

Introduction to Firedrake

2022 年 10 月 27 日

目录

1	Poisson equation	2
1.1	Dirichlet 问题	3
1.1.1	简单算例	3
1.1.2	Firedrake 内建网格生成函数	4
1.1.3	UFL 表达式	5
1.1.4	函数空间创建	7
1.1.5	线性方程组参数设置	7
1.1.6	查看高斯积分公式	10
1.1.7	边界条件设置	11
1.1.8	Gmsh 网格边界设置	13
1.2	纯 Neumann 边界条件	15
1.2.1	Use nullspace of solve	15
1.2.2	Using Lagrange multiplier	17
1.3	计算收敛阶	19
1.3.1	生成网格序列	19
1.3.2	投影到细网格上的空间中	21
1.3.3	插值到细网格上的空间中	21
1.4	构造等参元	23
1.4.1	移动网格	23
1.4.2	简单映射边界点	24
1.4.3	同时移动边界单元的内点	25
1.4.4	数值实验	25
1.5	间断有限元方法	26
1.5.1	UFL 符号	26
1.5.2	UFL 测度	26
1.5.3	变分形式	27
1.6	Dirac Delta 函数	28
1.7	自由度映射关系	31
1.7.1	编号	31
1.7.2	有限元自由度	32

1.7.3	查看矩阵和向量 (PETSc)	33
2	NS 方程	33
2.1	函数空间	34
2.2	弱形式	34
2.3	定义 Solver	35
2.4	时间循环	35
2.4.1	Constant 用于时间依赖的表达式	36
2.5	ParaView 可视化计算结果	36
2.5.1	二维结果 (surf 图)	36
2.5.2	选择部分区域显示	36
3	多进程并行 (MPI 和 PETSc)	36
3.1	DMPlex	37
3.2	Star Forest	37
3.3	输出	39
3.4	communicator	40
4	Debug	41
4.1	常见问题	41
4.1.1	DIVERGED_LINEAR_SOLVE	41
4.1.2	PyErr_Occurred	41
4.1.3	Tips	41
4.2	调试 Python 代码	41
4.3	调试 C 代码 (gdb)	41
4.3.1	gdb 命令行说明	42
4.3.2	参数 (options)	42
4.3.3	gdb 命令:	42
4.3.4	示例 (调试 test.py)	42
4.4	并程序调试	43
4.4.1	PETSc 的参数 -start_in_debugger	43
4.4.2	工具 tmux-mpi	43
5	Appendix	44

1 Poisson equation

```
[1]: import matplotlib.pyplot
import matplotlib_inline
matplotlib_inline.backend_inline.set_matplotlib_formats('png', 'pdf') # for export pdf
```

1.1 Dirichlet 问题

求解如下 Poisson 方程

$$\begin{aligned} -\Delta u &= f & \text{in } \Omega, \\ u &= g_D & \text{on } \partial\Omega_D, \\ \frac{\partial u}{\partial n} &= g_N & \text{on } \partial\Omega_N, \end{aligned} \tag{1}$$

其中 $\partial\Omega_D \cap \partial\Omega_N = \partial\Omega$, 并且 $\int_{\partial\Omega_D} ds \neq 0$.

试验和测试函数空间

$$\begin{aligned} H_E^1 &:= \{u \in H^1 \mid u = g_D \text{ on } \partial\Omega_D\} \\ H_{E_0}^1 &:= \{u \in H^1 \mid u = 0 \text{ on } \partial\Omega_D\} \end{aligned} \tag{2}$$

变分问题

求解 $u \in H_E^1$, 使得

$$\int_{\Omega} \nabla u \cdot \nabla v = \int_{\Omega} f v + \int_{\partial\Omega_N} g_N v \quad \forall v \in H_{E_0}^1. \tag{3}$$

1.1.1 简单算例

- 区域 $\Omega = (0, 1) \times (0, 1)$,
- 右端项 $f = \sin(\pi x) \sin(\pi y)$
- 边界条件: $\partial\Omega_N = \emptyset$, $g_D = 0$ (齐次 Dirichlet)

```
[2]: from firedrake import *
import matplotlib.pyplot as plt

N = 8
test_mesh = RectangleMesh(nx=N, ny=N, Lx=1, Ly=1)
x, y = SpatialCoordinate(test_mesh)
f = sin(pi*x)*sin(pi*y)
g = Constant(0)

V = FunctionSpace(test_mesh, 'CG', degree=1)

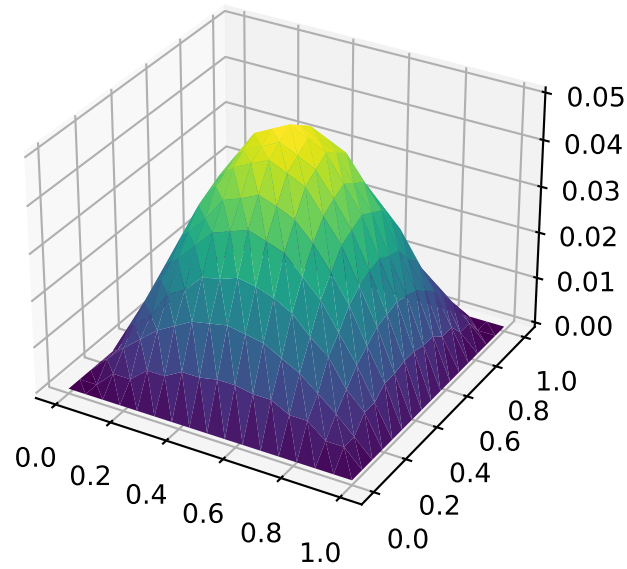
u, v = TrialFunction(V), TestFunction(V)
a = inner(grad(u), grad(v))*dx
L = inner(f, v)*dx # or f*v*dx

bc = DirichletBC(V, g=g, sub_domain='on_boundary')

u_h = Function(V, name='u_h')
solve(a == L, u_h, bcs=bc) # 有不同求解方式, 可添加求解参数
# solve(a == L, u_h, bcs=(bc,))

fig, ax = plt.subplots(figsize=[4, 4], subplot_kw=dict(projection='3d'))
trisurf(u_h, axes=ax)
```

```
[2]: <mpl_toolkits.mplot3d.art3d.Poly3DCollection at 0x7fa87ed27400>
```



1.1.2 Firedrake 内建网格生成函数

UnitDiskMesh, IntervalMesh, RectangleMesh, CubeMesh ...

```
[3]: from firedrake import utility_meshes
      from pprint import pprint

      pprint(utility_meshes.__all__)
```

```
['IntervalMesh',
 'UnitIntervalMesh',
 'PeriodicIntervalMesh',
 'PeriodicUnitIntervalMesh',
 'UnitTriangleMesh',
 'RectangleMesh',
 'TensorRectangleMesh',
 'SquareMesh',
 'UnitSquareMesh',
 'PeriodicRectangleMesh',
 'PeriodicSquareMesh',
 'PeriodicUnitSquareMesh',
 'CircleManifoldMesh',
 'UnitDiskMesh',
 'UnitTetrahedronMesh',
 'BoxMesh',
 'CubeMesh',
 'UnitCubeMesh',
 'PeriodicBoxMesh',
 'PeriodicUnitCubeMesh',
 'IcosahedralSphereMesh',
 'UnitIcosahedralSphereMesh',
```

```
'OctahedralSphereMesh',
'UnitOctahedralSphereMesh',
'CubedSphereMesh',
'UnitCubedSphereMesh',
'TorusMesh',
'CylinderMesh']
```

查看帮助 1. ?<fun-name> 2. help(<fun-name>)

[4]: ?CubeMesh

Signature:

```
CubeMesh(
    nx,
    ny,
    nz,
    L,
    reorder=None,
    distribution_parameters=None,
    comm=<mpi4py.MPI.Intracomm object at 0x7fa88bdc2f10>,
    name='firedrake_default',
)
```

Call signature: CubeMesh(*args, **kwargs)

Type: cython_function_or_method

String form: <cyfunction CubeMesh at 0x7fa884559c70>

File: ~/software/firedrake-mini-petsc/src/firedrake/firedrake/utility_meshes.py

Docstring:

Generate a mesh of a cube

```
:arg nx: The number of cells in the x direction
:arg ny: The number of cells in the y direction
:arg nz: The number of cells in the z direction
:arg L: The extent in the x, y and z directions
:kwarg reorder: (optional), should the mesh be reordered?
:kwarg comm: Optional communicator to build the mesh on (defaults to
    COMM_WORLD).
:kwarg name: Optional name of the mesh.
```

The boundary surfaces are numbered as follows:

```
* 1: plane x == 0
* 2: plane x == L
* 3: plane y == 0
* 4: plane y == L
* 5: plane z == 0
* 6: plane z == L
```

1.1.3 UFL 表达式

算子 DOC: https://fenics.readthedocs.io/projects/ufl/en/latest/manual/form_language.html#tensor-algebra-operators)

1. dot

张量缩并, dot(u, v) 对 u 的最后一个维度和 v 的第一个维度做缩并.

2. inner

张量内积 (分量对应乘积之和). 对第二个张量取复共轭.

3. grad and nabla_grad

1. grad

对张量求导, 新加维度为最后一个维度.

1. scalar

$$\text{grad}(u) = \nabla u = \frac{\partial u}{\partial x_i} \mathbf{e}_i$$

2. vector

$$\text{grad}(\mathbf{v}) = \nabla \mathbf{v} = \frac{\partial v_i}{\partial x_j} \mathbf{e}_i \otimes \mathbf{e}_j$$

3. tensor

设 \mathbf{T} 为秩为 r 的张量, 那么

$$\text{grad}(\mathbf{T}) = \nabla \mathbf{T} = \frac{\partial \mathbf{T}_\ell}{\partial x_i} \mathbf{e}_{\ell_1} \otimes \cdots \otimes \mathbf{e}_{\ell_r} \otimes \mathbf{e}_i$$

其中 ℓ 是长度为 r 的多指标 (multi-index).

2. nabla_grad

类似 grad, 不过新加维度为第一个维度

1. scalar (same with grad)

$$\text{nabla_grad}(u) = \nabla u = \frac{\partial u}{\partial x_i} \mathbf{e}_i$$

2. vector

$$\text{nabla_grad}(\mathbf{v}) = (\nabla \mathbf{v})^T = \frac{\partial v_j}{\partial x_i} \mathbf{e}_i \otimes \mathbf{e}_j$$

3. tensor

设 \mathbf{T} 为秩为 r 的张量, 那么

$$\text{nabla_grad}(\mathbf{T}) = \frac{\partial \mathbf{T}_\ell}{\partial x_i} \mathbf{e}_i \otimes \mathbf{e}_{\ell_1} \otimes \cdots \otimes \mathbf{e}_{\ell_r}$$

4. div and nabla_div

1. div

对最后一个维度的偏导数进行缩并.

设 \mathbf{T} 为秩为 r 的张量, 那么

$$\text{div}(\mathbf{T}) = \sum_i \frac{\partial \mathbf{T}_{\ell_1 \ell_2 \cdots \ell_{r-1} i}}{\partial x_i} \mathbf{e}_{\ell_1} \otimes \cdots \otimes \mathbf{e}_{\ell_{r-1}}$$

2. nabla_div

类似 div, 不过对第一个维度的偏导数进行缩并.

5. 两个表达式:

1. $(u \cdot \nabla)v \rightarrow \text{dot}(u, \text{nabla_grad}(v))$ or $\text{dot}(\text{grad}(v), u)$
2. $\Delta u \rightarrow \text{div}(\text{grad}(u))$

非线性函数 https://fenics.readthedocs.io/projects/ufl/en/latest/manual/form_language.html#basic-nonlinear-functions

- `abs`, `sign`
- `pow`, `sqrt`
- `exp`, `ln`
- `cos`, `sin`, ...
- ...

Measures

1. `dx`: the interior of the domain Ω (`dx`, cell integral);
2. `ds`: the boundary $\partial\Omega$ of Ω (`ds`, exterior facet integral);
3. `dS`: the set of interior facets Γ (`dS`, interior facet integral).

在区域内部的边界上积分时, 需要使用 `dS` 并使用限制算子 `+` 或 `-`, 如:

```
a = u('+')*v('+')*dS
```

1.1.4 函数空间创建

- `FunctionSpace` 标量函数空间
- `VectorFunctionSpace` 向量函数空间
- `MixedFunctionSpace` 混合空间

支持的单元类型: `CG`, `DG`, `RT`, `BDM`, ... (<https://firedrakeproject.org/variational-problems.html#supported-finite-elements>)

1.1.5 线性方程组参数设置

三种求解方程组 **Coding 方式** 仍然以上述 Poisson 方程为例: [Poisson Example](#)

可以使用 `%load` 加载文件内容到 notebook 中

```
%load poisson_example1.py
```

```
[5]: # %load poisson_example1.py
from firedrake import *
from firedrake.petsc import PETSc

methods = ['solve',
           'assemble',
           'LinearVariationalSolver']

# Get commandline args
opts = PETSc.Options()
case_index = opts.getInt('case_index', default=0)
```

```

if case_index < 0 or case_index > 2:
    raise Exception('Case index must be in [0, 2]')

case = methods[case_index]

N = 8
test_mesh = RectangleMesh(nx=N, ny=N, Lx=1, Ly=1)
x, y = SpatialCoordinate(test_mesh)
f = sin(pi*x)*sin(pi*y)
g = Constant(0)

V = FunctionSpace(test_mesh, 'CG', degree=1)

u, v = TrialFunction(V), TestFunction(V)

a = inner(grad(u), grad(v))*dx
L = inner(f, v)*dx          # or f*v*dx

bc = DirichletBC(V, g=g, sub_domain='on_boundary')

u_h = Function(V, name='u_h')

if case == 'solve':
    PETSc.Sys.Print('Case: solve')
    # solve(a == L, u_h, bcs=bc)
    solve(a == L, u_h, bcs=bc,
          solver_parameters={          # 设置方程组求解算法
              'ksp_view': None,
              'ksp_type': 'preonly',
              'pc_type': 'lu',
              'pc_factor_mat_solver_type': 'mumps'
          },
          options_prefix='test'      # 命令行参数前缀
    )

elif case == 'assemble':
    PETSc.Sys.Print('Case: assemble')
    A = assemble(a, bcs=bc)
    b = assemble(L, bcs=bc)
    solve(A, u_h, b,
          options_prefix='test'
    )

elif case == 'LinearVariationalSolver':
    PETSc.Sys.Print('Case: LinearVariationalSolver')
    problem = LinearVariationalProblem(a, L, u_h, bcs=bc)
    solver = LinearVariationalSolver(problem,
                                     solver_parameters={
                                         # 'ksp_view': None,
                                         'ksp_monitor': None,
                                         'ksp_converged_reason': None,
                                         'ksp_type': 'cg',
                                         'pc_type': 'none'
                                     },
                                     options_prefix='test')

    solver.solve()

```



```

else:
    raise Exception(f'Unknow case: {case}')

File('pvd/poisson_example.pvd').write(u_h)
print('Done!')

```

Case: solve
Done!

- KSP [scalable linear equations solvers, Krylov subspace solver with preconditioner](https://petsc.org/main/docs/manual/ksp/#tab-kspdefaults)
参数: <https://petsc.org/main/docs/manual/ksp/#tab-kspdefaults>
- PC
参数: <https://petsc.org/main/docs/manual/ksp/#tab-pcdefaults>
– 外部包 pc 参数: <https://petsc.org/main/docs/manual/ksp/#tab-externaloptions>

命令行参数 参数说明

1. mat_type: aij 或 matfree
2. ksp_type: 设置迭代法
3. pc_type: 设置预处理方式
4. ksp_monitor: 输出每步迭代的残差
5. ksp_view: 迭代完成后输出 ksp 的设置等内容
6. ksp_converged_reason: 输出收敛的原因

LU 分解参数设置

Ref: <https://petsc.org/release/src/dm/impls/stag/tutorials/ex4.c.html>

```
-ksp_type -pc_type lu -pc_factor_mat_solver_type mumps
```

这里 pc_factor_mat_solver_type 设置 LU 分解使用的 package (如 petsc, mumps, umfpack, superlu). 其他选项见: <https://petsc.org/release/docs/manualpages/Mat/MatSolverType/>

多重网格

<https://nbviewer.org/github/firedrakeproject/firedrake/blob/master/docs/notebooks/07-geometric-multigrid.ipynb>

终端演示: 设置命令行参数控制线性方程组的求解

```
python poisson_example1.py -case solve \
    -ksp_monitor -ksp_converged_reason \
    -ksp_type cg -pc_type jacobi
```

```
python poisson_example1.py -case assemble \
    -ksp_monitor -ksp_converged_reason \
    -ksp_type gmres -pc_type none
```

```
python poisson_example1.py -case LinearVariationalSolver \
    -ksp_monitor -ksp_converged_reason \
    -ksp_type minres -pc_type none
```

1.1.6 查看高斯积分公式

```
[6]: import FIAT
import finat

ref_cell = FIAT.reference_element.UFCTriangle()

from pprint import pprint
ret = {}
for i in range(0, 5):
    qrule = finat.quadrature.make_quadrature(ref_cell, i)
    ret[i] = {'points': qrule.point_set.points, 'weights': qrule.weights}

pprint(ret)

{0: {'points': array([[0.33333333, 0.33333333]]), 'weights': array([0.5])},
 1: {'points': array([[0.33333333, 0.33333333]]), 'weights': array([0.5])},
 2: {'points': array([[0.16666667, 0.16666667],
                    [0.16666667, 0.66666667],
                    [0.66666667, 0.16666667]]),
    'weights': array([0.16666667, 0.16666667, 0.16666667])},
 3: {'points': array([[0.65902762, 0.23193337],
                    [0.65902762, 0.10903901],
                    [0.23193337, 0.65902762],
                    [0.23193337, 0.10903901],
                    [0.10903901, 0.65902762],
                    [0.10903901, 0.23193337]]),
    'weights': array([0.08333333, 0.08333333, 0.08333333, 0.08333333,
                    0.08333333, 0.08333333])},
 4: {'points': array([[0.81684757, 0.09157621],
                    [0.09157621, 0.81684757],
                    [0.09157621, 0.09157621],
                    [0.10810302, 0.44594849],
                    [0.44594849, 0.10810302],
                    [0.44594849, 0.44594849]]),
    'weights': array([0.05497587, 0.05497587, 0.05497587, 0.11169079,
                    0.11169079, 0.11169079])}}
```

显示选择积分公式

```
[7]: set_log_level(CRITICAL) # Disable warnings

mesh = RectangleMesh(nx=8, ny=8, Lx=1, Ly=1)
V = FunctionSpace(mesh, 'CG', 1)
cell = V.finat_element.cell

x, y = SpatialCoordinate(mesh)
f = x**3 + y**4 + x**2*y**2

for i in range(0, 5):
    qrule = finat.quadrature.make_quadrature(ref_cell, i)
    ret[i] = {'points': qrule.point_set.points, 'weights': qrule.weights}
    v = assemble(f*dx(rule=qrule))
    print(f'degree={i}, v = {v}', )

print('Default: v =', assemble(f*dx(rule=None)))
```

```

degree=0, v = 0.5579329125675148
degree=1, v = 0.5579329125675148
degree=2, v = 0.5611099431544168
degree=3, v = 0.5611100938585061
degree=4, v = 0.5611111111111102
Default: v = 0.5611111111111102

```

1.1.7 边界条件设置

内建网格边界编号

RectangleMesh:

- 1: plane $x == 0$
- 2: plane $x == L_x$
- 3: plane $y == 0$
- 4: plane $y == L_y$

```

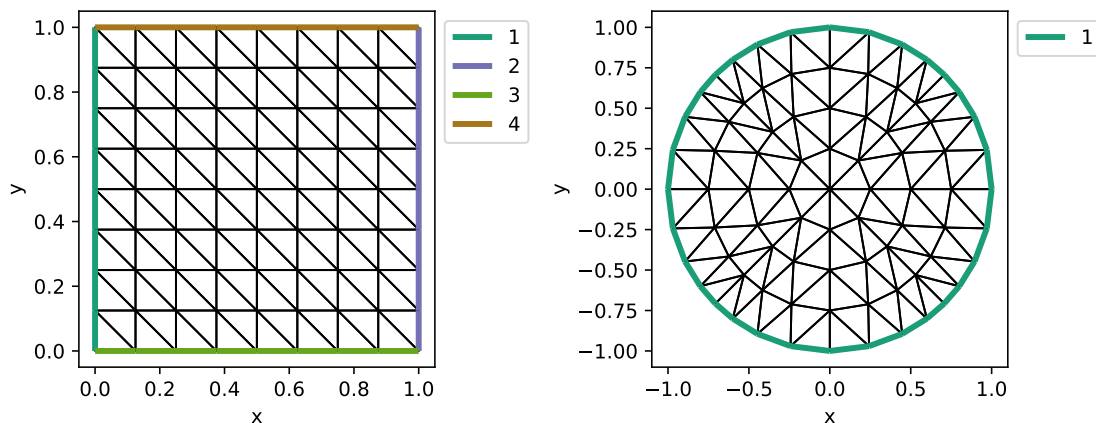
[8]: from firedrake import *
import matplotlib.pyplot as plt

def plot_mesh_with_label(mesh, axes=None):
    if axes is None:
        fig, axes = plt.subplots(figsize=[4, 4])
        triplot(mesh, axes=axes, boundary_kw={'lw': 3})
        axes.set_aspect('equal')
        # ax.set_axis_off()
        axes.legend(loc='upper left', bbox_to_anchor=(1, 1))
        axes.set_xlabel('x')
        axes.set_ylabel('y')

N = 8
rect_mesh = RectangleMesh(nx=N, ny=N, Lx=1, Ly=1)
circ_mesh = UnitDiskMesh(2)

fig, ax = plt.subplots(1, 2, figsize=[8, 4])
plot_mesh_with_label(rect_mesh, axes=ax[0])
plot_mesh_with_label(circ_mesh, axes=ax[1])
fig.tight_layout()

```



设置边界条件

```
[9]: N = 8
test_mesh = RectangleMesh(nx=N, ny=N, Lx=1, Ly=1)
x, y = SpatialCoordinate(test_mesh)

g = x*2 + y*2
V = FunctionSpace(test_mesh, 'CG', degree=1)

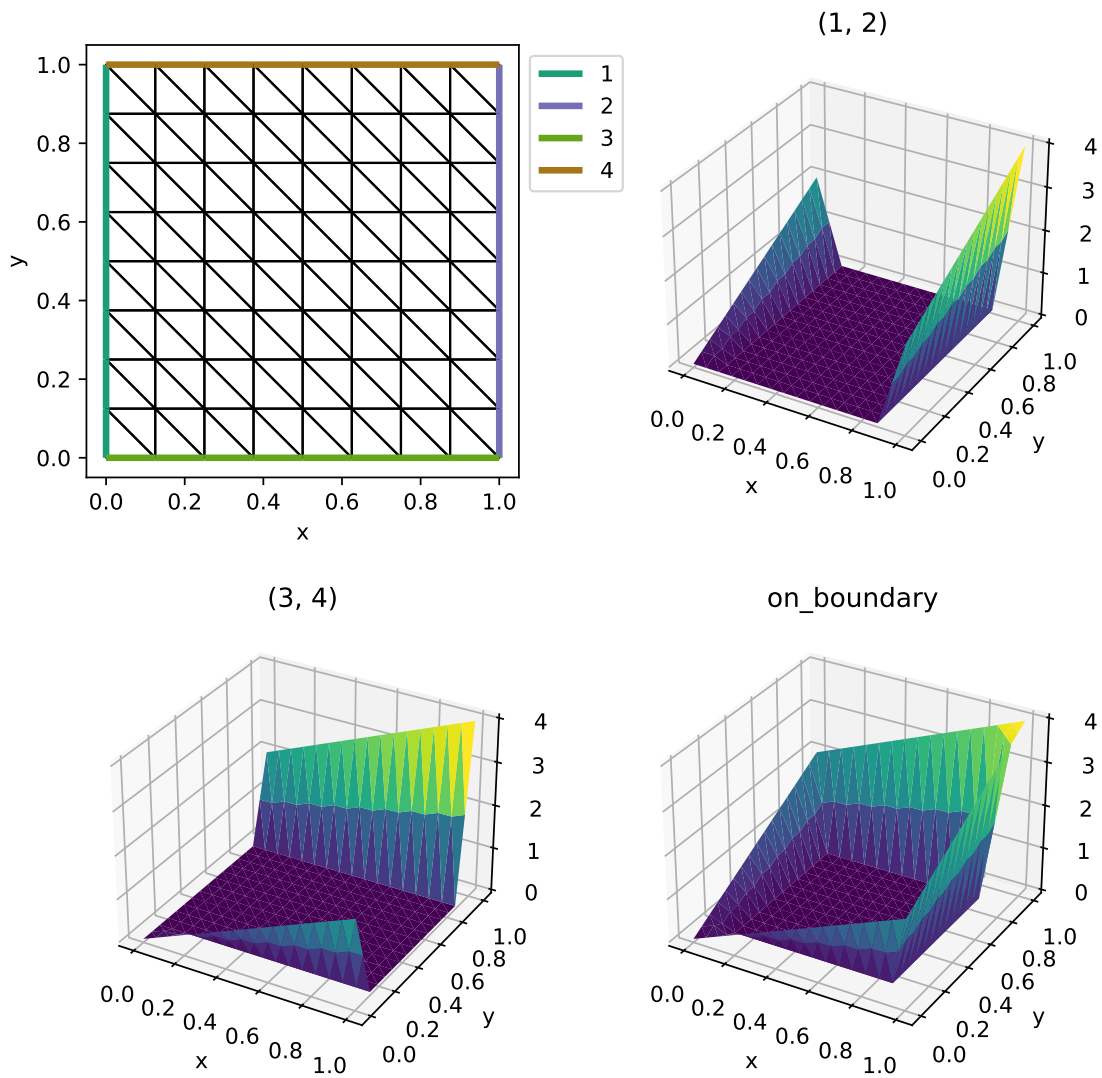
def trisurf_bdy_condition(V, g, sub_domain, axes=None):
    bc = DirichletBC(V, g=g, sub_domain=sub_domain)
    g = Function(V)
    bc.apply(g)

    trisurf(g, axes=axes)
    if axes:
        axes.set_xlabel('x')
        axes.set_ylabel('y')
        axes.set_title(sub_domain)

[10]: # plot the mesh and boundry conditons
fig, ax = plt.subplots(2, 2, figsize=[7, 7], subplot_kw=dict(projection='3d'))
ax = ax.flat

ax[0].remove()
ax[0] = fig.add_subplot(2, 2, 1)
plot_mesh_with_label(test_mesh, ax[0])

sub_domains = [(1, 2), (3, 4), 'on_boundary']
for i in range(3):
    trisurf_bdy_condition(V, g=g, sub_domain=sub_domains[i], axes=ax[i+1])
fig.tight_layout()
```



1.1.8 Gmsh 网格边界设置

需要在 gmsh 中给相应的边界加上标签 (Physical Tag)

gmsh gui 演示: 生成如下 geo 文件和 msh 文件

File: gmsh/rectangle.geo

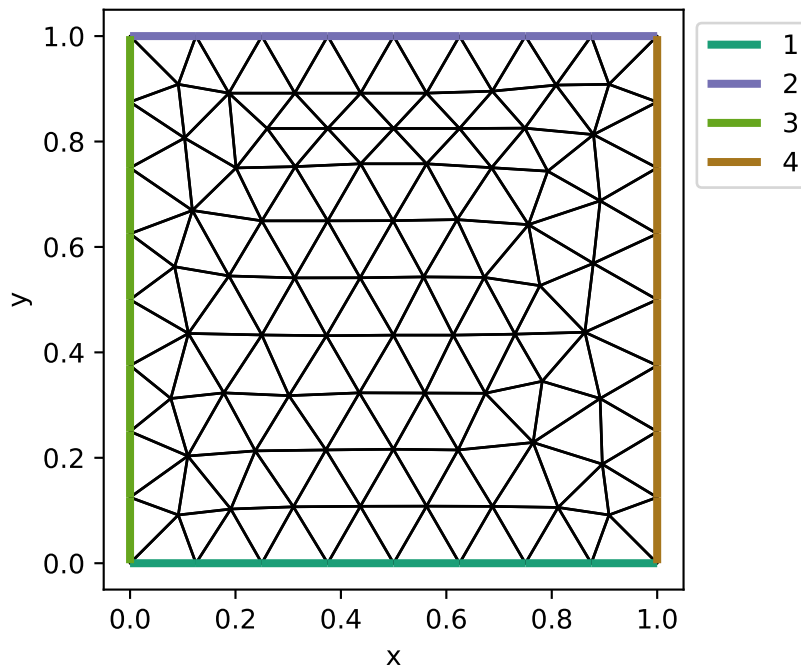
```
// Gmsh project created on Tue Sep 30 15:09:53 2022
SetFactory("OpenCASCADE");
//+
Rectangle(1) = {0, 0, 0, 1, 1, 0};
//+
Physical Curve("lower", 1) = {1};
//+
Physical Curve("upper", 2) = {3};
//+
```

```
Physical Curve("left", 3) = {4};
//+
Physical Curve("right", 4) = {2};
//+
Physical Surface("domain", 1) = {1};
```

Gmsh file: gmsh/rectangle.msh

```
[11]: # opts = PETSc.Options()
# opts.insertString('-dm_plex_gmsh_mark_vertices True')

gmsh_mesh = Mesh('gmsh/rectangle.msh')
plot_mesh_with_label(gmsh_mesh)
```



使用 *gmsh* 的 *python SDK*: [gmsh](#) 或者 [pygmsh](#)

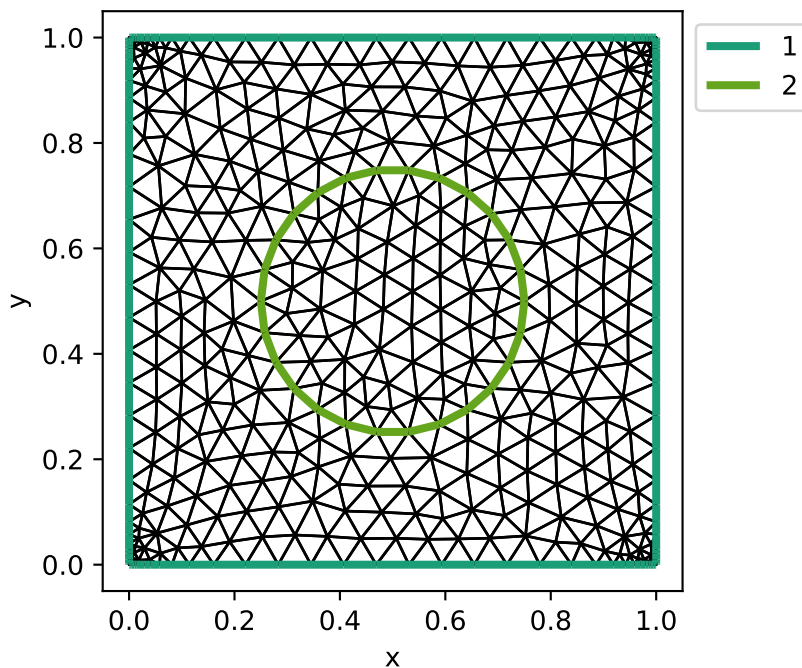
example: [make_mesh_circle_in_rect.py](#)

```
[12]: from make_mesh_circle_in_rect import make_circle_in_rect
```

```
[13]: h = 1/16
filename = 'gmsh/circle_in_rect.msh'
make_circle_in_rect(h, filename, p=3, gui=False)

cr_mesh = Mesh(filename)
plot_mesh_with_label(cr_mesh)
```

```
Info      : Writing 'gmsh/circle_in_rect.msh'...
Info      : Done writing 'gmsh/circle_in_rect.msh'
```



1.2 纯 Neumann 边界条件

求解如下 Poisson 方程

$$\begin{aligned} -\Delta u &= f & \text{in } \Omega, \\ \frac{\partial u}{\partial n} &= g_N & \text{on } \partial\Omega, \end{aligned} \quad (4)$$

变分问题

求 $u \in H^1$, 且 $\int_{\Omega} u = 0$ 使得

$$\int_{\Omega} \nabla u \cdot \nabla v = \int_{\Omega} f v + \int_{\partial\Omega} g_N v \quad \forall v \in H^1. \quad (5)$$

兼容性条件

$$\int_{\Omega} f v + \int_{\partial\Omega} g_N v = 0$$

1.2.1 Use nullspace of solve

[14]:

```

N = 8
test_mesh = RectangleMesh(nx=N, ny=N, Lx=1, Ly=1)
x, y = SpatialCoordinate(test_mesh)
f = sin(pi*x)*sin(pi*y)

subdomain_id = None # None for all boundary, 或者单个编号 如 1, 或者使用 list 或 tuple 如: (1, 2)

```

```

if True:
    # 不满足兼容性条件
    g = Constant(1)
else:
    # 满足兼容性条件
    L = assemble(1*ds(domain=test_mesh, subdomain_id=subdomain_id))
    g = Constant(-assemble(f*dx)/L)

V = FunctionSpace(test_mesh, 'CG', degree=1)
u, v = TrialFunction(V), TestFunction(V)
a = inner(grad(u), grad(v))*dx
L = inner(f, v)*dx + inner(g, v)*ds(subdomain_id=subdomain_id)

u1_h = Function(V, name='u1_h')

nullspace = VectorSpaceBasis(constant=True)

solve(a == L, u1_h,
      solver_parameters={
          # 'ksp_view': None,
          'ksp_monitor': None,
      },
      options_prefix='test1',
      nullspace=nullspace,
      transpose_nullspace=None)

u2_h = Function(V, name='u2_h')
solve(a == L, u2_h,
      solver_parameters={
          # 'ksp_view': None,
          'ksp_monitor': None,
      },
      options_prefix='test2',
      nullspace=nullspace,
      transpose_nullspace=nullspace)

fig, ax = plt.subplots(1, 2, figsize=[8, 4], subplot_kw=dict(projection='3d'))
trisurf(u1_h, axes=ax[0])
ax[0].set_title('only nullspace')
trisurf(u2_h, axes=ax[1])
ax[1].set_title('transpose nullspace')

```

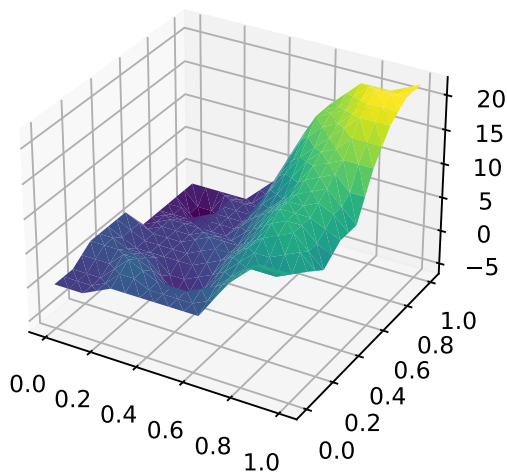
```

Residual norms for test1_ solve.
0 KSP Residual norm 7.133205795309e-01
1 KSP Residual norm 4.463009742158e+01
Residual norms for test2_ solve.
0 KSP Residual norm 5.188828525840e-01
1 KSP Residual norm 1.256141430046e-14

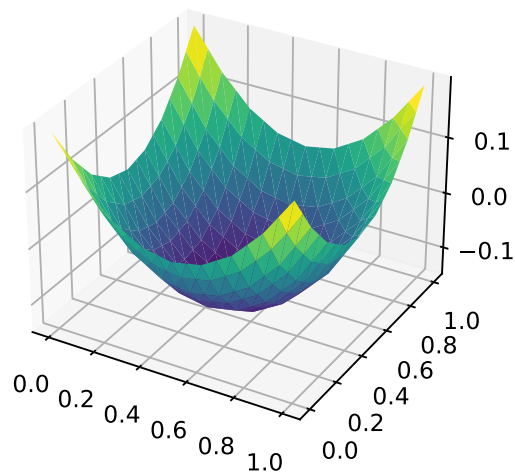
```

```
[14]: Text(0.5, 0.92, 'transpose nullspace')
```


only nullspace



transpose nullspace



1.2.2 Using Lagrange multiplier

变分问题

求 $u \in H^1, \mu \in \mathbb{R}$ 使得

$$\begin{aligned} \int_{\Omega} \nabla u \cdot \nabla v + \mu \int_{\Omega} v - \int_{\Omega} f v - \int_{\partial\Omega} g_N v &= 0, \quad \forall v \in H^1 \\ \eta \int_{\Omega} u &= 0, \quad \forall \eta \in \mathbb{R} \end{aligned} \quad (6)$$

```
[15]: # %load poisson_neumann_lagrange.py
from firedrake import *
from firedrake.petsc import PETSc

opts = PETSc.Options()
N = opts.getInt('N', default=8)
test_mesh = RectangleMesh(nx=N, ny=N, Lx=1, Ly=1)

x, y = SpatialCoordinate(test_mesh)
f = sin(pi*x)*sin(pi*y)
g_N = Constant(1)

V = FunctionSpace(test_mesh, 'CG', degree=1)
R = FunctionSpace(test_mesh, 'R', 0)

W = MixedFunctionSpace([V, R]) # or W = V*R

u, mu = TrialFunction(W)
v, eta = TestFunction(W)

a = inner(grad(u), grad(v))*dx + inner(mu, v)*dx + inner(u, eta)*dx
L = inner(f, v)*dx + inner(g_N, v)*ds
```

```
w_h = Function(W)
solve(a == L, w_h, options_prefix='test')

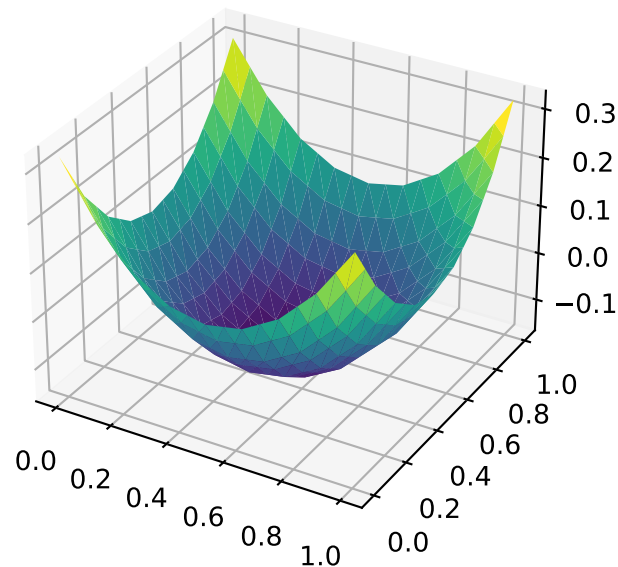
u_h, mu_h = w_h.split()

filename = 'pvd/u_h_neumann.pvd'
PETSc.Sys.Print(f'Write pvd file: {filename}')
File(filename).write(u_h)
```

Write pvd file: pvd/u_h_neumann.pvd

```
[16]: fig, ax = plt.subplots(figsize=[4, 4], subplot_kw=dict(projection='3d'))
      trisurf(u_h, axes=ax)
```

```
[16]: <mpl_toolkits.mplot3d.art3d.Poly3DCollection at 0x7fa879624700>
```



终端演示

```
$ python possion_neumann_lagrange.py -test_ksp_monitor -test_ksp_converged_reason -N 64
Number of Dofs: 4226
firedrake:WARNING Real block detected, generating Schur complement elimination PC
Residual norms for test_solve.
0 KSP Residual norm 2.501422711621e-01
1 KSP Residual norm 1.747929427611e-01
2 KSP Residual norm 1.071502741145e-14
Linear test_solve converged due to CONVERGED_RTOL iterations 2
Write pvd file: pvd/u_h_neumann.pvd

$ mpiexec -n 2 python possion_neumann_lagrange.py \
  -test_ksp_monitor -test_ksp_converged_reason -N 64
Number of Dofs: 4226
firedrake:WARNING Real block detected, generating Schur complement elimination PC
```

```

Residual norms for test_solve.
0 KSP Residual norm 2.501422711621e-01
1 KSP Residual norm 2.085403806063e-02
2 KSP Residual norm 9.317076546546e-16
Linear test_solve converged due to CONVERGED_RTOL iterations 2
Write pvd file: pvd/u_h_neumann.pvd

```

1.3 计算收敛阶

- 和真解对比
- 和参考解对比
- 相邻三层之间对比 (Cauchy 序列): [poission_convergence.py](#)

1.3.1 生成网格序列

```

base = RectangleMesh(N, N, 1, 1)
meshes = MeshHierarchy(test_mesh, refinement_levels=4)

```

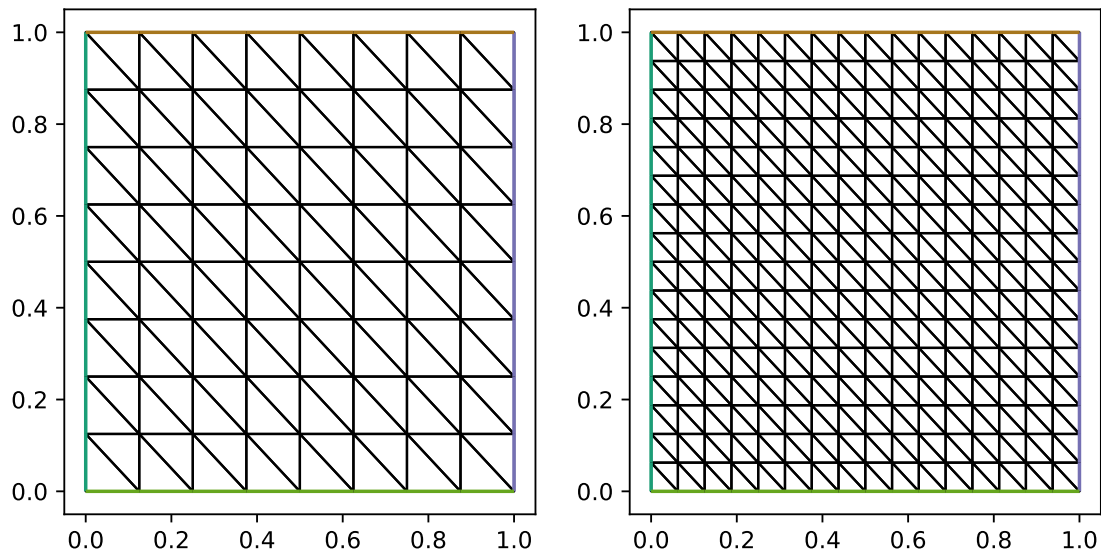
```

[17]: from firedrake import *
import matplotlib.pyplot as plt

N = 8
base = RectangleMesh(N, N, 1, 1)
meshes = MeshHierarchy(base, refinement_levels=3)

n = len(meshes)
m = min(2, n)
fig, ax = plt.subplots(1, m, figsize=[4*m, 4])
for i in range(m):
    triplot(meshes[i], axes=ax[i])

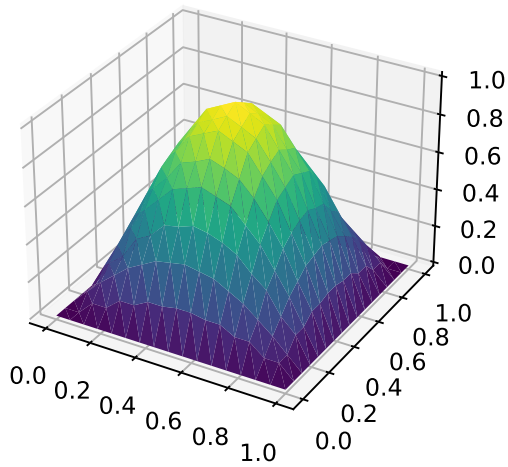
```



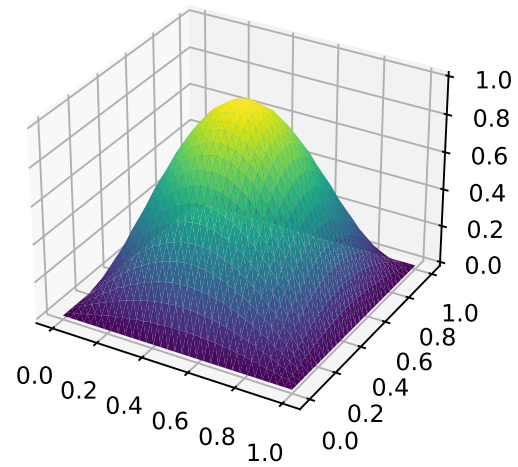
```
[18]: us = []
for mesh in meshes:
    x, y = SpatialCoordinate(mesh)
    f = sin(pi*x)*sin(pi*y)
    V = FunctionSpace(mesh, 'CG', degree=1)
    u = Function(V).interpolate(f)
    us.append(u)

m = min(4, n)
fig, ax = plt.subplots(2, 2, figsize=[4*2, 4*2], subplot_kw=dict(projection='3d'))
ax = ax.flat
for i in range(n):
    trisurf(us[i], axes=ax[i])
    ax[i].set_title(f'$h=1/{N*2**i}$')
```

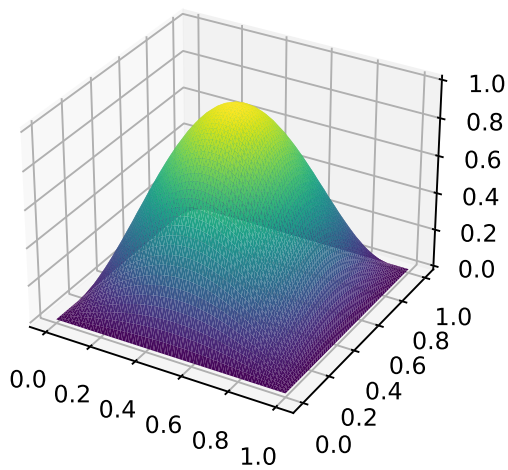
$h = 1/8$



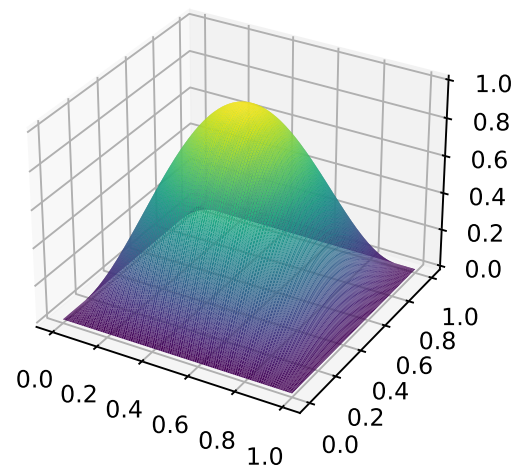
$h = 1/16$



$h = 1/32$



$h = 1/64$



1.3.2 投影到细网格上的空间中

目前 Firedrake 只能投影函数到相邻层的网格上 (由 MeshHierarchy 生成的网格), 和最密网格比较时可以多次投影, 直至最密网格, 然后比较结果.

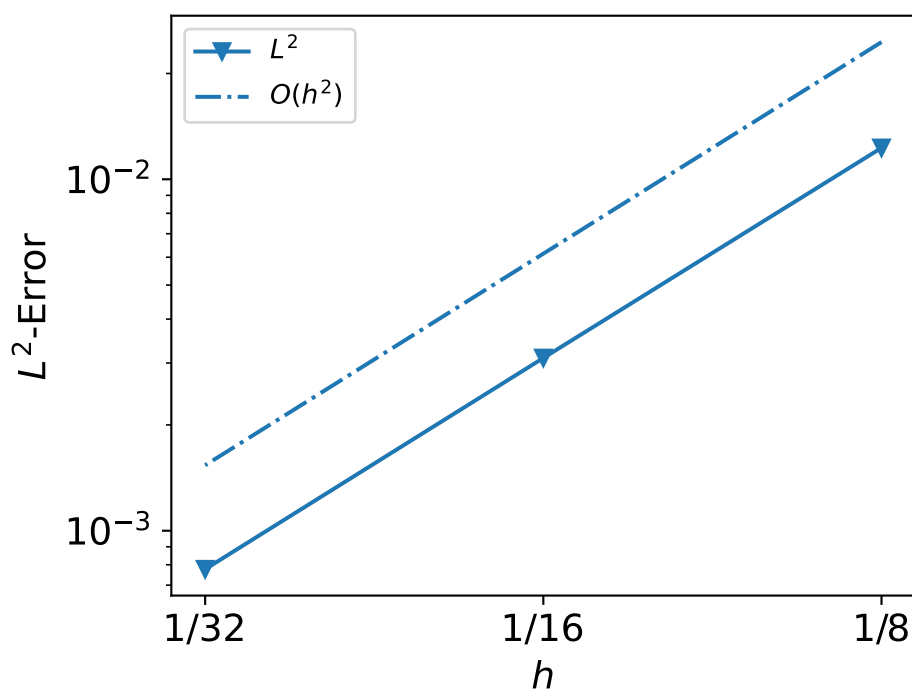
下面我们仅比较相邻层的误差

```
[19]: errors = []
      hs = []
      for i, u in enumerate(us[:-1]):
          u_ref = us[i+1]
          u_inter = project(u, u_ref.function_space())
          error = errornorm(u_ref, u_inter)
          errors.append(error)
          hs.append(1/(N*2**i))

      hs, errors

[19]: ([0.125, 0.0625, 0.03125],
      [0.012284003199971324, 0.003100763810085325, 0.0007770614161052795])

[20]: from intro_utils import plot_errors
      plot_errors(hs, errors, expect_order=2)
```



1.3.3 插值到细网格上的空间中

- VertexOnlyMesh:
- PointCloud: <https://github.com/lrtfm/fdutils>

Example of PointCloud Interpolate function f1 on mesh m1 to function f2 on mesh m2

```
[21]: import firedrake as fd
from fdutils import PointCloud
from fdutils.tools import get_nodes_coords
import matplotlib.pyplot as plt

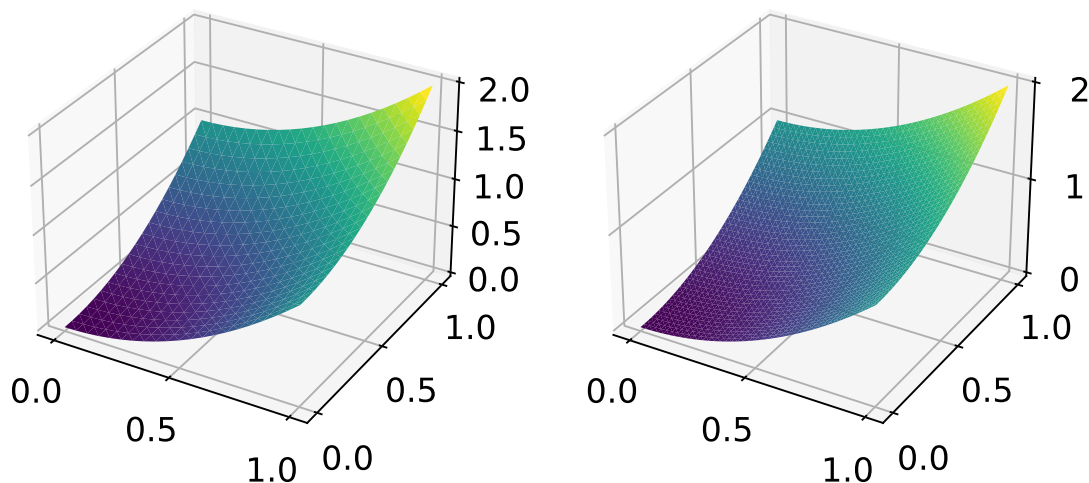
m1 = fd.RectangleMesh(10, 10, 1, 1)
V1 = fd.FunctionSpace(m1, 'CG', 2)
x, y = fd.SpatialCoordinate(m1)
f1 = fd.Function(V1).interpolate(x**2 + y**2)

m2 = fd.RectangleMesh(20, 20, 1, 1)
V2 = fd.FunctionSpace(m2, 'CG', 3)
f2 = fd.Function(V2)

points = get_nodes_coords(f2)
pc = PointCloud(m1, points, tolerance=1e-12)
f2.dat.data_with_halos[:] = pc.evaluate(f1)

fig, ax = plt.subplots(1, 2, figsize=[8, 4], subplot_kw=dict(projection='3d'))
fd.trisurf(f1, axes=ax[0])
fd.trisurf(f2, axes=ax[1])
```

```
[21]: <mpl_toolkits.mplot3d.art3d.Poly3DCollection at 0x7fa86b874fa0>
```



计算误差

```
[22]: from fdutils.tools import errornorm as my_errornorm

my_errors_0 = []
for i, u in enumerate(us[:-1]):
    # 和相邻层结果比较
    my_errors_0.append(my_errornorm(u, us[i+1], tolerance=1e-12))
```

```
my_errors_0
```

```
[22]: [0.012284003212205772, 0.003100763847789638, 0.0007770614201377909]
```

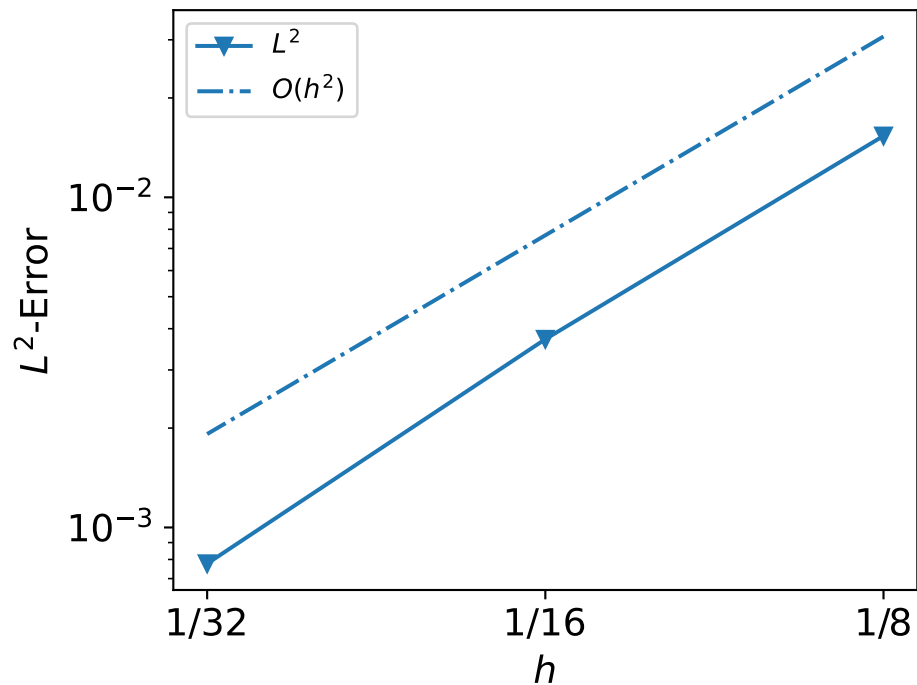
```
[23]: from fdutils.tools import errornorm as my_errornorm

my_errors = []
for i, u in enumerate(us[:-1]):
    # 和最密层结果比较
    my_errors.append(my_errornorm(u, us[-1], tolerance=1e-12))

my_errors
```

```
[23]: [0.015349062780286471, 0.0037181920308195534, 0.0007770614201377909]
```

```
[24]: from intro_utils import plot_errors
plot_errors(hs, my_errors, expect_order=2)
```



1.4 构造等参元

Firedrake 中坐标是通过函数 `Function` 给出的, 可以通过更改该函数的值来移动网格或者构造等参元对应的映射.

1.4.1 移动网格

坐标的存储 (numpy 数组)

```

mesh = RectangleMesh(10, 10, 1, 1)
mesh.coordinates.dat.data
mesh.coordinates.dat.data_ro
mesh.coordinates.dat.data_with_halos
mesh.coordinates.dat.data_ro_with_halos

```

```

[25]: import numpy as np

# test_mesh = UnitDiskMesh(refinement_level=3)
test_mesh = RectangleMesh(10, 10, 1, 1)

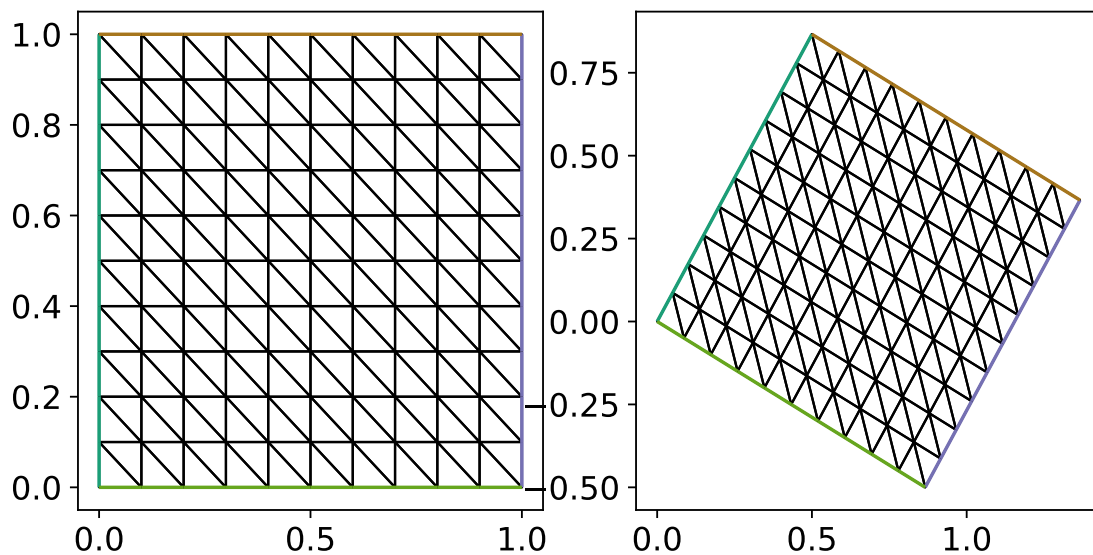
fig, ax = plt.subplots(1, 2, figsize=[8, 4])
handle = triplot(test_mesh, axes=ax[0])

theta = np.pi/6
R = np.array([[np.cos(theta), - np.sin(theta)],
               [np.sin(theta),  np.cos(theta)]])

# test_mesh.coordinates.dat.datas[:] = test_mesh.coordinates.dat.data_ro[:]@R
test_mesh.coordinates.dat.data_with_halos[:] = test_mesh.coordinates.dat.data_ro_with_halos[:]@R

handle = triplot(test_mesh, axes=ax[1])

```



1.4.2 简单映射边界点

等参元映射通过更改坐标向量场实现: 从线性网格开始构造, 把边界上的自由度移动到边界上.

```

def make_high_order_mesh_map_bdy(m, p):
    coords = m.coordinates
    V_p = VectorFunctionSpace(m, 'CG', p)
    coords_p = Function(V_p, name=f'coords_p{i}').interpolate(coords)

    bc = DirichletBC(V_p, 0, 'on_boundary')
    points = coords_p.dat.data_ro_with_halos[bc.nodes]

```



```

coords_p.dat.data_with_halos[bc.nodes] = points2bdy(points)

return Mesh(coords_p)
def points2bdy(points):
    r = np.linalg.norm(points, axis=1).reshape([-1, 1])
    return points/r

```

1.4.3 同时移动边界单元的内点

Reference: 1. M. Lenior, *Optimal Isoparametric Finite Elements and Error Estimates For Domains Involving Curved Boundaries*. SIAM. J. Numer. Anal. 23(3). 1986. pp 562–580.

等参元映射通过更改坐标向量场实现: 从线性网格开始构造, 把边界上的自由度移动到边界上, 同时移动边界单元的内部自由度.

```

def make_high_order_mesh_simple(m, p):
    if p == 1:
        return m

    coords_1 = m.coordinates
    coords_i = coords_1
    for i in range(2, p+1):
        coords_im1 = coords_i
        V_i = VectorFunctionSpace(m, 'CG', i)
        bc = DirichletBC(V_i, 0, 'on_boundary')
        coords_i = Function(V_i, name=f'coords_p{i}').interpolate(coords_im1)
        coords_i.dat.data_with_halos[bc.nodes] = \
            points2bdy(coords_i.dat.data_ro_with_halos[bc.nodes])

    return Mesh(coords_i)

```

注: 这是一个简单的实现, 并不完全符合文献 [1] 中等参元映射构造方式, 一个完整的实现方式见文件 `make_mesh_circle_in_rect.py` 中的函数 `make_high_order_coords_for_circle_in_rect`: 该函数实现了内部具有一个圆形界面的矩形区域上的等参映射.

1.4.4 数值实验

精确解为 $u = 1 - (x^2 + y^2)^{3.5}$

[26]: `%run possion_convergence_circle.py`

```

p = 1; Use iso: False; Only move bdy: False.
orders: [2.01284527 2.01420928]

p = 2; Use iso: False; Only move bdy: False.
orders: [2.07953299 2.0391775 ]

p = 2; Use iso: True; Only move bdy: False.
orders: [3.07968268 3.04739627]

p = 3; Use iso: False; Only move bdy: False.
orders: [2.06225857 2.03084755]

p = 3; Use iso: True; Only move bdy: True.
orders: [3.63334435 3.56916446]

```

```

p = 3; Use iso: True; Only move bdy: False.
orders: [4.15838886 4.09188043]

p = 4; Use iso: False; Only move bdy: False.
orders: [2.05924173 2.02916455]

p = 4; Use iso: True; Only move bdy: True.
orders: [3.50007466 3.49278383]

p = 4; Use iso: True; Only move bdy: False.
orders: [5.19566749 5.10742164]

```

1.5 间断有限元方法

1.5.1 UFL 符号

- $+$:
 $u(' -')$
- $-$:
 $u(' +')$
- avg :
 $(u(' +') + u(' -'))/2$
- jump :
 $\text{jump}(u, n) = u(' +')*n(' +') + u(' -')*n(' -')$
 $\text{jump}(u) = u(' +') - u(' -')$
- FacetNormal :
 边界法向
- CellDiameter :
 网格尺寸

1.5.2 UFL 测度

1. ds 外部边
2. dS 内部边

1.5.3 变分形式

$$\begin{aligned}
& \int_{\Omega} \nabla u \cdot \nabla v - \int_{EI} (\{\nabla u\}[vn] + [un]\{\nabla v\}) - \frac{\alpha}{h} \int_{EI} [un][vn] \\
& - \int_{EO} (vn \nabla u + un \nabla v) - \frac{\alpha}{h} \int_{EO} uv \\
& - \int_{\Omega} f v - \int_{\partial\Omega_N} g_N v = 0
\end{aligned} \tag{7}$$

其中 $[vn] = v^+ n^+ + v^- n^-$, $\{u\} = (u^+ + u^-)/2$

```
[27]: mesh = RectangleMesh(8, 8, 1, 1)

DG1 = FunctionSpace(mesh, 'DG', 1)
u, v = TrialFunction(DG1), TestFunction(DG1)

x, y = SpatialCoordinate(mesh)
f = sin(pi*x)*sin(pi*y)

h = Constant(2.0)*Circumradius(mesh)
alpha = Constant(1)
gamma = Constant(1)

n = FacetNormal(mesh)

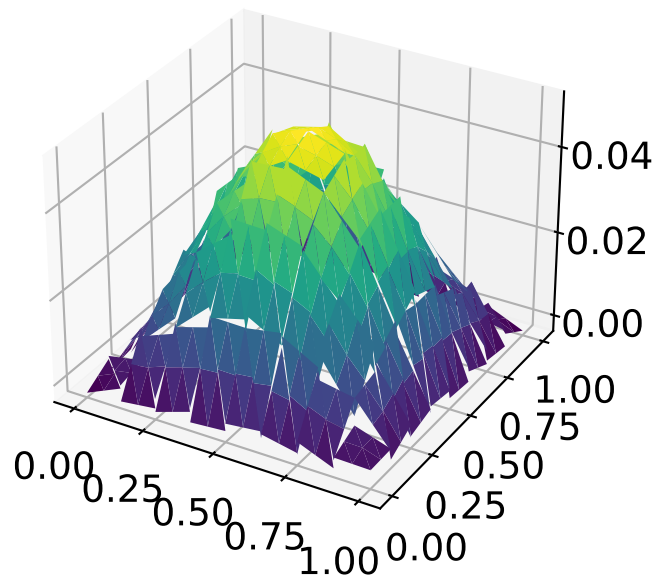
a = inner(grad(u), grad(v))*dx \
    - dot(avg(grad(u)), jump(v, n))*dS \
    - dot(jump(u, n), avg(grad(v)))*dS \
    + alpha/avg(h)*dot(jump(u, n), jump(v, n))*dS \
    - dot(grad(u), v*n)*ds \
    - dot(u*n, grad(v))*ds \
    + gamma/h*u*v*ds

L = f*v*dx

u_h = Function(DG1, name='u_h')
bc = DirichletBC(DG1, 0, 'on_boundary')
solve(a == L, u_h, bcs=bc)
```

```
[28]: fig, ax = plt.subplots(figsize=[8, 4], subplot_kw=dict(projection='3d'))
trisurf(u_h, axes=ax)
```

```
[28]: <mpl_toolkits.mplot3d.art3d.Poly3DCollection at 0x7fa86b3f34f0>
```



1.6 Dirac Delta 函数

通过数值积分公式实现 dirac delta 函数

```
[14]: from firedrake import *

from firedrake.petsc import PETSc
from pyop2 import op2
from pyop2.datatypes import ScalarType
from mpi4py import MPI
import finat
import numpy as np

import matplotlib.pyplot as plt

set_level(CRITICAL) # Disbale warnings

# TODO: change name to dirac_operator?
def delta_expr(m, x0):
    """Make dirac function at point

    Args:
        m: mesh
        x0: source point
    Return:
        delta function

    Example:
        delta = delta_expr(m, x0)
        f = Function(V)
```

```

    f_x0 = assemble(delta(f))
    """
    V = FunctionSpace(m, 'DG', 0)
    cell_marker = Function(V, name='cell_marker', dtype=ScalarType)
    qrule = finat.quadrature.make_quadrature(V.finat_element.cell, 0)
    cell, X = m.locate_cell_and_reference_coordinate(x0, tolerance=1e-6)

    # c = 0 if X is None else 1
    n_cell_local = len(cell_marker.dat.data)
    if X is not None and cell < n_cell_local:
        c = 1
    else:
        c = 0

    comm = m.comm
    s = comm.size - comm.rank
    n = comm.allreduce(int(s*c), op=MPI.MAX)

    if n == 0:
        raise BaseException("Points not found!")

    k = int(comm.size - n) # get the lower rank which include the point x0

    if c == 1 and comm.rank == k:
        X[X<0] = 0
        X[X>1] = 1
        cell_marker.dat.data[cell] = 1
        comm.bcast(X, root=k)
    else:
        cell_marker.dat.data[:] = 0 # we must set this otherwise the process will hangup
        X = comm.bcast(None, root=k)

    cell_marker.dat.global_to_local_begin(op2.READ)
    cell_marker.dat.global_to_local_end(op2.READ)

    qrule.point_set.points[0] = X
    qrule.weights[0] = qrule.weights[0]/np.real(assemble(cell_marker*dx))

    return lambda f: f*cell_marker*dx(rule=qrule)

```

测试 delta_expr

```

[15]: def test_dirca_delta():
    test_mesh = RectangleMesh(10, 10, 1, 1)
    V = FunctionSpace(test_mesh, 'CG', 3)
    x1 = 0.683
    x2 = 0.333
    source = Constant([x1,x2])
    x0 = source
    delta = delta_expr(test_mesh, source)

    x, y = SpatialCoordinate(test_mesh)
    g = Function(V).interpolate(x**3 + y**3)

    expected_value = g.at([x1, x2])
    value = assemble(delta(g))
    PETSc.Sys.Print(f"value = {value}, expected value = {expected_value}")

```

```
test_dirca_delta()
```

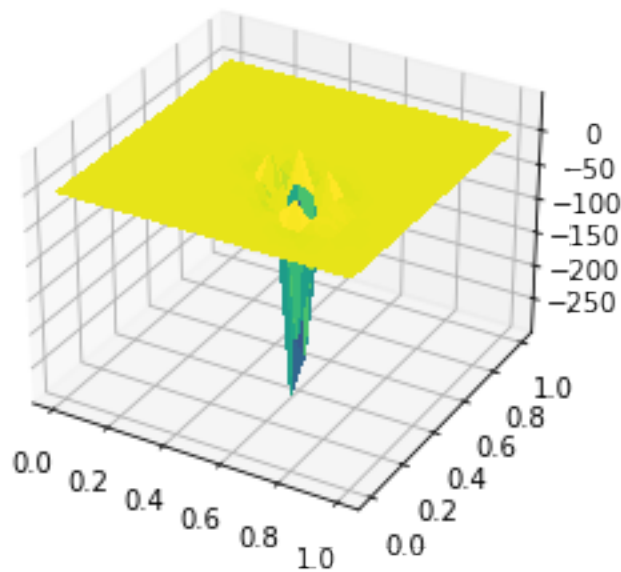
value = 0.35553802400000073, expected value = 0.3555380240000015

Dirac delta 函数的 L2 投影

```
[16]: test_mesh = RectangleMesh(10, 10, 1, 1)
V = FunctionSpace(test_mesh, 'CG', 3)
delta = delta_expr(test_mesh, [0.638, 0.33])
bc = DirichletBC(V, 0, 'on_boundary')
u, v = TrialFunction(V), TestFunction(V)
sol = Function(V)
solve(u*conj(v)*dx == delta(conj(v)), sol, bcs=bc)

fig, ax = plt.subplots(figsize=[8, 4], subplot_kw=dict(projection='3d'))
trisurf(sol, axes=ax)
```

```
[16]: <mpl_toolkits.mplot3d.art3d.Poly3DCollection at 0x7fe65975a3a0>
```



```
[17]: sol.dat.data.max(), sol.dat.data.min()
```

```
[17]: (788.6598262630521, -267.53195381302527)
```

求解源项为 Dirac delta 函数的 Poisson 方程

```
[23]: x0 = [0, 0]
# N = 500
# m = SquareMesh(N, N, 1)
```

```

m = UnitDiskMesh(refinement_level=3)

V = FunctionSpace(m, 'CG', 1)

v = TestFunction(V)
u = TrialFunction(V)

a = inner(grad(u), grad(v))*dx
L = delta_expr(m, x0)(v)
u = Function(V, name='u')

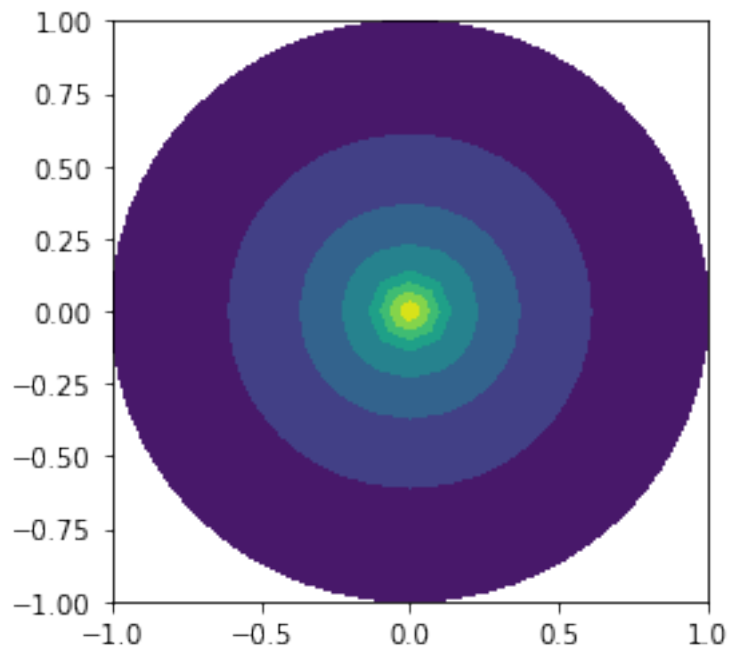
bc = DirichletBC(V, 0, 'on_boundary')

solve(a == L, u, bcs=bc)

# solve(a == L, u)
fig, ax = plt.subplots(figsize=[4, 4])
tricontourf(u, axes=ax)

```

[23]: <matplotlib.tri.tricontour.TriContourSet at 0x7fe658d96b20>



1.7 自由度映射关系

1.7.1 编号

- `V.dim()`: 自由度个数
- `V.cell_node_list`: 局部编号与全局编号

```
[29]: mesh = RectangleMesh(8, 8, 1, 1)
      V = FunctionSpace(mesh, 'CG', 1)
      V.dim(), V.cell_node_list[:5]
```

```
[29]: (81,
      array([[0, 1, 2],
            [1, 2, 3],
            [2, 3, 4],
            [1, 3, 5],
            [3, 4, 6]], dtype=int32))
```

Example: 第一个三角形的坐标

```
[30]: coords = mesh.coordinates
```

```
[31]: # get the cell node map
      V_c = coords.function_space()
      V_c.cell_node_list[:2]
```

```
[31]: array([[0, 1, 2],
            [1, 2, 3]], dtype=int32)
```

```
[32]: # another way to get the cell node map
      coords.cell_node_map().values[:2]
```

```
[32]: array([[0, 1, 2],
            [1, 2, 3]], dtype=int32)
```

```
[33]: coords.dat.data_ro_with_halos[[0, 1, 2]]
```

```
[33]: array([[0.   , 0.   ],
            [0.   , 0.125],
            [0.125, 0.   ]])
```

1.7.2 有限元自由度

```
[34]: V = FunctionSpace(mesh, 'CG', 2)
      # V.dim(), V.cell_node_list[:5]

      element = V.finat_element
      element.degree, element.cell,
```

```
[34]: (2, <FIAT.reference_element.UFCTriangle at 0x7fa86be29a60>)
```

```
[35]: V.finat_element.entity_dofs()
```

```
[35]: {0: {0: [0], 1: [1], 2: [2]}, 1: {0: [3], 1: [4], 2: [5]}, 2: {0: []}}
```

```
[36]: V.finat_element.entity_support_dofs()
```



```
[36]: {0: {0: [0], 1: [1], 2: [2]},
      1: {0: [1, 2, 3], 1: [0, 2, 4], 2: [0, 1, 5]},
      2: {0: [0, 1, 2, 3, 4, 5]}}
```

1.7.3 查看矩阵和向量 (PETSc)

[Introduction to PETSc](#)

DOC: https://web.corral.tacc.utexas.edu/CompEdu/pdf/pcse/petsc_p_course.pdf

PETSc git repo: [petsc4py demo](#)

保存矩阵到文件: [matvecio.py](#)

```
[37]: test_mesh = RectangleMesh(nx=4, ny=4, Lx=1, Ly=1)
      x, y = SpatialCoordinate(test_mesh)
      f = sin(pi*x)*sin(pi*y)

      V = FunctionSpace(test_mesh, 'CG', degree=1)

      u, v = TrialFunction(V), TestFunction(V)

      a = inner(grad(u), grad(v))*dx
      L = inner(f, v)*dx
```

```
[38]: A = assemble(a)
      b = assemble(L)
      type(A), type(b)
```

```
[38]: (firedrake.matrix.Matrix, firedrake.function.Function)
```

```
[39]: type(A.petscmat)
```

```
[39]: petsc4py.PETSc.Mat
```

```
[40]: with b.dat.vec_ro as vec:
      print(type(vec))
```

```
<class 'petsc4py.PETSc.Vec'>
```

2 NS 方程

Navier-Stokes 方程:

$$\begin{cases} \partial_t u - \mu \Delta u + (u \cdot \nabla)u + \nabla p = f, & \text{in } \Omega \times (0, T] \\ \nabla \cdot u = 0, & \text{in } \Omega \times (0, T] \end{cases} \quad (8)$$

初边值条件

$$\begin{cases} u = 0, & \text{on } \partial\Omega \times (0, T] \\ u_0 = (y, -x) & \text{in } \Omega \text{ at } t = 0 \end{cases} \quad (9)$$

```
[41]: from firedrake import *

mu = 1
T = 0.25

N_S = 16
N_T = 128

tau = T/N_T
h = 1/N_S

mesh = RectangleMesh(N_S, N_S, 1, 1)

x = SpatialCoordinate(mesh)
# u_0 = as_vector((x[1] - 0.5, - x[0] + 0.5))
u_0 = as_vector((x[1], - x[0]))
f = as_vector([0, -1])
```

2.1 函数空间

采用 MINI 元, 即 $P1 \times P1b$.

$P1b$ 由 $P1$ 加上 Bubble 组成.

`NodalEnrichedElement`, `EnrichedElement`

`VectorFunctionSpace` 构造向量空间

```
[42]: cell = mesh.ufl_cell()
tdim = cell.topological_dimension()

# Mini element: P1 X P1b
P1 = FiniteElement("CG", cell, 1)
B = FiniteElement("B", cell, tdim+1)
P1b = P1 + B # or P1b = NodalEnrichedElement(P1, B)

V_u = VectorFunctionSpace(mesh, P1b)
V_p = FunctionSpace(mesh, "CG", 1)
V = MixedFunctionSpace([V_u, V_p])
```

2.2 弱形式

$$\begin{cases} \frac{1}{\tau}(u^n - u^{n-1}, v) + \mu(\nabla u^n, \nabla v) + ((u^n \cdot \nabla)u^n, v) - (p^n, \nabla \cdot v) = (f^n, v) \\ (q, \nabla \cdot u^n) = 0 \end{cases} \quad (10)$$

- `TrialFunctions`, `TestFunctions`:

以 `tuple` 返回函数空间中的试验/测试函数,

主要用于 `MixedFunctionSpace`.

- `split`, `Function.split`

- `split`: 以索引的方式获取 `MixedFunctionSpace` 中函数的分量 (保留 UFL 关联信息, 用于定义变分形式)
- `Function.split`: 以存储共享的方式获取分量 (生成新的变量, 只是共享原存储空间)

由于该问题是非线性问题, 我们打算用 `NonlinearVariationalSolver` 进行求解, 所以下面定义 `w` 使用了 `Function` 而不是 `TrialFunction/TrialFunctions`.

```
[43]: w = Function(V) # u and p
      u, p = split(w)

      v, q = TestFunctions(V)

      w_nm1 = Function(V)
      u_nm1, p_nm1 = w_nm1.split()
      u_nm1.rename('u_h') # for visualization in paraview
      p_nm1.rename('p_h')

      Re = Constant(mu)

      F = \
          Constant(1/tau)*inner(u - u_nm1, v)*dx \
          + Re*inner(grad(u+u_nm1)/2, grad(v))*dx \
          + inner(dot(grad(u), (u+u_nm1)/2), v)*dx \
          - p*div(v)*dx \
          + div(u)*q*dx \
          - inner(f, v)*dx
```

2.3 定义 Solver

类似于纯 Neumann 问题, 我们将使用 `nullspace` 参数.

注意下面混合空间中, 边界条件和 `nullspace` 的定义.

```
[44]: bc = DirichletBC(V.sub(0), 0, 'on_boundary')
      nullspace = MixedVectorSpaceBasis(V, [V.sub(0), VectorSpaceBasis(constant=True)])

      problem = NonlinearVariationalProblem(F, w, bcs=bc) # F = 0
      solver = NonlinearVariationalSolver(problem,
                                          options_prefix='ns',
                                          solver_parameters=None, # {'snes_converged_reason': None,
                                          ↪ 'snes_max_it': 100},
                                          nullspace=nullspace
                                          )
```

2.4 时间循环

```
[45]: from tqdm.notebook import tqdm # progress bar

      u_, p_ = w.split()

      output = File('pvd/ns-equation.pvd')

      u_nm1.project(u_0)
      output.write(u_nm1, p_nm1, time=0)
```

```

for i in tqdm(range(N_T)):
    t = tau*(i+1)

    solver.solve()

    u_nm1.assign(u_)
    p_nm1.assign(p_)

    output.write(u_nm1, p_nm1, time=t)

```

0%| | 0/128 [00:00<?, ?it/s]

2.4.1 Constant 用于时间依赖的表达式

[46]:

```

from firedrake import *
mesh = RectangleMesh(10, 10, 1, 1)
C1 = Constant(0)

x, y = SpatialCoordinate(mesh)
expr = C1*(x+y)

v = []
for i in range(5):
    t = i*0.1
    C1.assign(t)
    v.append(
        assemble(expr*dx)
    )

print(v)

```

[0.0, 0.09999999999999991, 0.19999999999999982, 0.29999999999999966,
0.39999999999999963]

2.5 ParaView 可视化计算结果

Pipeline 和 Filter

2.5.1 二维结果 (surf 图)

Filter: Wrap by scalar

2.5.2 选择部分区域显示

View -> Find Data

3 多进程并行 (MPI 和 PETSc)

使用 `mpiexec` 运行 python 文件即可.

我们使用 `ipyparallel` 介绍并行程序的一些内容, 需要先安装 `ipyparallel`

```
[25]: import ipyparallel as ipp
import os

cluster = ipp.Cluster(profile="mpi", n=2)
client = cluster.start_and_connect_sync()
```

```
Starting 2 engines with <class
'ipyparallel.cluster.launcher.MPIEngineSetLauncher'>

0%|          | 0/2 [00:00<?, ?engine/s]
```

3.1 DMPlex

并行时, 网格会被划分成不同的块, 分配到各个进程.

网格由 PETSc 中的 DMPlex 管理.

DMPlex Reference: 1. Lange, M., Mitchell, L., Knepley, M. G., & Gorman, G. J. Efficient mesh management in firedrake using PETSC DMPLEX. SISC, 2016, 38(5), S143-S155. 2. Hapla, V., Knepley, M. G., Afanasiev, M., Boehm, C., van Driel, M., Krischer, L., & Fichtner, A. Fully parallel mesh I/O using PETSc DMPlex with an application to waveform modeling. SISC, 2021, 43(2), C127-C153.

```
[2]: %%px --block
from firedrake import *

mesh = RectangleMesh(8, 8, 1, 1)
mesh.topology_dm.view()

%px: 0%|          | 0/2 [00:00<?, ?tasks/s]

[stdout:0] DM Object: firedrake_default_topology 2 MPI processes
type: plex
firedrake_default_topology in 2 dimensions:
  Number of 0-cells per rank: 45 45
  Number of 1-cells per rank: 108 108
  Number of 2-cells per rank: 64 64
Labels:
  depth: 3 strata with value/size (0 (45), 1 (108), 2 (64))
  celltype: 3 strata with value/size (0 (45), 1 (108), 3 (64))
  Face Sets: 2 strata with value/size (1 (8), 3 (8))
  exterior_facets: 1 strata with value/size (1 (16))
  interior_facets: 1 strata with value/size (1 (92))
```

3.2 Star Forest

Reference:

[1] J. Zhang et al., [The PetscSF Scalable Communication Layer](#), IEEE Transactions on Parallel and Distributed Systems, 33(4), 2022.

```
[3]: %%px --block
from firedrake import *
from firedrake.petsc import PETSc
```

```

from petsc4py import PETSc
import numpy as np

# 6-----7-----8
# |         |         |
# 3-----4-----5
# |         |         |
# 0-----1-----2

def test_SFDDistributeSection():
    comm = COMM_WORLD
    if comm.rank == 0:
        cells = np.asarray(
            [[0, 1, 3],
             [1, 2, 4],
             [1, 4, 3],
             [2, 5, 4],
             [3, 4, 6],
             [4, 5, 7],
             [4, 7, 6],
             [5, 8, 7]], dtype=np.int32)
        coords = np.asarray(
            [[0. , 0. ],
             [0.5, 0. ],
             [1. , 0. ],
             [0. , 0.5],
             [0.5, 0.5],
             [1.0, 0.5],
             [0. , 1. ],
             [0.5, 1. ],
             [1. , 1. ]], dtype=np.double)
    else:
        cells = np.zeros([0, 3], dtype=np.int32)
        coords = np.zeros([0, 2], dtype=np.double)
    dim = 2
    plex = PETSc.DMPlex().createFromCellList(dim, cells, coords, comm=comm)
    rootSection = PETSc.Section().create(comm=comm)
    pStart, pEnd = plex.getHeightStratum(2)
    rootSection.setChart(*plex.getChart())
    for p in range(pStart, pEnd):
        rootSection.setDof(p, 1)
    rootSection.setUp()
    rootSection.viewFromOptions('-section_view')

    dplex = plex.clone()
    msf = dplex.distribute()

    if msf is None:
        PETSc.Sys.Print("Warning: plex has not been distributed!")
        return
    dplex.viewFromOptions('-dm_view')

    def isEqualSF(ssf0, ssf1):
        nroots0, local0, remote0 = ssf0.getGraph()
        nroots1, local1, remote1 = ssf1.getGraph()
        return (nroots0 == nroots1) \
            and np.array_equal(local0, local1) \
            and np.array_equal(remote0, remote1)

```

```

remoteOffsets0, leafSection0 = msf.distributeSection(rootSection)
ssf0 = msf.createSectionSF(rootSection, remoteOffsets0, leafSection0)

remoteOffsets1, leafSection1 = msf.distributeSection(rootSection, None)
ssf1 = msf.createSectionSF(rootSection, remoteOffsets1, leafSection1)

leafSection2 = PETSc.Section()
remoteOffsets2, leafSection2 = msf.distributeSection(rootSection, leafSection2)
ssf2 = msf.createSectionSF(rootSection, remoteOffsets2, leafSection2)

leafSection3 = PETSc.Section()
remoteOffsets3, _ = msf.distributeSection(rootSection, leafSection3)
ssf3 = msf.createSectionSF(rootSection, remoteOffsets3, leafSection3)

leafSection4 = PETSc.Section().create(dplex.getComm())
remoteOffsets4, leafSection4 = msf.distributeSection(rootSection, leafSection4)
ssf4 = msf.createSectionSF(rootSection, remoteOffsets4, leafSection4)

leafSection5 = PETSc.Section().create(dplex.getComm())
remoteOffsets5, _ = msf.distributeSection(rootSection, leafSection5)
ssf5 = msf.createSectionSF(rootSection, remoteOffsets5, leafSection5)

assert isEqualSF(ssf0, ssf1)
assert isEqualSF(ssf0, ssf2)
assert isEqualSF(ssf0, ssf3)
assert isEqualSF(ssf0, ssf4)
ssf0.view()

```

```

[4]: %%px --block
# Add back after upgrad the firedrake
test_SF DistributeSection()

```

```

[stdout:0] PetscSF Object: 2 MPI processes
type: basic
[0] Number of roots=9, leaves=6, remote ranks=1
[0] 0 <- (0,0)
[0] 1 <- (0,1)
[0] 2 <- (0,3)
[0] 3 <- (0,4)
[0] 4 <- (0,6)
[0] 5 <- (0,7)
[1] Number of roots=0, leaves=6, remote ranks=1
[1] 0 <- (0,1)
[1] 1 <- (0,2)
[1] 2 <- (0,4)
[1] 3 <- (0,5)
[1] 4 <- (0,7)
[1] 5 <- (0,8)
MultiSF sort=rank-order

```

3.3 输出

[intro_utils.py](#)

```
[49]: %%px --block
from firedrake import *
from firedrake.petsc import PETSc
from mpi4py import MPI

PETSc.Sys.Print('This is first line (from rank 0)')
```

```
[stdout:0] This is first line (from rank 0)
```

```
[50]: %%px --block
PETSc.Sys.syncPrint('This is second line (from all rank)')
PETSc.Sys.syncFlush()
```

```
[stdout:0] This is second line (from all rank)
This is second line (from all rank)
```

```
[51]: %%px --block
print('This msg from all rank')
```

```
[stdout:0] This msg from all rank
```

```
[stdout:1] This msg from all rank
```

3.4 communicator

```
[52]: %%px --block

mesh = RectangleMesh(8, 8, 1, 1)
PETSc.Sys.syncPrint(mesh.comm.rank, mesh.comm.size)
PETSc.Sys.syncFlush()
```

```
[stdout:0] 0 2
1 2
```

```
[53]: %%px --block

PETSc.Sys.syncPrint(COMM_WORLD.rank, COMM_WORLD.size)
PETSc.Sys.syncFlush()
```

```
[stdout:0] 0 2
1 2
```

```
[ ]: %%px --block

PETSc.Sys.syncPrint(COMM_SELF.rank, COMM_SELF.size)
PETSc.Sys.syncFlush()
```

```
[stdout:0] 0 1
0 1
```


有些时候需要在某个进程上, 做指定的操作或运算, 如只在第 0 个进程上画图

```
if COMM_WORLD.rank == 0:
    plot(...)
```

4 Debug

4.1 常见问题

4.1.1 DIVERGED_LINEAR_SOLVE

The errors are like this.

```
File "/home/yyz/firedrake/src/firedrake/firedrake/adjoint/solving.py", line 50, in wrapper
    output = solve(*args, **kwargs)
File "/home/yyz/firedrake/src/firedrake/firedrake/solving.py", line 129, in solve
    _solve_varproblem(*args, **kwargs)
File "/home/yyz/firedrake/src/firedrake/firedrake/solving.py", line 161, in _solve_varproblem
    solver.solve()
File "/home/yyz/firedrake/src/firedrake/firedrake/adjoint/variational_solver.py", line 75, in wrapper
    out = solve(self, **kwargs)
File "/home/yyz/firedrake/src/firedrake/firedrake/variational_solver.py", line 278, in solve
    solving_utils.check_snes_convergence(self.snes)
File "/home/yyz/firedrake/src/firedrake/firedrake/solving_utils.py", line 139, in check_snes_convergence
    raise ConvergenceError(r"Nonlinear solve failed to converge after %d nonlinear iterations.
firedrake.exceptions.ConvergenceError: Nonlinear solve failed to converge after 0 nonlinear iterations.
Reason:
    DIVERGED_LINEAR_SOLVE
```

Reasons for this: 1. Your equation is not closed. Maybe you wrote wrong boundary conditions. Check the boundary condition carefully. 2. The resulting system is singular? ## Maybe 3. ...

4.1.2 PyErr_Occurred

```
python: src/petsc4py.PETSc.c:348918: __Pyx_PyCFunction_FastCall: Assertion '!PyErr_Occurred()' failed.
```

This may be caused by your python code (with programmer error, such as undefined variables) called by PETSc

4.1.3 Tips

在程序开始添加如下代码, 可能会有更详细信息

```
PETSc.Sys.popErrorHandler()
```

4.2 调试 Python 代码

运行中抛出异常, 定位出错代码, 检查相关的变量是否有异常值存在. 例如在 Jupyter notebook 中, %debug 可打开调试器, 检查相关变量.

4.3 调试 C 代码 (gdb)

由于 firedrake 基于 PETSc 进行网格管理和线性方程组求解, 有时出错会在 PETSc 中, 例如运行如下代码:

TODO: 找个示例, 这个示例不行

```
# filename: test.py
import sys
import petsc4py
petsc4py.init(sys.argv)
from petsc4py import PETSc
if PETSc.COMM_WORLD.rank == 0:
    PETSc.Vec().create(comm=PETSc.COMM_SELF).view()
```

出错信息如下:

```
$ python test.py
Vec Object: 1 MPI process
    type not yet set
Traceback (most recent call last):
  File "test.py", line 7, in <module>
    PETSc.Vec().create(comm=PETSc.COMM_SELF).view()
  File "PETSc/Vec.pyx", line 140, in petsc4py.PETSc.Vec.view
petsc4py.PETSc.Error: error code 56
[0] VecView() at /home/yyz/software/firedrake-mini-petsc/src/petsc/src/vec/vec/interface/vector.c:715
[0] No support for this operation for this object type
[0] No method view for Vec of type (null)
```

这时可以使用 gdb 等调试工具.

4.3.1 gdb 命令行说明

```
gdb [options] --args executable-file [inferior-arguments ...]
```

4.3.2 参数 (options)

1. -ex COMMAND: 执行 gdb 命令
2. --args exe [exe-args] 传递参数给 exe
3. --pid <pid> 调试正在运行的程序

4.3.3 gdb 命令:

1. bt: 查看函数调用栈
2. run: 运行可执行文件
3. l: 查看代码
4. p: 打印变量

4.3.4 示例 (调试 test.py)

```
$ gdb -ex run --args $(which python3) test.py
```

4.4 并行程序调试

4.4.1 PETSc 的参数 `-start_in_debugger`

Reference: 1. <https://petsc.org/main/docs/manualpages/Sys/PetscInitialize/> 2. <https://petsc.org/main/docs/manualpages/Sys/PetscSetDebugTerminal/>

可以选择使用 PETSc 的参数 `-start_in_debugger` 给每个进程启动调试器如下:

```
mpiexec -n 3 $(which python) test.py -start_in_debugger
```

默认会启动多个 xterm 窗口.

Notes: 修改 xterm 窗口显示效果 (Ref: <http://www.futurile.net/2016/06/14/xterm-setup-and-truetype-font-configuration/>)

```
$ cat ~/.Xdefaults
xterm*faceName: Monospace
xterm*faceSize: 12
xterm*foreground: rgb:a8/a8/a8
xterm*background: rgb:00/00/00
```

4.4.2 工具 `tmux-mpi`

Reference: <https://github.com/firedrakeproject/firedrake/wiki/Parallel-MPI-Debugging-with-tmux-mpi>

另外我们也可以选择使用工具 `tmux-mpi`.

安装 `tmux-mpi`

1. 安装 `tmux`

```
sudo apt-get install tmux
```

2. 安装 `dtach` (`tmux-mpi` 依赖)

先编译 `dtach`, 然后拷贝二进制文件到某个在 `PATH` 中的路径, 如 `$HOME/bin`.

```
git clone https://github.com/crigler/dtach
cd dtach
./configure
make
cp dtach $HOME/bin
```

运行 `which dtach` 确认安装是否成功

3. 安装 `tmux-mpi`

使用 `pip` 安装 `bash` `pip install --upgrade --no-cache-dir git+https://github.com/wrs20/tmux-mpi@master`

调试命令

1. 启动调试器 `bash` `tmux-mpi 3 gdb -ex run --args $(which python) test.py`
2. Attach 到相应的的伪终端, 每个进程一个窗口. (这里是 `tmux` 的一个 `session`, 有多个 `window`)

```
tmux attach -t tmux-mpi
```

3. 使用 gdb 调试命令调试

5 Appendix

1. 本文档 pdf 版本生成

```
jupyter nbconvert --to pdf 02_firedrake_intro.ipynb
```