" runbratu`
- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4 -snes_fd -mat_view" runbratu`
- Entries are too big, we forgot to initialize the matrix
Update to Revision 20
Solving the Bratu Problem: $P_2$

- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 0.4" runbratu`
- `make EXTRA_Args="-structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 6.8" runbratu`
- `make NP=2 EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.00125 -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 6.8" runbratu`
- `make EXTRA_Args="-dim 3 -structured 0 -generate -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 6.8" runbratu`
- `make EXTRA_Args="-dim 3 -structured 0 -generate -refinement_limit 0.00125 -snes_monitor -ksp_monitor -ksp_rtol 1.0e-9 -lambda 6.8" runbratu`
Solving the Bratu Problem: \( P_1 \)

- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -lambda 0.4" runbratu`

- `make EXTRA_ARGS="-structured 0 -generate -snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -lambda 6.8" runbratu`

- `make NP=2 EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.00125 -snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -lambda 6.8" runbratu`

- `make EXTRA_ARGS="-dim 3 -structured 0 -generate -refinement_limit 0.01 -snes_monitor -ksp_monitor -ksp rtol 1.0e-9 -lambda 6.8" runbratu`
Outline

1. Creating a PETSc Application
2. Creating a Simple Mesh
3. Defining a Function
4. Discretization
5. Defining an Operator
6. Solving Systems of Equations
7. Optimal Solvers
8. The Undiscovered Country
I will define *optimal* as an $O(N)$ solution algorithm.

These are generally hierarchical, so we need

- hierarchy generation
- assembly on subdomains
- restriction and prolongation
Multigrid is *optimal* in that it does $O(N)$ work for $||r|| < \epsilon$

- Brandt, Briggs, Chan & Smith
- Constant work per level
  - Sufficiently strong solver
  - Need a constant factor decrease in the residual
- Constant factor decrease in dof
  - Log number of levels
Structured Meshes

The DMMG allows multigrid with some simple options:

- `-dmmg_nlevels, -dmmg_view`
- `-pc mg_type, -pc mg_cycle_type`
- `-mg_levels_1_ksp_type, -dmmg_levels_1_pc_type`
- `-mg_coarse_ksp_type, -mg_coarse_pc_type`
Optimal Solvers

Unstructured Meshes

- Same DMMG options as the structured case
- Mesh refinement
  - Ruppert algorithm in Triangle and TetGen
- Mesh coarsening
  - Talmor-Miller algorithm in PETSc
- More advanced options
  - -dmmg_refine
  - -dmmg_hierarchy
- Current version only works for linear elements
make EXTRA_ARGS="-dmmg_nlevels 2 -dmmg_view -snes_monitor -ksp_monitor -ksp_rtol 1e-9" runbratu

Notice that the solver on each level can be customized
number of KSP iterations is approximately constant
make EXTRA_ARGS="-da_grid_x 10 -da_grid_y 10 -dmmg_nlevels 8 -dmmg_view -snes_monitor -ksp_monitor -ksp_rtol 1e-9" runbratu

Notice that there are over 1 million unknowns!

Coarsening is not currently implemented
Coarse Hierarchy

Hierarchy created using Talmor-Miller coarsening
Solving with Unstructured Multigrid

- make EXTRA_ARGS="-structured 0 -generate -bc_type neumann -dmmg_nlevels 2 -dmmg_view -snes_monitor -ksp_monitor -ksp_rtol 1e-9 -vec_view" runbratu

- Compare to explicitly refined solution

- make EXTRA_ARGS="-structured 0 -generate -bc_type neumann -snes_monitor -ksp_monitor -ksp_rtol 1e-9 -refinement_limit 0.0625 -vec_view" runbratu

- We would really like to coarsen an existing mesh

- make EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.03125 -bc_type neumann -dmmg_nlevels 3 -dmmg_refine 0 -dmmg_hierarchy -dmmg_view -snes_monitor -ksp_monitor -ksp_rtol 1e-9 -vec_view" runbratu

- make EXTRA_ARGS="-structured 0 -generate -refinement_limit 0.03125 -bc_type neumann -snes_monitor -ksp_monitor -ksp_rtol 1e-9 -vec_view" runbratu

- Notice that here we refine both meshes to the same level
Outline

1. Creating a PETSc Application
2. Creating a Simple Mesh
3. Defining a Function
4. Discretization
5. Defining an Operator
6. Solving Systems of Equations
7. Optimal Solvers
What We Have Not Covered

- Unstructured hexes
  - Structured hex FEM

- \textit{a posteriori} Error Estimation

- Exotic elements

- Semi-Lagrangian Schemes
What We Have Not Focused On

- Linear and Nonlinear Solvers
  - MANY other PETSc tutorials on this

- Unstructured mesh framework
  - Several preprints on Sieve architecture

- Structure of multilevel methods
  - Barry’s talk from SIAM PP 2006

- Preconditioning
  - Very problem dependent (best left to applications?)

- Scalability and Performance
  - Coming soon…
Documentation: http://www.mcs.anl.gov/petsc/docs
  - PETSc Users manual
  - Manual pages
  - Many hyperlinked examples
  - FAQ, Troubleshooting info, installation info, etc.
Publications: http://www.mcs.anl.gov/petsc/publications
  - Research and publications that make use PETSc
MPI Information: http://www.mpi-forum.org
Using MPI (2nd Edition), by Gropp, Lusk, and Skjellum
Domain Decomposition, by Smith, Bjorstad, and Gropp
Proof is not currently enough to examine solvers
