

Chapter 1

Example problem: 2D driven cavity flow in a quarter-circle domain with spatial adaptation.

In this example we shall demonstrate

- how easy it is to adapt the code for the solution of the driven cavity problem in a square domain, discussed in a [previous example](#) , to a different domain shape,
- how to apply body forces (e.g. gravity) in a Navier-Stokes problem,
- how to switch between the stress-divergence and the simplified forms of the incompressible Navier-Stokes equations.

1.1 The example problem

In this example we shall illustrate the solution of the steady 2D Navier-Stokes equations in a modified driven cavity problem: The fluid is contained in a quarter-circle domain and is subject to gravity which acts in the vertical direction. We solve the problem in two different formulations, using the stress-divergence and the simplified form of the Navier-Stokes equations, respectively, and by applying the gravitational body force via the gravity vector, \mathbf{G} , and via the body force function, \mathbf{B} , respectively.

Problem 1:

**The 2D driven cavity problem in a quarter circle domain with gravity,
using the stress-divergence form of the Navier-Stokes equations**

Solve

$$Re \, u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{Re}{Fr} G_i + \frac{\partial}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right), \quad (1)$$

and

$$\frac{\partial u_i}{\partial x_i} = 0,$$

in the quarter-circle domain $D = \{x_1 \geq 0, x_2 \geq 0 \text{ and } x_1^2 + x_2^2 \leq 1\}$, subject to the Dirichlet boundary conditions

$$\mathbf{u}|_{\partial D} = (0, 0), \quad (2)$$

on the curved and left boundaries; and

$$\mathbf{u}|_{\partial D} = (1, 0), \quad (3)$$

Generated by Doxygen

on the bottom boundary, $x_2 = 0$. Gravity acts vertically downwards so that $(G_1, G_2) = (0, -1)$.

When discussing the implementation of the Navier-Stokes equations in an [earlier example](#), we mentioned that `oomph-lib` allows the incompressible Navier-Stokes equations to be solved in the simplified, rather than the (default) stress-divergence form. We will demonstrate the use of this feature by solving the following problem:

Problem 2:

**The 2D driven cavity problem in a quarter circle domain with gravity,
using the simplified form of the Navier-Stokes equations**

Solve

$$Re u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + B_i + \frac{\partial^2 u_i}{\partial x_j^2}, \quad (1)$$

and

$$\frac{\partial u_i}{\partial x_i} = 0,$$

in the quarter-circle domain $D = \{x_1 \geq 0, x_2 \geq 0 \text{ and } x_1^2 + x_2^2 \leq 1\}$, subject to the Dirichlet boundary conditions

$$\mathbf{u}|_{\partial D} = (0, 0), \quad (2)$$

on the curved and left boundaries; and

$$\mathbf{u}|_{\partial D} = (1, 0), \quad (3)$$

on the bottom boundary, $x_2 = 0$. To make this consistent with Problem 1, we define the body force function as $(B_1, B_2) = (0, -Re/Fr)$.

Note that in Problem 2, the gravitational body force is represented by the body force rather than the gravity vector.

1.1.1 Switching between the stress-divergence and the simplified forms of the Navier-Stokes equations

The two forms of the Navier-Stokes equations differ in the implementation of the viscous terms, which may be represented as

$$\frac{\partial^2 u_i}{\partial x_j^2} \quad \text{or} \quad \frac{\partial}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right).$$

For an incompressible flow, $\partial u_i / \partial x_i = 0$, both forms are mathematically equivalent but the stress-divergence form is required for [problems with free surfaces](#), or for [problems in which traction boundary conditions are to be applied](#).

In order to be able to deal with both cases, `oomph-lib`'s Navier-Stokes elements actually implement the viscous term as

$$\frac{\partial}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \Gamma_i \frac{\partial u_j}{\partial x_i} \right).$$

By default the components of the vector Γ_i , are set to 1.0, so that the stress-divergence form is used. The components Γ_i are stored in the static data member

```
static Vector<double> NavierStokesEquations<DIM>::Gamma
```

of the `NavierStokesEquations<DIM>` class which forms the basis for all Navier-Stokes elements in `oomph-lib`. Its entries are initialised to 1.0. The user may over-write these assignments and thus re-define the values of Γ being used for a specific problem. [In principle, it is possible to use stress-divergence form for the first component of the momentum equations, and the simplified form for the second one, say. However, we do not believe that this is a particularly useful/desirable option and have certainly never used such (slightly bizarre) assignments in any of our own computations.]

1.1.2 Solution to problem 1

The figure below shows "carpet plots" of the velocity and pressure fields as well as a contour plot of the pressure distribution with superimposed streamlines for Problem 1 at a Reynolds number of $Re = 100$ and a ratio of Reynolds and Froude numbers (a measure of gravity on the viscous scale) of $Re/Fr = 100$. The velocity vanishes along the entire domain boundary, apart from the bottom boundary ($x_2 = 0$) where the moving "lid" imposes a unit tangential velocity which drives a large vortex, centred at $(x_1, x_2) \approx (0.59, 0.22)$. The pressure singularities created by the velocity discontinuities at $(x_1, x_2) = (0, 0)$ and $(x_1, x_2) = (1, 0)$ are well resolved. The pressure plot shows that away from the singularities, the pressure decreases linearly with x_2 , reflecting the effect of the gravitational body forces which acts in the negative x_2 — direction.



Figure 1.1 Plot of the velocity and pressure fields for problem 1 with $Re=100$ and $Re/Fr=100$, computed with adaptive Taylor-Hood elements.

1.1.3 Solution to problem 2

The next figure shows the computational results for Problem 2, obtained from a computation with adaptive Crouzeix-Raviart elements.



Figure 1.2 Plot of the velocity and pressure fields for problem 2 with $Re=100$ and $Re/Fr=100$, computed with adaptive Crouzeix-Raviart elements.

1.2 The code

We use a namespace `Global_Physical_Variables` to define the various parameters: The Reynolds number,

```
//==start_of_namespace=====
// Namespace for physical parameters
//=====
namespace Global_Physical_Variables
```

```
{
  /// Reynolds number
  double Re=100;
```

the gravity vector \mathbf{G} , and the ratio of Reynolds and Froude number, Re/Fr , which represents the ratio of gravitational and viscous forces,

```
/// Reynolds/Froude number
double Re_invFr=100;
```

```
/// Gravity vector
Vector<double> Gravity(2);
```

In Problem 2, gravity is introduced via the body force function \mathbf{B} which we define such that Problems 1 and 2 are equivalent. (We use the gravity vector $\mathbf{G} = (0, -1)$ to specify the direction of gravity, while indicating its magnitude by Re/Fr .)

```
/// Functional body force
void body_force(const double& time, const Vector<double>& x,
                Vector<double>& result)
{
  result[0]=0.0;
  result[1]=-Re_invFr;
}
```

Finally we define a body force function, which returns zero values, for use when solving Problem 1.

```
/// Zero functional body force
void zero_body_force(const double& time, const Vector<double>& x,
                    Vector<double>& result)
{
  result[0]=0.0;
  result[1]=0.0;
}
// end_of_namespace
```

1.3 The driver code

First we create a `DocInfo` object to control the output, and set the maximum number of spatial adaptations to three.

```
//==start_of_main=====
// Driver for QuarterCircleDrivenCavityProblem test problem
//=====
int main()
{
    // Set output directory and initialise count
    DocInfo doc_info;
    doc_info.set_directory("RESULT");
    // Set max. number of black-box adaptation
    unsigned max_adapt=3;
```

To solve problem 1 we define the direction of gravity, $\mathbf{G} = (0, -1)$, and set the entries in the `NavierStokesEquations<2>::Gamma` vector to $(1, 1)$, so that the stress-divergence form of the equation is used [In fact, this step is not strictly necessary as it simply re-assigns the default values.]

```
// Solve problem 1 with Taylor-Hood elements
//-----
{
    // Set up downwards-Gravity vector
    Global_Physical_Variables::Gravity[0] = 0.0;
    Global_Physical_Variables::Gravity[1] = -1.0;

    // Set up Gamma vector for stress-divergence form
    NavierStokesEquations<2>::Gamma[0]=1;
    NavierStokesEquations<2>::Gamma[1]=1;
```

Next we build problem 1 using Taylor-Hood elements and passing a function pointer to the `zero_body_force(...)` function (defined in the namespace `Global_Physical_Variables`) as the argument.

```
// Build problem with Gravity vector in stress divergence form,
// using zero body force function
QuarterCircleDrivenCavityProblem<RefineableQTaylorHoodElement<2> >
problem(&Global_Physical_Variables::zero_body_force);
```

Now problem 1 can be solved as in [the previous example](#).

```
// Solve the problem with automatic adaptation
problem.newton_solve(max_adapt);

// Step number
doc_info.number()=0;
// Output solution
problem.doc_solution(doc_info);
} // end of problem 1
```

To solve problem 2 we set the entries in the `NavierStokesEquations<2>::Gamma` vector to zero (thus choosing the simplified version of the Navier-Stokes equations), define $\mathbf{G} = (0, 0)$, and pass a function pointer to the `body_force(...)` function to the problem constructor.

```
// Solve problem 2 with Taylor Hood elements
//-----
{
    // Set up zero-Gravity vector
    Global_Physical_Variables::Gravity[0] = 0.0;
    Global_Physical_Variables::Gravity[1] = 0.0;
    // Set up Gamma vector for simplified form
    NavierStokesEquations<2>::Gamma[0]=0;
    NavierStokesEquations<2>::Gamma[1]=0;
    // Build problem with body force function and simplified form,
    // using body force function
    QuarterCircleDrivenCavityProblem<RefineableQTaylorHoodElement<2> >
    problem(&Global_Physical_Variables::body_force);
```

Problem 2 may then be solved as before.

```
// Solve the problem with automatic adaptation
problem.newton_solve(max_adapt);

// Step number
doc_info.number()=1;
// Output solution
problem.doc_solution(doc_info);
} // end of problem 2
} // end of main
```

1.4 The problem class

The problem class is very similar to that used in the [previous example](#), with two exceptions:

- We pass a function pointer to the body force function \mathbf{B} to the constructor and
- store the function pointer to the body force function in the problem's private member data.

```

//==start_of_problem_class=====
/// Driven cavity problem in quarter circle domain, templated
/// by element type.
//=====
template<class ELEMENT>
class QuarterCircleDrivenCavityProblem : public Problem
{
public:

    /// Constructor
    QuarterCircleDrivenCavityProblem(
        NavierStokesEquations<2>::NavierStokesBodyForceFctPt body_force_fct_pt);

    /// Destructor: Empty
    ~QuarterCircleDrivenCavityProblem() {}

    /// Update the after solve (empty)
    void actions_after_newton_solve() {}

    /// Update the problem specs before solve.
    /// (Re-)set velocity boundary conditions just to be on the safe side...
    void actions_before_newton_solve()
    {
        // Setup tangential flow along boundary 0:
        unsigned ibound=0;
        unsigned num_nod= mesh_pt()->nboundary_node(ibound);
        for (unsigned inod=0; inod<num_nod; inod++)
        {
            // Tangential flow
            unsigned i=0;
            mesh_pt()->boundary_node_pt(ibound, inod)->set_value(i, 1.0);
            // No penetration
            i=1;
            mesh_pt()->boundary_node_pt(ibound, inod)->set_value(i, 0.0);
        }

        // Overwrite with no flow along all other boundaries
        unsigned num_bound = mesh_pt()->nboundary();
        for (unsigned ibound=1; ibound<num_bound; ibound++)
        {
            unsigned num_nod= mesh_pt()->nboundary_node(ibound);
            for (unsigned inod=0; inod<num_nod; inod++)
            {
                for (unsigned i=0; i<2; i++)
                {
                    mesh_pt()->boundary_node_pt(ibound, inod)->set_value(i, 0.0);
                }
            }
        }
    } // end_of_actions_before_newton_solve

    /// After adaptation: Unpin pressure and pin redundant pressure dofs.
    void actions_after_adapt()
    {
        // Unpin all pressure dofs
        RefineableNavierStokesEquations<2>::
            unpin_all_pressure_dofs(mesh_pt()->element_pt());
        // Pin redundant pressure dofs
        RefineableNavierStokesEquations<2>::
            pin_redundant_nodal_pressures(mesh_pt()->element_pt());

        // Now pin the first pressure dof in the first element and set it to 0.0
        fix_pressure(0, 0, 0.0);
    } // end_of_actions_after_adapt

    /// Doc the solution
    void doc_solution(DocInfo& doc_info);

private:

    /// Pointer to body force function
    NavierStokesEquations<2>::NavierStokesBodyForceFctPt Body_force_fct_pt;

    /// Fix pressure in element e at pressure dof pdof and set to pvalue
    void fix_pressure(const unsigned &e, const unsigned &pdof,
                     const double &pvalue)
    {
        //Cast to proper element and fix pressure
        dynamic_cast<ELEMENT*>(mesh_pt()->element_pt(e))->
            fix_pressure(pdof, pvalue);
    } // end_of_fix_pressure
}; // end_of_problem_class

```

1.5 The problem constructor

We store the function pointer to the body force function in the private data member `Body_force_fct_pt`.

```

//==start_of_constructor=====
// Constructor for driven cavity problem in quarter circle domain
//=====
template<class ELEMENT>
QuarterCircleDrivenCavityProblem<ELEMENT>::QuarterCircleDrivenCavityProblem(
    NavierStokesEquations<2>::NavierStokesBodyForceFctPt body_force_fct_pt) :
    Body_force_fct_pt(body_force_fct_pt)

```

As usual the first task is to create the mesh. We now use the `RefineableQuarterCircleSectorMesh<ELEMENT>`, which requires the creation of a `GeomObject` to describe geometry of the curved wall: We choose an ellipse with unit half axes (i.e. a unit circle).

```

{
    // Build geometric object that parametrises the curved boundary
    // of the domain
    // Half axes for ellipse
    double a_ellipse=1.0;
    double b_ellipse=1.0;
    // Setup elliptical ring
    GeomObject* Wall_pt=new Ellipse(a_ellipse,b_ellipse);
    // End points for wall
    double xi_lo=0.0;
    double xi_hi=2.0*atan(1.0);
    //Now create the mesh
    double fract_mid=0.5;
    Problem::mesh_pt() = new
        RefineableQuarterCircleSectorMesh<ELEMENT>(
            Wall_pt,xi_lo,fract_mid,xi_hi);

```

Next the error estimator is set, the boundary nodes are pinned and the Reynolds number is assigned, as **before**

```

// Set error estimator
Z2ErrorEstimator* error_estimator_pt=new Z2ErrorEstimator;
dynamic_cast<RefineableQuarterCircleSectorMesh<ELEMENT>*>(
    mesh_pt())->spatial_error_estimator_pt(error_estimator_pt;

// Set the boundary conditions for this problem: All nodes are
// free by default -- just pin the ones that have Dirichlet conditions
// here: All boundaries are Dirichlet boundaries.
unsigned num_bound = mesh_pt()->nboundary();
for(unsigned ibound=0;ibound<num_bound;ibound++)
{
    unsigned num_nod= mesh_pt()->nboundary_node(ibound);
    for (unsigned inod=0;inod<num_nod;inod++)
    {
        // Loop over values (u and v velocities)
        for (unsigned i=0;i<2;i++)
        {
            mesh_pt()->boundary_node_pt(ibound,inod)->pin(i);
        }
    }
} // end loop over boundaries
//Find number of elements in mesh
unsigned n_element = mesh_pt()->nelement();
// Loop over the elements to set up element-specific
// things that cannot be handled by constructor: Pass pointer to Reynolds
// number
for(unsigned e=0;e<n_element;e++)
{
    // Upcast from GeneralisedElement to the present element
    ELEMENT* el_pt = dynamic_cast<ELEMENT*>(mesh_pt()->element_pt(e));
    //Set the Reynolds number, etc
    el_pt->re_pt() = &Global_Physical_Variables::Re;

```

Within this loop we also pass the pointers to Re/Fr , the gravity vector and the body-force function to the elements.

```

//Set the Re/Fr
el_pt->re_invfr_pt() = &Global_Physical_Variables::Re_invFr;
//Set Gravity vector
el_pt->g_pt() = &Global_Physical_Variables::Gravity;
//set body force function
el_pt->body_force_fct_pt() = Body_force_fct_pt;
} // end loop over elements

```

The `RefineableQuarterCircleSectorMesh<ELEMENT>` contains only three elements and therefore provides a very coarse discretisation of the domain. We refine the mesh uniformly twice before pinning the redundant pressure degrees of freedom, pinning a single pressure degree of freedom, and assigning the equation numbers, as **before**.

```

// Initial refinement level
refine_uniformly();
refine_uniformly();
// Pin redundant pressure dofs

```

```

RefineableNavierStokesEquations<2>::
    pin_redundant_nodal_pressures(mesh_pt()->element_pt());

// Now pin the first pressure dof in the first element and set it to 0.0
fix_pressure(0,0,0.0);

// Setup equation numbering scheme
cout << "Number of equations: " << assign_eqn_numbers() << std::endl;

} // end of constructor

```

1.6 Post processing

The post processing function remains the same as in the [previous examples](#).

```

//==start_of_doc_solution=====
/// Doc the solution
//=====
template<class ELEMENT>
void QuarterCircleDrivenCavityProblem<ELEMENT>::doc_solution(DocInfo& doc_info)
{
    ofstream some_file;
    char filename[100];
    // Number of plot points
    unsigned npts=5;
    // Output solution
    sprintf(filename,"%s/soln%i.dat",doc_info.directory().c_str(),
        doc_info.number());
    some_file.open(filename);
    mesh_pt()->output(some_file,npts);
    some_file.close();
} // end of doc_solution

```

1.7 Comments and Exercises

1. Try making the curved boundary the driving wall [Hint: this requires a change in the wall velocities prescribed in `Problem::actions_before_newton_solve()`. The figure below shows what you should expect.]



Figure 1.3 Plot of the velocity and pressure distribution for a circular driven cavity in which the flow is driven by the tangential motion of the curvilinear boundary.

1.8 Source files for this tutorial

- The source files for this tutorial are located in the directory:


```
demo_drivers/navier_stokes/circular_driven_cavity/
```

- The driver code is:

```
demo_drivers/navier_stokes/circular_driven_cavity/circular_driven_↵  
cavity.cc
```

1.9 PDF file

A [pdf version](#) of this document is available.