

NTNU

PROJECT TMA4500

Application of nek5000 to dispersion simulations

Author:

Magnus AARSKAUG RUD

Supervisor:

Anne Kværnø

Research Group Name

IMF

January 2016

NTNU

Abstract

Faculty Name

IMF

Master Thesis

Application of nek5000 to dispersion simulations

by Magnus AARSKAUG RUD

Sed ullamcorper quam eu nisl interdum at interdum enim egestas. Aliquam placerat justo sed lectus lobortis ut porta nisl porttitor. Vestibulum mi dolor, lacinia molestie gravida at, tempus vitae ligula. Donec eget quam sapien, in viverra eros. Donec pellen-tesque justo a massa fringilla non vestibulum metus vestibulum. Vestibulum in orci quis felis tempor lacinia. Vivamus ornare ultrices facilisis. Ut hendrerit volutpat vulputate. Morbi condimentum venenatis augue, id porta ipsum vulputate in. Curabitur luctus tempus justo. Vestibulum risus lectus, adipiscing nec condimentum quis, condimentum nec nisl.

CONTENTS

Abstract	i
Contents	ii
Notation	iv
1 Introduction	1
2 Problem description	2
2.1 The Incompressible Navier-Stokes equations	2
2.1.1 Boundary conditions	4
2.1.2 Weak formulation of N-S	5
2.2 The passive scalar equation	6
2.3 Resolving the turbulent term using LES	7
2.3.1 Filter	7
2.3.2 dynamic Smagorinsky-Lilly SGS-model	8
3 Numerical algorithms	11
3.1 Numerical Concepts on The Stokes problem	11
3.2 Finite element method	13
3.3 Spectral methods	14
3.4 Spectral element method	17
3.4.1 Filtering	20
3.4.2 Aliasing	21
3.4.3 Gordon-Hall algorithm	21
3.5 Time integration for incompressible N-S	23
3.5.1 Operator-splitting techniques	24
3.5.2 Operator integrating factor schemes (OIFS)	24
3.5.3 Fractional step	26
4 Application of Nek5000	28
4.1 Nek5000 Basics	28
4.2 Editable files	29
4.2.1 SIZE	30
4.2.2 .rea	30
4.2.3 .usr	31
4.3 Steps in the solver	31

4.4	Nek5000 for complex geometries	32
5	Implementation	33
5.1	Case 1: Gas dispersion in a simplified urban area	33
5.2	Case 2: Drag and lift on a cylinder	34
5.3	Advances in the mesh-generation routine	37
5.4	Additional projection algorithm	38
5.4.1	Application on the hill of Ekeberg	41
5.5	Spatial averaging routine	42
6	Results	43
6.1	Drag and lift on a cylinder	43
6.1.1	Parameter adjustments in Nek5000	44
6.2	Gas dispersion in a simplified urban area	44
6.2.1	Dynamic Smagorinsky	44
6.3	Discussion and Conclusion	45
A	Fundamental basics of numerical analysis	50
A.1	GLL-quadrature	50
A.2	Essential polynomials	50
A.3	Preliminary concepts	50
A.4	Lax-Milgram theorem	50
B	Variables and Functions in Nek5000	51
B.1	Variables	51
B.2	Functions	51
B.2.1	standard calculations found in math.f or navier1.f	51
B.2.2	Functions regarding mesh and distribution of GLL-points	53
B.2.3	Additional auxiliary functions implemented for this thesis	53
	Bibliography	55

NOTATION

CONVENTION we let subscript h denote the discretized variables

CHAPTER 1

INTRODUCTION

Sed ullamcorper quam eu nisl interdum at interdum enim egestas. Aliquam placerat justo sed lectus lobortis ut porta nisl porttitor. Vestibulum mi dolor, lacinia molestie gravida at, tempus vitae ligula. Donec eget quam sapien, in viverra eros. Donec pellentesque justo a massa fringilla non vestibulum metus vestibulum. Vestibulum in orci quis felis tempor lacinia. Vivamus ornare ultrices facilisis. Ut hendrerit volutpat vulputate. Morbi condimentum venenatis augue, id porta ipsum vulputate in. Curabitur luctus tempus justo. Vestibulum risus lectus, adipiscing nec condimentum quis, condimentum nec nisl. Aliquam dictum sagittis velit sed iaculis. Morbi tristique augue sit amet nulla pulvinar id facilisis ligula mollis. Nam elit libero, tincidunt ut aliquam at, molestie in quam. Aenean rhoncus vehicula hendrerit.

CHAPTER 2

PROBLEM DESCRIPTION

What should be said about the stokes problem and the infsup condition??

2.1 THE INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

The physics regarding fluids in motion are described mathematically by the Navier-Stokes (N-S) equations. The equations can be derived in many ways, and it is referred to [1] for a complete description of the necessary assumptions and simplifications. The general idea is to conserve momentum and mass in a control domain providing a system of two equations. In this thesis only the incompressible N-S equations will be considered, with the assumption of an incompressible flow the conservation of mass results in a divergence free restriction on the solution \mathbf{u} . Without further introduction the incompressible N-S equations are stated as

$$\begin{aligned}\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} &= -\nabla p + \frac{1}{Re} \Delta \mathbf{u} + \mathbf{f}, \\ \nabla \cdot \mathbf{u} &= 0.\end{aligned}\tag{2.1}$$

These equations have been studied for centuries and different physical situations lead to different simplifications and sets of equations. A common simplification is to omit the convective term, the resulting set of equations are referred to as the Stokes problem and is a common starting point for analysis of the full problem. Before attempting to solve these equations it is important to understand the role of each term and their mathematical influence on the problem.

- $\partial \mathbf{u} / \partial t$ - The time-derivative of the flow, for a steady state flow this term will be equal zero. The discretization of this term is often based on some implicit scheme.
- $\mathbf{u} \cdot \nabla \mathbf{u}$ - The convective term, describes the transport due to the flow itself on each of its components. The term is not present in Stokes problem. The mathematical operator corresponding to this term is non-linear and non-symmetric, and does therefore require an explicit scheme. Even with linear advection the operator quickly leads to instability.
- ∇p - The pressure gradient, Removal of this term results in a pure advection diffusion problem.
- Re - The Reynolds number defined as UL/ν where U is the velocity scale, L is the length scale and ν is the viscosity of the fluid. Re describes the relation between the largest length scales of the flow and the viscous length scales. Notice that for large Reynolds number the unstable non-linear term dominates the transportation. Turbulent flows are characterized by high Reynolds number.
- $\Delta \mathbf{u}$ - The diffusive term describes the natural diffusion of the fluid. The contribution of the diffusional term is inversely proportional to the Reynolds number. The corresponding mathematical operator stabilizes the system and it is therefore generally easier to solve the N-S equations for high-viscosity fluids. It should be mentioned that this term is actually a simplification of the reynolds stress tensor which can be made under the assumption of incompressibility. The more general formulation which will be used in chapter 2.3 is $2\nabla s_{ij}$, the tensor s_{ij} is known as the strain-rate tensor.
- \mathbf{f} - General term describing external body forces such as gravity. For incompressible flow the gravity term is incorporated in the pressure term, $\nabla p = \nabla p + \rho \mathbf{g}$.
- $\nabla \cdot \mathbf{u}$ - The divergence free condition from the mass equation.

For large Reynolds number the huge span in length scales requires mesh that is both very fine and very large if 2.1 are to be solved exactly. Because a fine mesh implies a high computational cost a direct numerical solution (DNS) is not feasible for problems of a certain geometrical complexity. There are many different approaches on how to resolve the turbulent term and in this thesis the main approach will be through Large Eddy

Simulations (LES) which will be discussed in section 2.3. The Navier-Stokes equations can only be solved if the boundary conditions are given. The boundary conditions are not stated explicitly with in 2.1 because they depend on the physical situation and belong as a specification to each individual case. The next subsection gives a quick overview of the different boundary conditions applied in this thesis.

2.1.1 BOUNDARY CONDITIONS

Depending on the kind of flow and geometry a particular problem different boundary conditions are applied. In this section \mathbf{n} and \mathbf{t} will denote the normal and tangent vector to the surface. **tangent vector to a twodimensional surface** The boundary conditions applied for the cases investigated in this thesis will be given the names I,O,SYM and W for Inflow, Outflow, Symmetry and Wall. Their mathematical formulation and physical interpretation are given as

- I - Inflow, defining the velocity field at a particular boundary. Mathematically this is equivalent to non-homogenous Dirchlet conditions.

$$\mathbf{u} = \mathbf{g}(\mathbf{x}, t). \quad (2.2)$$

- O - Outflow, letting the fluid flow “effortlessly” out through this boundary. Formally stated as

$$\frac{1}{Re} \nabla \mathbf{u} \cdot \mathbf{n} - p \mathbf{n} = 0. \quad (2.3)$$

- SYM - Symmetry, denying any flux through this boundary without disturbing the tangential velocity. Convenient to apply in an open channel where the streamwise direction is parallell to the boundary. Mathematically this is described as

$$\mathbf{u} \cdot \mathbf{n} = 0 \quad , \quad (\nabla \mathbf{u} \cdot \mathbf{t}) \cdot \mathbf{n} = 0. \quad (2.4)$$

- W - Wall, Representing a physical object. Also known as the no-slip condition which is based on the assumption that the fluid closest to the object moves with the same speed as the object itself. In this thesis all objects and geometries is kept

still so mathematically this is equivalent to homogenous Dirchlet conditions.

$$\mathbf{u} = 0. \quad (2.5)$$

2.1.2 WEAK FORMULATION OF N-S

The numerical algorithms applied in this thesis requires a weak formulation of 2.1. Before the weak form is derived a few operators will be defined in order to simplify the final expression.

$$(\mathbf{u}, \mathbf{v})_{L_2} = \int_{\Omega} \mathbf{u} \cdot \mathbf{v} d\Omega \quad (2.6)$$

$$\mathcal{A}(\mathbf{u}, \mathbf{v}) = (\nabla \mathbf{u}, \nabla \mathbf{v})_{L_2} \quad (2.7)$$

$$\mathcal{B}(\mathbf{u}, p) = (\nabla \cdot \mathbf{u}, p)_{L_2} \quad (2.8)$$

$$\mathcal{C}(\mathbf{w}; \mathbf{u}, \mathbf{v}) = (\mathbf{w} \cdot \nabla \mathbf{u}, \mathbf{v})_{L_2} \quad (2.9)$$

A weak formulation is obtained by multiplying with a test function \mathbf{v} and integrating over the entire domain.

$$\begin{aligned} \int_{\Omega} \frac{\partial \mathbf{u}}{\partial t} \cdot \mathbf{v} d\Omega + \int_{\Omega} (\mathbf{u} \cdot \nabla) \mathbf{u} \cdot \mathbf{v} d\Omega &= - \int_{\Omega} \nabla p \cdot \mathbf{v} d\Omega + \nu \int_{\Omega} \Delta \mathbf{u} \cdot \mathbf{v} d\Omega + \int_{\Omega} \mathbf{f} \cdot \mathbf{v} d\Omega \\ \int_{\Omega} (\nabla \cdot \mathbf{u}) \mathbf{q} d\Omega &= 0. \end{aligned} \quad (2.10)$$

Introducing the compact inner product notation and applying the divergence theorem on the right hand side of the first equation yields

$$\begin{aligned} \left(\frac{\partial \mathbf{u}}{\partial t}, \mathbf{v} \right) + (\mathbf{u} \cdot \nabla \mathbf{u}, \mathbf{v}) &= (\nabla \cdot \mathbf{v}, p) - \nu (\nabla \mathbf{u}, \nabla \mathbf{v}) + (\mathbf{f}, \mathbf{v}) \\ (\nabla \cdot \mathbf{u}, q) &= 0. \end{aligned} \quad (2.11)$$

The contributions from the boundary as a result from the application of the divergence theorem is included in the last force term. Does the boundary integral needs to be adressd?

The choice of search spaces for the velocity and pressure will be justified in chapter 3, and will just be stated here in order to present the weak formulation. Let $V \subset H^1(\Omega)^3$ and $Q \subset L^2(\Omega)$ be subspaces that include the boundary conditions given by any particular flow situation. By using the notation introduced in 2.9 the weak formulation of the incompressible N-S equations can be stated as:

Find $(\mathbf{u}, p) \in V \times Q$ such that

$$\begin{aligned} \left(\frac{\partial \mathbf{u}}{\partial t}, \mathbf{v}\right) + \mathcal{C}(\mathbf{u}; \mathbf{u}, \mathbf{v}) &= \mathcal{B}(\mathbf{v}, p) - \nu \mathcal{A}(\mathbf{u}, \mathbf{v}) + (\mathbf{f}, \mathbf{v}), \\ \mathcal{B}(\mathbf{u}, q) &= 0, \end{aligned} \quad (2.12)$$

$\forall (\mathbf{v}, q) \in V \times Q$.

In order to solve this equation numerically, everything has to be discretized and expressed using a particular set of basis functions. This procedure will be discussed more thoroughly in chapter 3, but the idea is that the solution (\mathbf{u}, p) is approximated by a discretized solution (\mathbf{u}_h, p_h) and the bilinear operators can be represented by matrices. The discretized system of equations can be stated as

$$M \frac{\partial \mathbf{u}_h}{\partial t} + C(\mathbf{u}_h) \mathbf{u}_h = D p_h - \nu A \mathbf{u}_h + M \mathbf{f}_h, \quad (2.13)$$

$$D^T \mathbf{u}_h = 0. \quad (2.14)$$

2.2 THE PASSIVE SCALAR EQUATION

The N-S equations explain how a fluid will behave and solving it provides a complete pressure-velocity field of the domain of interest, but it does not provide the answer of how heat will distribute in this flow, or how a gas will spread. The equation corresponding to this physical problem will be referred to as the passive scalar equation for any scalar ϕ and is stated as

$$\rho c_p \left(\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi \right) = \nabla \cdot (k \nabla \phi) + q_{vol}. \quad (2.15)$$

The constants k , and ρc_p are interpreted depending on the scalar transported. For dispersion of neutral gas with ϕ as the volume concentration of the gas they resemble the viscosity and mass flux. The last term on the right hand side q_{vol} is the source term and is not included in this thesis since all gas enters the control volume as a boundary condition.

write more about this equation and how it is solved ...

2.3 RESOLVING THE TURBULENT TERM USING LES

tydeliggjør filteret!, og bruk en annen τ , filtervidde, tydeligg diff antakelse When DNS is not feasible due to high Reynolds number LES is one of the most powerful tools for simulating turbulent flows. The idea is based on the fact that the small turbulent structures behave homogenously and are therefore easy to model. This way the larger structures driven by geometry, inflow conditions and external forces can be simulated using a coarser grid while the effect of the small structures are modelled.

Some source on the importance of SGS-model

2.3.1 FILTER

The idea behind LES starts with defining a filter, which separates the modelled structures from the resolved ones. A filter in its general form is given as

$$U^r(\mathbf{x}, t) = \int_{\Omega} G_r(\mathbf{r}, \mathbf{x}) U(\mathbf{x} - \mathbf{r}, t) d\mathbf{r} \quad (2.16)$$

Define $U!!$ and filter with l_r In order to perform a LES a filter width a certain width Δ has to be chosen, this is commonly chosen to be the grid size.

It is generally assumed that the filter commutes with the differential operators ∇ , Δ and $\partial/\partial t$, these assumptions allows the filtered N-S equations to be stated as

$$\begin{aligned} \frac{\partial \mathbf{u}^r}{\partial t} + \mathbf{u}^r \cdot \nabla \mathbf{u}^r &= -\nabla p^r + \nu \Delta \mathbf{u}^r - \nabla \cdot \tau, \\ \nabla \cdot \mathbf{u}^r &= 0. \end{aligned} \quad (2.17)$$

Where τ in this case denotes the subgrid-scale (SGS) stress given as

$$\tau_{ij}(u_i, u_j) = (u_i u_j)^r - u_i^r u_j^r. \quad (2.18)$$

This tensor is a consequence of applying the filter on the non-linear advection term, and it is this term that is modelled by subgrid-scale model. (When was this introduced?)

2.3.2 DYNAMIC SMAGORINSKY-LILLY SGS-MODEL

The problem is now reduced to modelling the tensor τ_{ij} , one of the most common SGS-models is the dynamic Smagorinsky-Lilly model. The initial progress of this model was made by Lilly in 1967 [2] suggesting the following model for the SGS-tensor

$$\tau_{ij} = -2C_s l^2 \mathcal{S}^r s_{ij}^r, \quad (2.19)$$

$$s_{ij}^r = \frac{1}{2} \left(\frac{\partial u_i^r}{\partial x_j} + \frac{\partial u_j^r}{\partial x_i} \right), \quad (2.20)$$

$$\mathcal{S}^r = \sqrt{2s_{ij}^r s_{ij}^r}. \quad (2.21)$$

Where l denotes the filter width, which for this thesis is equivalent to the grid size. The resolved strain rate s_{ij}^r can be calculated from the filtered velocity and the problem is now reduced to determining the constant C_s . There were several attempts to determine this constant for the entire domain, but finally in 1991 [3] a dynamic subgrid-scale model was presented calculating a C_d which replaces $C_s l^2$ in 2.21. C_d is called the dynamic Smagorinsky constant, and varies in both time and space. The general idea is that C_d is independent of the filter width and from this assumption an expression is developed for the dynamical constant.

Let a, b denote two distinct filters with corresponding filter widths l_a, l_b . Throughout this thesis l_a will be the grid size and $l_b \approx 2l_a$. Filter b is often referred to as the test filter and is only included to provide an estimation of the dynamic Smagorinsky constant. Let τ_{ij} and T_{ij} denote the stresses based on single- and double filtering operations on the N-S equations

$$\begin{aligned} \tau_{ij} &= (u_i u_j)^a - u_i^a u_j^a, \\ T_{ij} &= ((u_i u_j)^a)^b - (u_i^a)^b (u_j^a)^b. \end{aligned} \quad (2.22)$$

Applying the b filter on the first tensor in eq 2.22 allows us to define a new tensor L_{ij} which depends only on the a -filtered variables. The identity is known as the Germano identity and was first introduced in 1991 [3],

$$L_{ij} = T_{ij} - (\tau_{ij})^b = (u_i^a u_j^a)^b - (u_i^a)^b (u_j^a)^b. \quad (2.23)$$

This tensor now depends on the a -filtered solution and not the resolved one, hence the identity in 2.23 provides a computable expression for L_{ij} .

Substituting the stress-tensors with their corresponding expression from 2.21 and assuming a dynamic constant unaffected by the filter one obtains an approximation for L_{ij} which is also computable,

$$\begin{aligned} L_{ij} &\approx 2C_s l_b^2 \mathcal{S}^{ab} s_{ij}^{ab} - 2(C_s l_a^2 \mathcal{S}^a s_{ij}^a)^b \\ &\approx 2C_s l_a^2 [\lambda^2 \mathcal{S}^{ab} s_{ij}^{ab} - (\mathcal{S}^a s_{ij}^a)^b] \\ &= 2C_d M_{ij}. \end{aligned} \tag{2.24}$$

$$M_{ij} = \lambda^2 \mathcal{S}^{ab} s_{ij}^{ab} - (\mathcal{S}^a s_{ij}^a)^b \tag{2.25}$$

$$C_d = C_s l_a^2 \tag{2.26}$$

$$\lambda = l_b/l_a \tag{2.27}$$

Minimizing the mean-square error between the exact L_{ij} as expressed in 2.23 and the Boussinesq-based approximation in 2.24 yields the best approximation for the dynamic Smagorinsky constant

$$C_s = \frac{M_{ij} L_{ij}}{2M_{kl} M_{kl}}. \tag{2.28}$$

Note that the double indices implies summing. This expression is however not a stable option and to deal with this most implementations apply some sort of mean or smoothening in either time and/or space when calculating the constant. In this thesis the smoothening is done in both time and space for the denominator and the numerator in 2.28. Another interesting property is that the constant C_d and the final SGS-tensor $\tau = -2C_d \mathcal{S}^a s_{ij}^a$ are independent of the filter width, the only necessary variable is the coefficient $\lambda = l_b/l_a$. The assumption made in this model is that turbulence behaves as diffusion, similar to the kinematic viscosity a turbulent viscosity ν_t is defined which for this case is given as $\nu_t = C_s \mathcal{S}^a$.

Let us end this section by stating the filtered N-S equations with the LES using dynamical Smagorinsky subgrid scale model, and remember that the diffusive term is written

in general terms as $2\nabla \cdot \nu s_{ij}$.

$$\begin{aligned} \frac{\partial \mathbf{u}^a}{\partial t} + \mathbf{u}^a \cdot \nabla \mathbf{u}^a &= -\nabla p^a + 2\nabla \cdot (\nu + \nu_t) s_{ij}^a \\ \nabla \cdot \mathbf{u}^a &= 0. \end{aligned} \tag{2.29}$$

Notice that if ν_t were to be a constant in the entire domain this equation would be equivalent to the one for a fluid with viscosity $\nu' = \nu + \nu_t$. The idea is that ν_t will be larger when the subgrid-structures are significant and closer to zero when the flow is laminar. This is just one of many types of models but is attractive due to its simple derivation from physical principles.

CHAPTER 3

NUMERICAL ALGORITHMS

Ta med konkrete konvergensteoremer!

Rettskrivning!

Dropp hattfunksjoner, make ediff pretty! figur som viser energispekteret, ha et konkret eksempel med

3.1 NUMERICAL CONCEPTS ON THE STOKES PROBLEM

In the previous chapter the N-S equations was presented and reformulated in several ways without any details on how to actually solve the equations. This chapter aims to give a more detailed description of the solution methods applied. The choice of algorithms and solution spaces required a more thorough analysis which is normally performed on the steady Stokes problem. The steady Stokes problem does not include the convection term but the highest order terms are all present and is therefore a valid equation to perform this necessary analysis [4]. Analysis of the time schemes applied will be left for a later chapter. The steady Stokes problem with homogenous boundary conditions is given as

$$\begin{aligned} -\mu\Delta\mathbf{u} + \nabla p &= \mathbf{f}, \\ \nabla \cdot \mathbf{u} &= 0, \\ \mathbf{u} &= 0 \text{ on } \partial\Omega. \end{aligned} \tag{3.1}$$

Applying the weak formulation to the Stokes problem implies a minimum requirement on the spaces for \mathbf{u} and p , and their testfunctions. These spaces will be defined as

$$\begin{aligned} H_0^1(\Omega) &= \{v \in H^1(\Omega) \mid v = 0 \text{ on } \partial\Omega\}, \\ L_0^2(\Omega) &= \left\{q \in L^2(\Omega) \mid \int_{\Omega} q dx = 0\right\}, \end{aligned} \quad (3.2)$$

The formulation can easily be extended to include inhomogenous Dirchlet conditions on \mathbf{u} by defining a lifting function as described in [5]. Note also that the pressure is only present through its gradient and is therefore not uniquely defined if the extra constraint on the mean is defined, hence the 0 in L_0^2 . The weak form can now be stated as

Find $(\mathbf{u}, p) \in H_0^1(\Omega)^3 \times L_0^2(\Omega)$ such that

$$\begin{aligned} \mathcal{B}(\mathbf{v}, p) + \nu \mathcal{A}(\mathbf{u}, \mathbf{v}) &= (\mathbf{f}, \mathbf{v}), \\ \mathcal{B}(\mathbf{u}, q) &= 0. \end{aligned} \quad (3.3)$$

$$\forall (\mathbf{v}, q) \in H^1(\Omega)^3 \times L^2(\Omega).$$

The numerical solution of this problem requires a discrete formulation of the weak form, with $(\mathbf{u}_h, p_h) \in V \times Q$ as the discrete solution. The discrete spaces V, Q are subspaces of H_0^1, L_0^2 equipped with the discrete H_1 - and L^2 -norm denoted $\|\cdot\|_V$ and $\|\cdot\|_Q$. For the discrete weak form to be well-posed it has to meet the requirements stated by the inf-sup condition,

$$\inf_{q \in Q} \sup_{\mathbf{v} \in V} \frac{\mathcal{B}(v, q)}{\|\mathbf{v}\|_V \|q\|_Q} \geq \beta. \quad (3.4)$$

β is some positive constant. This condition often known as the Babuska-Brezzi condition is often met by creating a staggered grid, such that the pressure and the velocity are evaluated at different points.

Too early for this part?? For a Spectral Element formulation of this problem which will be elaborated in chapter 3.4, a common choice of subspaces (X, Q) is $(P_N \cap H_0^1, P_{N-2} \cap L_0^2)$. This will be referred to as the $P_N P_{N-2}$ -formulation where P_N denotes the space of polynomials up to degree N . It was however proved by Guermond in [6] that the fractional step algorithm requires a form of the Stokes problem which automatically fullfills the inf-sup condition and therefore does not require any particular discrete subspaces to be chosen. For the sake of convenience the polynomial degree of the pressure

and the velocity will be the same and the discrete pair of subspaces (X, Q) is chosen to $(P_N \cap H_0^1, P_N \cap L_0^2)$, known as the $P_N P_N$ -formulation.

Before the analysis of the N-S equations can be taken any further the theory behind the Spectral Element Method will be presented in the following subsections.

3.2 FINITE ELEMENT METHOD

Finite element method (FEM) is one of the most widely used numerical methods applied on problems within construction, flow simulation and many other areas. It offers a precise mathematical foundation and due to the connectivity properties of the elements it guaranties a sparse system. The decomposition of the geometrical domain into a finite amount of elements chosen according to the problem wanted to solve, makes it possible to create general algorithms applicable to all kinds of geometries. For the full mathematical foundation of FEM it will be referred to [5], but some of the key properties will be stated here in order to provide a thorough understanding of the spectral element method (SEM). Throughout this section p denotes the polynomial degree of the basis-functions, h represents the average grid spacing, E is the total number of elements and d is the number of dimensions.

FEM provides an algorithm for solving any well-posed boundary value problem (BVP) and the mathematical formulation is obtained by first finding the Galerkin formulation with a corresponding search space X and then choosing a discrete subspace $X_h^p \subset X$ spanned by the finite element basis functions $\{\phi_i^p\}_{i \in (1,E)}$. The key property of the basis functions is that they only have local support in a very small part of the domain. This is what gives rise to the resulting sparse linear system. By increasing the polynomial order the number of grid points used to define the polynomial will need to increase as well. This implies either reducing the distance between the grid points or increasing the support of each basis function. Both approaches will reduce the sparsity of the final matrix. Another important aspect of FEM is the treatment of the domain Ω , on which a triangulation $\{\mathcal{T}_h\}$ is defined such that the original domain is divided into elements. By defining a reference element and a general mapping function, all the local contributions can be calculated by a generalized quadrature rule before being added to

the global system of equations. This is a process tailored for parallelization, and can be generalized for a wide range of problems.

FEM is called a projection method since the solution $u_h \in X_h^p$ is a projection of the actual solution u of the BVP onto the discrete space X_h^p . Provided that the initial BVP is well-posed there exists to constants $M, \alpha > 0$ known as the bounded and coercivity constant such that the error of the solution can be reduced to a pure interpolation error. The result is known as Cea's lemma,

$$\|u - u_h\|_X \leq \frac{M}{\alpha} \|u - I_h u\|_X. \quad (3.5)$$

Where I_h is the interpolation operator.

does this interpolation operator need to be defined or explained in any way ?

Before this section ends it is important to understand the two ways to increase accuracy and the effects these two ways have on the algorithm. Assume the solution of the BVP to be infinitely smooth and the domain be sufficiently regular. This yields an error $e = Ch^p$, C being some positive constant. Factors such as geometric complexity, condition-number, non-linear operators and the continuity of the solution will all provide slightly more complicated error estimates. However for a simpler BVP such as the Poisson problem on the unit square, the error estimate is valid. performing a h -refinement will lead to an algebraic convergence of order p , while the sparsity of the system is conserved and the total algorithm does not change in any other way than increasing the number of elements. Keeping h constant and increasing p will provide spectral convergence, but the sparsity will be reduced and all integrals solved will require quadrature rules of higher order. A formal statement and numerical validation of the error estimate can be found in [4] chapter 2.6.

3.3 SPECTRAL METHODS

Spectral methods (SM) share a some of the mathematical ideas as FEM, but are not as widely used in real life problems. There are many ways to apply SM, and in this thesis only the Galerkin version with numerical integration (known as G-NI) will be considered and will be referred to only as SM. For a full introduction to SM and its applications

to BVP see [7]. SM can be reduced to a interpolation problem such as FEM, and are very interesting from a theoretical point of view due to its spectral convergence rate which allows you to obtain solutions of extremely high accuracy. The most important draw-back of SM are the difficulties with applications to complex geometries. Although the system of equations surging from a BVP can be constructed in an elegant way it is rarely sparse and often result in expensive calculations.

Applying SM on a BVP in one dimension requires a set of basis functions $\{\psi_i\}_N$ defined on the whole domain Ω . The discrete space $X_h(\Omega)$ spanned by the basis functions involves all polynomials up to degree N . A function u is projected onto X_h by the relation

$$u_h(x) = \sum_{i=0}^N a_i \psi_i(x). \quad (3.6)$$

Where the coefficients a_i are called the expansion coefficients. There are many possible choices for the basis and the belonging coefficients, in this thesis and the algorithms used the functions ψ_i will be the Lagrange polynomials based on the Gauss-Lobatto-Legendre (GLL) nodes. The reason for choosing these nodes is because it enables us to apply the Gauss-Lobatto quadrature rule. This is one of several existing Gauss-quadratures, and the only one allowing fixed endpoints which is the case for this thesis. For more detailed information on GL-quadrature and other quadrature rules it is referred to [8].

The GLL-nodes $\{\xi_i\}_{N+1}$ are given as the solutions of the equation

$$(1 - \xi^2)L'_N(\xi) = 0. \quad (3.7)$$

L_N being the Legendre polynomial of degree N , defined from the Sturm-Louville problem

$$\frac{d}{dx} \left[(1 - x^2) \frac{d}{dx} L_n(x) \right] + n(n+1)L_n(x) = 0. \quad (3.8)$$

With equations 3.7 and 3.8 the local spectral basis functions ψ_j can be stated as

$$\psi_j(x) = \prod_{i \neq j}^N \frac{x - x_i}{x_j - x_i}. \quad (3.9)$$

$\{x_i\}$ being the solutions to 3.7. Note that $\psi_j(x_i) = \delta_{ij}$. The expansion coefficients in 3.6 are then chosen as $a_i = u_i := u(x_i)$.

This definition of the expansion coefficients is very convenient since the actual value of the function in any point can just be read directly from the coefficients without having to sum all the contributions from the different polynomials. Creating a basis for 2 and 3 dimensions is done simply by taking the tensor product of the basis functions in each direction. let i, j, k denote the one dimensional indexes in each direction running from 1 to N while m, l, n denote three dimensional indexes running from 1 to N^d . for 3 dimensions the basis functions Ψ is given as

$$\Psi_l(\mathbf{x}) = \psi_i(x)\psi_j(y)\psi_k(z). \quad (3.10)$$

This expansion to multiple dimensions preserves the $\Psi_l(\mathbf{x}_m) = \delta_{lm}$. In order to clarify some of the concepts the SM approach will be applied on the Helmholtz equation

$$-\Delta u + \lambda u = f \quad \text{in } \Omega, \quad (3.11)$$

$$u = 0 \quad \text{on } \partial\Omega. \quad (3.12)$$

Ω will for this example be defined as the unit square $[-1, 1]^2$. Let us start by defining the space $V = H_0^1(\Omega)$ and assuming $f \in L^2(\Omega)$. The weak formulation after applying the divergence theorem is the given as

Find $u \in V$ st.

$$\int_{\Omega} \nabla u \cdot \nabla v d\Omega + \lambda \int_{\Omega} u v d\Omega = \int_{\Omega} f v d\Omega \quad \forall v \in V \quad (3.13)$$

In order to solve this using SM the discrete space $V_h \subset V$ is defined as $\text{span}\{\Psi_l\}$ following the preceding definitions the discrete weak formulation is stated as Find $u_h \in V_h$ st.

$$\sum_l \left(u_l \int_{\Omega} \nabla \Psi_l \cdot \nabla \Psi_m d\Omega + u_l \lambda \int_{\Omega} \Psi_l \Psi_m d\Omega \right) = \int_{\Omega} f \Psi_m d\Omega \quad \forall \Psi_m \in V_h. \quad (3.14)$$

The following step of this particular SM method is evaluating the integrals by using the GLL-quadrature rule, the resulting system of equations is then given as

$$\sum_l \left(u_l \sum_n \rho_n \nabla \Psi_l(\mathbf{x}_n) \cdot \nabla \Psi_m(\mathbf{x}_n) + u_l \lambda \sum_n \rho_n \Psi_l(\mathbf{x}_n) \Psi_m(\mathbf{x}_n) \right) \quad (3.15)$$

$$= \sum_n \rho_n f \Psi_m(\mathbf{x}_n) \quad \forall \Psi_m(\mathbf{x}_n) \in V_h. \quad (3.16)$$

ρ_n is the quadrature weight for the n th node, and \mathbf{x}_n is the vector containing the coordinates to the n th node. Note that all the indices $l, m, n = 1, \dots, N_x N_y$. This can be written in a compact matrix form as

$$(A + \lambda M)u_h = \tilde{f}. \quad (3.17)$$

Where the elements in the matrices and vectors are given as

$$\begin{aligned} A_{lm} &= \sum_n \rho_n \nabla \Psi_l(\mathbf{x}_n) \cdot \nabla \Psi_m(\mathbf{x}_n), \\ M_{lm} &= \sum_n \rho_n \Psi_l(\mathbf{x}_n) \Psi_m(\mathbf{x}_n) = \rho_l \delta_{lm}, \\ (u_h)_l &= u(\mathbf{x}_l), \\ \tilde{f}_m &= \sum_n \rho_n f(\mathbf{x}_n) \Psi_m(\mathbf{x}_n) = \rho_m f(\mathbf{x}_j). \end{aligned} \quad (3.18)$$

From these equations it is clear that the mass matrix M is diagonal and the right hand side vector \tilde{f} is easily calculated, while the stiffness matrix A is symmetric but full.

error/ convergence theorem

3.4 SPECTRAL ELEMENT METHOD

In the early 1980's the idea to combine FEM and SM came along in order to obtain the robustness and sparse properties of FEM combined with the spectral convergence rate provided by SM. The result was the Spectral element method. Several formulations was investigated mainly by Patera and Maday in the papers [9], [10], [11] with important contributions from Fischer, Rønquist and several more. It is important to understand that when solving the N-S equations the efficiency of the solution method is extremely important. The algorithm has to be parallelizable and the development of the super

computers and computational clusters has played an important role in deciding the method applied today. The idea is to divide the domain of the BVP into elements as in FEM and then use spectral basis functions of higher degree with support only within one single element.

In the previous subsection the power of spectral methods was illustrated on the unit square in two dimensions. But the limitations when it comes to more complex geometry rapidly affects the spectral convergence rate. Let $\hat{\Omega}$ be the reference element $[-1, 1]^d$, the standard procedure when working on a deformed geometry Ω with SM is to first create a map $\mathcal{F} : \hat{\Omega} \rightarrow \Omega$. An example of this map is the Gordon-Hall procedure described in chapter 3.4.3. The Jacobian is then given as the transposed tensor derivative of \mathcal{F} , which in two dimension is written as

$$\mathbf{J} = (\nabla \otimes \mathcal{F})^T = \begin{bmatrix} \frac{\partial \mathcal{F}_1}{\partial x} & \frac{\partial \mathcal{F}_1}{\partial y} \\ \frac{\partial \mathcal{F}_2}{\partial x} & \frac{\partial \mathcal{F}_2}{\partial y} \end{bmatrix}, \quad (3.19)$$

$$J = \det(\mathbf{J}). \quad (3.20)$$

This allows us to transform both derivatives and integrals to the reference domain, let $\boldsymbol{\xi} = [\xi, \eta]^T$ denote the axis in the reference domain corresponding to $\mathbf{x} = [x, y]^T$ in the deformed domain. The transformation is performed according to the following identities

$$\begin{aligned} d\mathbf{x} &= \mathbf{J}d\boldsymbol{\xi} \\ \int_{\Omega} f(\mathbf{x})d\mathbf{x} &= \int_{\hat{\Omega}} \hat{f}Jd\boldsymbol{\xi} \\ \nabla u &= \mathbf{J}^{-T}\hat{\nabla}\hat{u}. \end{aligned} \quad (3.21)$$

Here \hat{u}, \hat{f} are obtain by simply substituting \mathbf{x} with $\mathcal{F}(\boldsymbol{\xi})$ and $\hat{\nabla}$ is the partial differential operator wrt. $\boldsymbol{\xi}$. The important thing to notice here is that whenever an integral is solved and a derivative is introduced the Jacobian appears in the equation. When applying the GLL-quadrature to solve the integrals equality is guaranteed if and only if the function integrated is of polynomial degree $2n - 1$ or less, and the error gets bigger with increasing polynomial degree. A higher order jacobian could imply a large error in the quadrature.

Although the whole domain Ω is deformed, the deformation of each element $\{\Omega_k\}$ is normally a lot less crucial. This gives SEM a huge advantage and allows it to obtain

accurate results even in complicated domains.

Let us again consider the Helmholtz problem 3.12, but this time on a more general domain Ω . The set of elements $\{\Omega_k\}$ is defined such that $\Omega_i \cap \Omega_j$ is either empty, a vertex or a line and $\Omega = \bigcup_{k=1}^K \Omega_k$. The weak formulation in equation 3.13 can then be written according to the SEM formulation

For all elements Ω_k Find $u_{h,k} \in X_k^N$ such that

$$\int_{\Omega_k} \nabla u \cdot \nabla v d\Omega + \lambda \int_{\Omega_k} u v d\Omega = \int_{\Omega_k} f v d\Omega \quad \forall v \in X_k^N. \quad (3.22)$$

Where $X_k^N = H_0^1(\Omega_k) \cap \mathbb{P}^N(\Omega_k)$. The same discretization procedure as performed for the pure spectral case is now done for each of the sub domains Ω_k ,

$$\sum_i \left(u_i \int_{\Omega_k} \nabla \psi_i \cdot \nabla \psi_j d\Omega + u_i \lambda \int_{\Omega_k} \psi_i \psi_j d\Omega \right) = \int_{\Omega_k} f \psi_j d\Omega \quad \forall \psi_j \in V_h. \quad (3.23)$$

Since the elements can be deformed a Gordon-Hall map is constructed to map the coordinates to the reference element $\hat{\Omega} = [-1, 1]^d$. Applying the identities from 3.21 to 3.23 yields

$$\begin{aligned} \sum_i \left(u_i \int_{\hat{\Omega}_k} (\mathbf{J}^{-T} \hat{\nabla} \hat{\psi}_i)^T (\mathbf{J}^{-T} \hat{\nabla} \hat{\psi}_j) J d\hat{\Omega} + u_i \lambda \int_{\hat{\Omega}_k} \hat{\psi}_i \hat{\psi}_j J d\hat{\Omega} \right) &= \int_{\hat{\Omega}_k} \hat{f} \hat{\psi}_j J d\hat{\Omega} \quad \forall \hat{\psi}_j \in V_h. \\ \sum_i \left(u_i \int_{\hat{\Omega}_k} \hat{\nabla}^T \hat{\psi}_i \mathbf{J}^{-1} \mathbf{J}^{-T} \hat{\nabla} \hat{\psi}_j J d\hat{\Omega} + u_i \lambda \int_{\hat{\Omega}_k} \hat{\psi}_i \hat{\psi}_j J d\hat{\Omega} \right) &= \int_{\hat{\Omega}_k} \hat{f} \hat{\psi}_j J d\hat{\Omega} \quad \forall \hat{\psi}_j \in V_h. \end{aligned} \quad (3.24)$$

add a short argument for the danger of higher order jacobian ——— Notice how the integrals depend on the Jacobian \mathbf{J} and its determinant J . ——— the integral corresponding to the diffusion operator contains The local matrices A_k, M_k and the loading vector f_k are gathered from each element. Equivalently as for FEM the global matrices has to be assembled from all the local matrices corresponding to each subdomain. This procedure is general and can be performed on almost any deformed domain as oppose to SM.

Convergence estimate!!

3.4.1 FILTERING

Although SEM provides spectral convergence in space and 2nd or 3rd order accuracy in time the stability of a straight forward implementation is not guaranteed, spurious nodes appear as shown in chapter 2.4.1.2 in [4]. In [12] a filter-based stabilization is introduced for SEM applied on Navier Stokes equation. The idea is to project a part $0 < \alpha < 1$ of the highest mode onto a polynomial space of lower order, explicitly they define the filter F_α as

$$F_\alpha = \alpha I_{N-1} + (1 - \alpha)I_d. \quad (3.25)$$

Where I_{N-1} is the projector from \mathbb{P}_N to \mathbb{P}_{N-1} and I_d is the identity operator. α is recommended to be somewhere in the interval $(0.05, 0.3)$.

The explication of why F_α has a filtering effect is best explained by considering the more general explication by Pasquetti and Xu in [13]. A quick demonstration of how the filter works will however be given here.

Let $u = \sum_{i=0}^N \hat{u}_i L_i$ be the solution to some PDE, where L_i denote the Lagrange polynomial of order i and \hat{u}_i the corresponding coefficient. The effect of the filter can be given as

$$F_\alpha u = (1 - \alpha)\hat{u}_N L_N + \hat{u}_{N-1} L_{N-1} + (\hat{u}_{N-2} + \alpha\hat{u}_N) L_{N-2} + \sum_{i=0}^{N-3} \hat{u}_i L_i. \quad (3.26)$$

From this identity the effect of the filter becomes clear, it is simply removing a part of the highest order mode N to the mode $N - 2$. The rest of the coefficients remain unchanged. For a full derivation and discussion on this matter it is referred to chapter 6.5.1 in [4].

The filter is proved to be a very effective stabilization method and it preserves the spectral convergence rate. Another interesting property is that the filtering procedure does not imply dissipation of energy, let the energy norm be defined as $E(u) = \|u\|_{L_2}^2$. By applying Parseval's identity [14] the difference in energy between the original solution

and the filtered solution is given as

$$\epsilon_{diff} = E(u) - E(F_\alpha u) \quad (3.27)$$

$$= 2\alpha \hat{u}_N (\hat{u}_N \|L_N\|^2 + \hat{u}_{N-2} \|L_{N-2}\|^2) - \alpha^2 \hat{u}_N^2 (\|L_N\|^2 + \|L_{N-2}\|^2) \quad (3.28)$$

$$\approx \frac{2\alpha}{N} \left[\left(1 - \frac{\alpha}{2}\right) \hat{u}_N^2 + \hat{u}_N \hat{u}_{N-2} \right]. \quad (3.29)$$

Which can take both positive and negative values depending on the sign and size of $\hat{u}_N \hat{u}_{N-1}$. By applying the known norm of the Legendre polynomials the deduced absolute error ϵ_{diff} of the filtered energy is of order $\epsilon_{diff} \sim \alpha/N$. The approximation $\|L_N\|^2 \approx \|L_{N-2}\|^2 \approx 1/N$ have been used to achieve the result in 3.29.

3.4.2 ALIASING

When evaluating the integral surging from the non-linear term in the N-S equations the polynomial to be integrated is order $2N + (N - 1)$. The quadrature rule applied to solve the integrals are only exact up to an order $2N - 1$, hence the error surging from this evaluation can be of significant size. Applying a non-sufficient quadrature to an integral like this is called a “variational crime”. Applying a quadrature rule of a not sufficiently high order results in an aliasing effect of the lower modes, attempting to compensate for the higher order modes omitted. Since a spectral element method arguably has a good accuracy these variational crimes should not be committed and it is therefore common practice to solve this particular integral using a quadrature of order $3N$. The concept and illustrative examples are given in Chapter 2.4 in [4]. The effect of aliasing is made clear in chapter 6. This is one of the time vs. accuracy questions one have to decide for each problem. instead of applying the GLL-quadrature ”designed” for the basis-functions the functions has to be evaluated in a new set of GLL-points with $3/2$ as many nodes. This is a costly process and should only be applied when absolutely necessary.

3.4.3 GORDON-HALL ALGORITHM

In order to work on complex geometries some elements require a certain deformation in order to be able to describe the entire domain. It is necessary to evaluate all the integrals surging from the weak formulation over a reference domain $\hat{\Omega} = [-1, 1]^d$ for

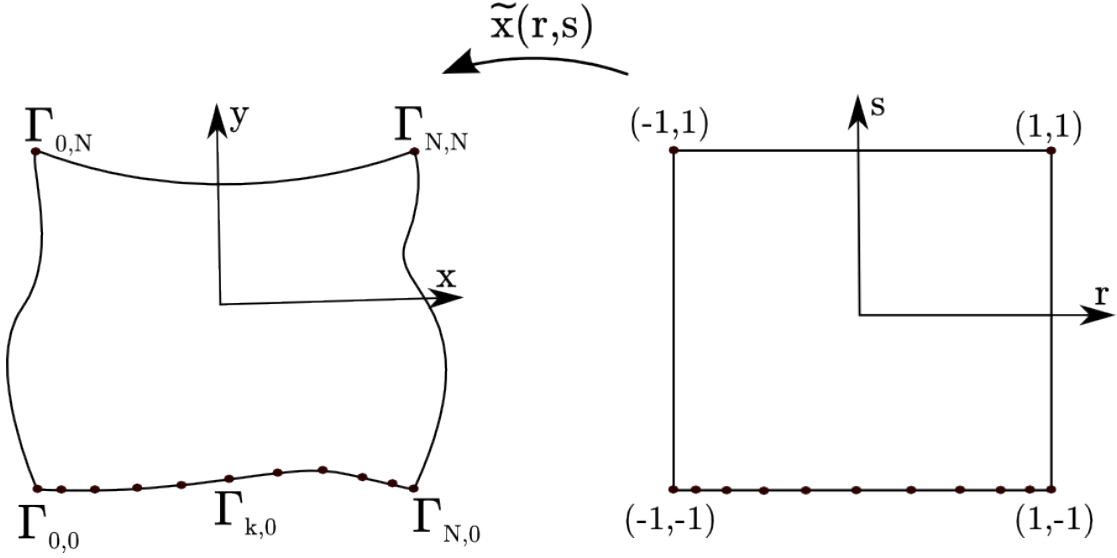


FIGURE 3.1: An illustration of how the Gordon-Hall algorithm creates a map from the deformed element to the reference element. The GLL-points are drawn along the edge $\Gamma_{k,0}$ on the deformed element which corresponds to $s = -1$ on the reference element.

sakes of efficiency and implementation purposes. The Gordon-Hall algorithm is a general method that creates an isometric map from an arbitrary simply connected domain to $\hat{\Omega}$. Let $\tilde{\mathbf{x}}$ be the mapping function from the reference domain to the physical domain given on the form

$$\tilde{\mathbf{x}} = \sum_i \sum_j \sum_k \mathbf{x}_{ijk} l_i(r) l_j(s) l_k(t). \quad (3.30)$$

l_i being the i th Lagrange polynomial. The full description of the algorithm with helpful figures can be found in [15] chapter 4. Without going to much into the mathematical foundation of this method a more intuitive and implementable presentation of the method will be provided in this chapter. For simplicity a two-dimensional domain will be considered here, and the 3D case will be an easy expansion of the algorithm presented here. Consider a deformed domain $\Omega \in \mathbb{R}^2$, with $\Gamma_{i,j}$ representing the discrete boundary coordinates. The four vertices can then be expressed as $\Gamma_{0,0}, \Gamma_{0,N}, \Gamma_{N,0}, \Gamma_{N,N}$. Let ϕ_0, ϕ_N be defined as

$$\phi_0(\xi) = \frac{1 - \xi}{2}, \quad \phi_N(\xi) = \frac{1 + \xi}{2}. \quad (3.31)$$

Let $\{\xi_0, \dots, \xi_N\}_{N+1} = \{-1, \dots, 1\}_{N+1}$ be the GLL-points corresponding to the Lagrange polynomial of order N . An important property for the functions in 3.31 is that $\phi_0(\xi_0) = \phi_N(\xi_N) = 1$ and $\phi_0(\xi_N) = \phi_N(\xi_0) = 0$.

The algorithm provides a stepwise routine depending on the complexity of the domain. The first step is to create a mapping to a polygon spanned from the vertices of Ω .

$$\begin{aligned}
\tilde{\mathbf{x}}_{i,j} = & \Gamma_{0,0}\phi_0(r_i)\phi_0(s_j) \\
& + \Gamma_{0,N}\phi_0(r_i)\phi_N(s_j) \\
& + \Gamma_{N,0}\phi_N(r_i)\phi_0(s_j) \\
& + \Gamma_{N,N}\phi_N(r_i)\phi_N(s_j)
\end{aligned} \tag{3.32}$$

If the edges are straight the algorithm ends here, but for curved edges a second step is performed adding the deformation of the edges.

$$\begin{aligned}
\tilde{\mathbf{x}}_{i,j} = & \tilde{\mathbf{x}}_{i,j} + (\Gamma_{i,0} - \tilde{\mathbf{x}}_{i,0})\phi_0(s_j) \\
& + (\Gamma_{i,N} - \tilde{\mathbf{x}}_{i,N})\phi_N(s_j) \\
& + (\Gamma_{0,j} - \tilde{\mathbf{x}}_{0,j})\phi_0(r_i) \\
& + (\Gamma_{N,j} - \tilde{\mathbf{x}}_{N,j})\phi_N(r_i)
\end{aligned} \tag{3.33}$$

In 3D the additional knowledge of the faces may be applied to create mappings from elements with deformed faces as a third step. The only difference when applying this algorithm in three dimensions is that you need to include ϕ for a third coordinate t_k and the number of vertices, and edges are 8 and 12 instead of 4 and 4.

3.5 TIME INTEGRATION FOR INCOMPRESSIBLE N-S

Rewrite this introduction

A non-linear set of equations requires a non-trivial solution method, and when the domain of the problem can be anything from a simple channel to a moving turbine there are many considerations that needs to be made. Although the equations have been known for over 200 years no one has been able to prove or disprove the well-posedness of the problem. Some of the most common algorithms will be discussed in the following subsections

3.5.1 OPERATOR-SPLITTING TECHNIQUES

In this chapter a_j, b_j will denote the coefficients for some explicit and implicit scheme. Let us consider a simplified transient problem

$$\frac{du}{dt} = f(u, t)u + g(t)u. \quad (3.34)$$

f is here a function of u and t , while g is only dependent of the time t . Let superscript denote the timestep, such that $g^n = g(n\Delta t)$ for some fixed timestep Δt . One step applying a k th order Backward Difference scheme (BDFk) yields

$$\sum_{j=0}^k \frac{b_j}{\Delta t} u^{n+1-j} = f^{n+1} u^{n+1} + g^{n+1} u^{n+1}. \quad (3.35)$$

Now notice that $f^{n+1} = f(u^{n+1}, t^{n+1})$ requires that u is known at time t^{n+1} which is not achievable at the current step. This term is therefore approximated by a k th order explicit scheme leading to

$$\sum_{j=0}^k \frac{b_j}{\Delta t} u^{n+1-j} = \sum_{j=0}^k a_j f^{n-j} u^{n-j} + g^{n+1} u^{n+1}. \quad (3.36)$$

Now the terms can be ordered such that only the implicit terms are present on the left hand side,

$$\left(\frac{b_0}{\Delta t} + g^{n+1}\right) u^{n+1} = - \sum_{j=1}^k \frac{b_j}{\Delta t} u^{n+1-j} + \sum_{j=0}^k a_j f^{n-j} u^{n-j}. \quad (3.37)$$

This way of solving 3.34 allows easy invertible terms to be solved implicitly while non-linear terms can be extrapolated. In the Navier-Stokes equation this strategy will be applied to split the non-linear term from the rest. In this thesis the schemes BDFk and EXTk for $k = 2, 3$ are applied, the coefficients can be found in [16].

3.5.2 OPERATOR INTEGRATING FACTOR SCHEMES (OIFS)

The operator-splitting method described in the previous chapter may lead to an unstable scheme, OIFS is a similar method but it offers a more stable scheme. The presentation

of the method is presented here in a computational fashion, for a full description and derivation of the method it is referred to Maday et al [17].

By considering the NS-equation in a general operational form

$$M \frac{d\mathbf{v}}{dt} + C\mathbf{v} = -A\mathbf{v} + Dp + Mf. \quad (3.38)$$

Now let $Q(t)$ be an operator such that $Q(t^{n+1}) = I$ and

$$\frac{dQ(t)M\mathbf{v}}{dt} = Q(t)M \frac{d\mathbf{v}}{dt} + \frac{d}{dt}(Q(t)M)\mathbf{v}, \quad (3.39)$$

$$= Q(t)M \frac{d\mathbf{v}}{dt} + Q(t)C\mathbf{v}. \quad (3.40)$$

This way 3.38 can be written as

$$\frac{dQ(t)M\mathbf{v}}{dt} = Q(t)(-A\mathbf{v} + Dp + Mf). \quad (3.41)$$

Evaluating this equation with a BDFk-scheme results in a system

$$\sum_{j=0}^k \beta_j Q(t^{n+1-j})M\mathbf{v}^{n+1-j} = \Delta t Q(t^{n+1})(-A\mathbf{v}^{n+1} + Dp^{n+1} + M\mathbf{f}^{n+1}). \quad (3.42)$$

Applying the fact that $Q(t^{n+1}) = I$ enables 3.43 to be written as

$$\beta_0 M\mathbf{v}^{n+1} + \sum_{j=1}^k \beta_j Q(t^{n+1-j})M\mathbf{v}^{n+1-j} = \Delta t(-A\mathbf{v}^{n+1} + Dp^{n+1} + M\mathbf{f}^{n+1}). \quad (3.43)$$

Notice how all the easily invertible operators are evaluated implicitly, while the convective non-linear term is hidden in the BDFk scheme. OIFS allows the terms in the sum to be calculated in a rather elegant fashion. First of all the auxiliary variable $\tilde{\mathbf{v}}_j$ is defined such that $Q(t^{n+1-j})M\mathbf{v}^{n+1-j} = M\tilde{\mathbf{v}}_j$ thus enabling the summation expression to be found by solving the initial value problem

$$\begin{aligned} M \frac{d\tilde{\mathbf{v}}_j}{ds} &= -C(\tilde{\mathbf{v}}_j(s))\tilde{\mathbf{v}}_j(s), & t^{n+1-j} \leq s \leq t^{n+1} \\ \tilde{\mathbf{v}}_j(t^{n+1-j}) &= \mathbf{v}(t^{n+1-j}). \end{aligned} \quad (3.44)$$

Notice how the integrational factor $Q(t)$ is never evaluated directly.

The final scheme applied for solving 3.38 when applying OIFS consists of one implicit

scheme for solving 3.43 and an explicit scheme for solving 3.44. When applied in this thesis the first scheme is an implicit BDFk-scheme while the second is an explicit 4th order Runge-Kutta scheme.

3.5.3 FRACTIONAL STEP

Fractional step is an algorithm that can be divided into four separate steps. For simplicity let us write the N-S equations as

$$\partial_t v = -Av + Dp - Cv. \quad (3.45)$$

Where ∂_t, A, D, C Denotes the time-derivate, linear, gradient and non-linear operator. A schematic overview of the method is stated below, where the equations on the right hand side are solved and the updated solution is stated on the left hand side.

$$\begin{aligned} v^* &\leftarrow \frac{dv}{dt} = Cv, \\ p^{n+1} &\leftarrow \Delta p = \nabla \cdot \frac{v^*}{\Delta t}, \\ v^{**} &\leftarrow \frac{dv^*}{dt} = Dp^{n+1}, \\ v^{n+1} &\leftarrow \frac{dv^{**}}{dt} = -Av^{**}. \end{aligned} \quad (3.46)$$

As earlier mentioned this method is convenient because it allows us to handle the different terms with different solution techniques. So since the first equation in 3.46 involves the non-linear skew-symmetric advection operator this equation is solved using a explicit Adam Bashford scheme. The second equation is the Poisson pressure equation which assures a divergence free velocity field. Note that $p \in L^2 \supset H^1$ hence the Poisson equation is somewhat different from the one normally studied in textbooks. Another difficulty is the treatment of the boundary conditions. Ideally the BC's should be determined by the velocity field v^{n+1} , but since this solution is yet to be calculated the intermediate velocity field v^* is used to impose the boundary conditions. With p^{n+1} known the third equation is simply an update of the velocity in order to impose the divergence free condition. Now the last equation is solved implicitly due to its nice symmetric structure. This results in a system equivalent to the Helmholtz problem which will be discussed in

detail in chapter 3. This can be easily shown by discretizing the equation,

$$\begin{aligned} (v^{n+1} - v^n)/\Delta t &= -Av^{n+1}, \\ Av^{n+1} + \frac{1}{\Delta t}v^{n+1} &= v^n. \end{aligned} \tag{3.47}$$

Knowing that A is the discrete Laplacian and v^n is a known variable this is similar to problem 3.12.

This method provides an efficient algorithm, but is known to produce errors of order $O(1)$. The problem is the pressure Poisson equation which is solved with incorrect boundary conditions. The first step can also be evaluated in an OIFS-way to gain stability, by rewriting the initial equation as described in the previous chapter. This allows you to solve the unstable advection operator with multiple substeps. Explicitly the first step in 3.46 would be solved by applying the discretization used in 3.43 with an empty right hand side since the rest of the terms are take care of in the next steps. The IVP 3.44 is then solved to obtain v^* .

CHAPTER 4

APPLICATION OF NEK5000

ta med flytteddiagram for l;snigen rydd opp i skjemaene! referer til en l;rerbok, beskriv vha numerisk
 Hver tydelig paa hva man itererer over. korte ned tekstene! There are many numerical solvers for turbulent flows available on the market. From large commercial softwares such as Fluent which runs as a black-box solver, to full open-source codes such as Nek5000 and openFOAM. The solvers can vary in the numerical method; Finite volume, Finite Differences, Finite Element Method, Spectral Element Method etc. The particular algorithm for resolving the Pressure-Velocity coupling, for instance Fractional Step, Poisson pressure and Uzawa. The type of simulation available also varies from solver to solver, whether they apply RANS, LES, DNS or something else. Although most solvers offer multiple of the settings listed above it is important to be aware of their strengths and weaknesses before choosing which one to use. This section will be devoted to the handling of Nek5000 in particular, and can serve as a brief introduction to the code.

4.1 NEK5000 BASICS

Nek5000 is a turbulent flow solver developed mainly by Paul Fischer and has through the past 20 years had several contributors. It is an open-source code applicable to many different types of flow and it has been put a lot of effort into the parallelization of the code, guaranteeing great speedup. All the parallelization is accessed through subroutines and functions, enabling the user to make advanced functions without having to deal

directly with the functions from the MPI library. With SEM as the numerical method applied it is possible to obtain very accurate results.

Nek provides a basic tool for generation of mesh. For more complex geometries this tool cannot compare with more visualized-based softwares such as ICEM from ANSYS which exports mesh to several numerical solvers such as Fluent and Nastran. It is therefore very useful to have an automatic way of converting a mesh created in ICEM to the format required by Nek5000.

So far Nek5000 has supported three automatic routines for generating curved edges; circles in 2-D geometries, spherical shell elements and a general 2nd degree interpolation. Further manipulation of the element edges is left to the user to define manually for each particular problem. One of the objectives of this thesis is to make Nek5000 more user-friendly and create automatic routines to handle complex geometry. Before the work regarding the mesh routines are further elaborated an overview of the file-structure will be presented.

4.2 EDITABLE FILES

In order to work with Nek there are some practical information that needs to be clarified. Nek is recompiled for every case and the user specify all the case specific information in the three files `{.rea,.usr,SIZE,}`. The `.usr` and `SIZE` file are compiled with the standard Nek library using `makenek` which creates the executable file `nek5000`, while the `.rea` file contains case-specific information read during initialization. The user guide [16] contains a tutorial which explains the necessary steps on how to get started with Nek. The next chapters will try to give some understanding on how the user is able to make the changes necessary for each case. Figure 4.1 illustrates how the files work together.

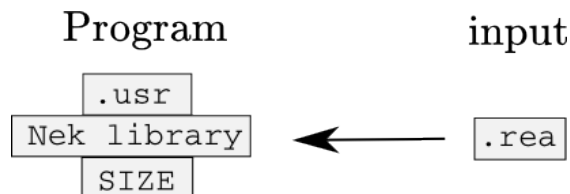


FIGURE 4.1: *Visualization of how the file structure in Nek5000 is built up.*

[h] Lines	Section Name	Specifications
103	PARAMETERS	All problem-specific variables
<i>K</i>	PASSIVE SCALAR DATA	Convective and diffusive constants for scalars
<i>K</i>	LOGICAL SWITCHES	Boolean variables defining the solution method
<i>E</i>	MESH DATA	All nodes and elements are specified here
<i>E</i>	CURVED SIDE DATA	All the curved sides are specified here
<i>E</i>	FLUID BC	BCtype for all elements and their faces
<i>E</i>	THERMAL BC	Thermal BCtype for all elements and their faces
<i>K</i>	PRESOLVE/RESTART	Filename(s) of an initialized solution
<i>K</i>	INITIAL CONDITIONS	possibilities to specify IC further
<i>K</i>	OUTPUT FIELD	information that will be written to file

TABLE 4.1: *An overview of the different sections in .rea. E is a predefined number depending on your problem which scales roughly as the number of elements, while $K \approx 1 - 25$ is user defined.*

4.2.1 SIZE

Since Nek is mostly based on Fortran77 all memory allocations are done statically and should be specified explicitly before runtime. Most of these variables are stated in **SIZE**. The size of the working arrays necessary to perform the calculations are mostly defined by the upper limits of elements, processors, scalars and of course the polynomial degree of the local Lagrange functions. These variables defines the sizes of almost all the arrays used in the program so it is important to define these variables as accurately as possible in order to optimize memory usage. The **SIZE** file can be considered as the necessary base for Nek5000.

4.2.2 .REA

In **.rea** all the problem specific parameters are given. While the content in **SIZE** is an absolute necessity to even compile the program the **.rea** file contains variables that are not used until the initialization of the case. The structure of the file is given in table 4.1. Of the 103 variables specified in the beginning of the file there are roughly 50 of them that are used. Note that apart from the mesh information the **.rea** file restricts itself to single variables and boolean flags while the **.usr** needs to be applied for more advanced implementations.

4.2.3 .USR

This file contains a series of standard routines open for modification by the user. In addition the user is free to specify new routines if needed. A description of these routines are given in the Nek5000 User manual [16]. A list of those frequently used for this thesis are described below,

- **userbc** - Define the boundary conditions on the inflow-boundary.
- **uservp** - Impose the eddy viscosity when applying LES.
- **userchk** - Read inflow-data, and specify the output.
- **usrdat2** - Project the geometry onto a deformed general surface. The details of how this routine is used will be specified further in chapter 5.
- **usrdat3** - Defines the interpolation algorithm that is applied to the inflow-data.

In addition to these routines all user-defined functions are specified in this file. The LES implementation in Nek is based on several subroutines specified in addition to those stated above. A list of some of the variables and functions applied for the implementations in this thesis are stated in Appendix B. The **.usr** file can be considered as the surface of Nek5000, easily accessible for the user.

4.3 STEPS IN THE SOLVER

To best understand the work flow of Nek the subroutine **nek_advance** in **drive1.f** should be examined. This routine includes all the steps in one iteration of the solver. The routine is adapted so that it is functional for a number of different problems and user defined settings. The most important boolean switches used in this routine are described in the list below

- **ifsplit** - whether the $\mathbb{P}_N - \mathbb{P}_N$ or the $\mathbb{P}_N - \mathbb{P}_{N-2}$ is to be used.
- **iftrans** - transient or steady flow.
- **ifheat** - solving for heat.

- ifnav - natural convection for the scalar fields (Boolean array)
- param(103) - activation of filtering.

further the routines of interests are `fluid` and `heat`, the solvers for N-S equations and passive scalars respectively. Both subroutines are found in `drive2.f`. The understanding of these solvers are best achieved by studying equation 2.1 and 2.15. For the settings chosen in this thesis a fractional step procedure as described in chapter 3.5.3 is applied.

Talk about how the preconditioners work in Nek?

4.4 NEK5000 FOR COMPLEX GEOMETRIES

For complex curved geometries such as bent cylinders, spheres, ellipsoids etc. the user has to be able to express these surfaces analytically and write a routine in `usrdat2` that projects the points of interest onto the surface. Even for a simple shape such as a sphere some implementation has to be done and it demands that the user has knowledge to Fortran77 and the structure of Nek5000.

The necessary implementation consists of two steps

1. determine the faces that belongs to the deformed surface
2. project the predefined GLL-points onto the deformed surface

This can be done without too much work for shapes with a known analytical expression such as a cylinder or a sphere, but for some general CAD geometry it is no way to perform this projection routine.

Many turbulent solvers does not support curved elements simply because the complex geometries are resolved with a sufficiently high resolution and it is of no interest to approximate them any better. However for a spectral element solver it is necessary to address this problem since the initial grid is a lot coarser compared to equivalent settings in other solvers.

Comment on the MOAB possibility??

CHAPTER 5

IMPLEMENTATION

Some introduction that motivates the work done in this thesis

henvis til NS, nevner eksplisitt de ligningene som brukes

Ta med et turbulensbilde, og bilde av plumen.

5.1 CASE 1: GAS DISPERSION IN A SIMPLIFIED URBAN AREA

The scenario investigated in this work is dispersion of a neutral gas in a rectangular tunnel with four cubic blocks placed as obstacles. The blocks have sides $h = 0.109\text{m}$ and represent a set of buildings forming a street canyon. The gas is released from a circular source on ground level and is translated by the wind field through the canyon, see figure 5.1. In this figure h have been used as the length scale. The dotted lines indicates the positions where data is collected.

Scaling the domain with the size of the boundary layer $H = 1\text{m}$ restricts it to the box $0.0 \leq x/H \leq 4.96, -1.75 \leq y/H \leq 1.75, 0 \leq z/H \leq 1.5$. The four cubic boxes are centered around $(1.4315, 0)$ with a distance h between each box. The source is placed with its center in $(0.396, 0)$ and radius $r = 0.0515$. The grid used for the computations consists of 4425 elements and with a polynomial degree of 12 the total number of nodes $N \approx 7,6\text{mill}$.

The simulations are performed using Large Eddy Simulation (LES) with the dynamic Smagorinsky-Lilly subgrid-scale model. The release of gas will result in a plume that is

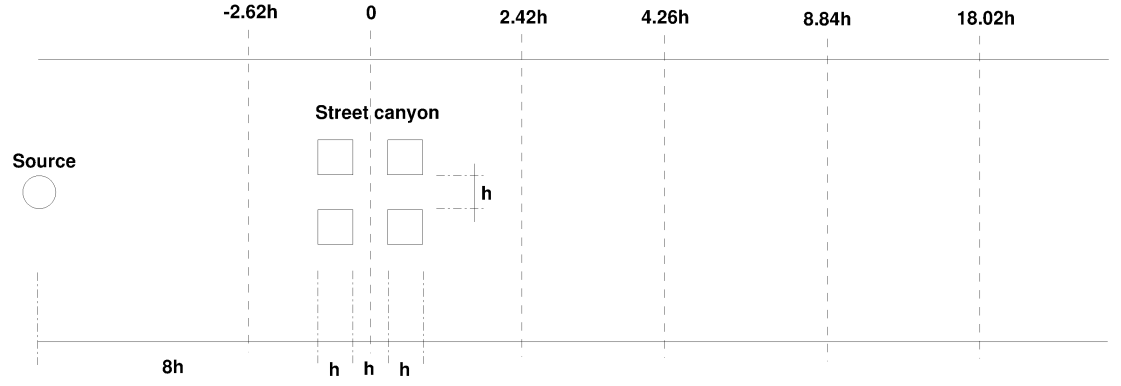


FIGURE 5.1: *Schematic overview of the domain from above. The data is collected along the dotted lines.*

Variable	value	unit	commentary
H	1	m	length scale of the domain
h	0.109	m	the sides of the cubic boxes
Q	50	dm ³ /min	gas release from source
U_{ref}^*	≈ 1.08	m/s	reference value of U

TABLE 5.1: *Essential variables, *this value is calculated as a time average of the velocity in x -direction at a point far away from the floor and walls and will therefore vary a small amount from case to case.*

advected with the wind field. The size and shape of the plume at the indicated positions in figure 5.1 are compared with experimental data and simulations performed in Fluent and CDP. The wind-field in the tunnel is created by an inflow condition that is defined from previous simulations in CDP [18]. For clarification some of the variables repeatedly mentioned throughout this thesis will be stated explicitly in table 5.1.

The inflow conditions had to be extrapolated onto the domain at each time step. The velocity field on the inflow was generated in CDP, in order to create a boundary layer similar to the one found in the wind-tunnel. The inflow velocity was written to file every 0.0013s for a total of 28 seconds. An interpolation algorithm had to be implemented in order to adjust the inflow-data to each mesh.

5.2 CASE 2: DRAG AND LIFT ON A CYLINDER

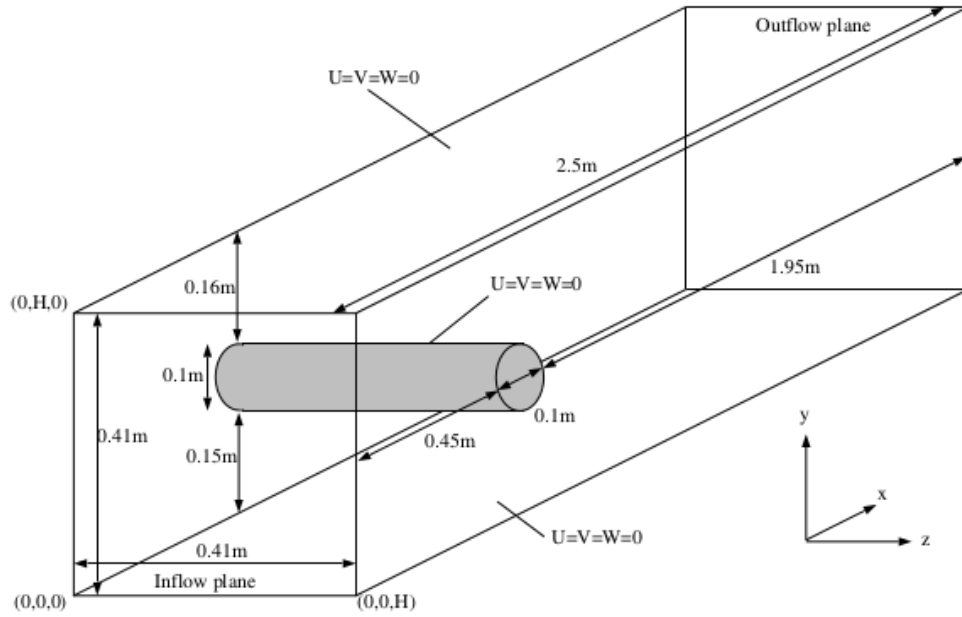
A standard benchmark case for flow solvers is presented in [19]. The case is to calculate the drag and lift coefficients on a cylinder in a rectangular channel. The setup for the

domain and boundary conditions are given in figure 5.2. The constants applied in the description of the geometry and the coefficient scales are listed in table 5.2. The flow is

Constant	Value	Property
H	0.41m	Width and height for the channel
D	0.1m	Diameter of the cylinder and length scale
U	0.2m/s	Velocity scale
ν	$10^{-3}\text{m}^2/\text{s}$	Kinematic viscosity of the fluid
Re	20	Reynolds number

TABLE 5.2: *Constants for case 2*

laminar with Reynolds number $Re = 20$ so all the challenges arising when dealing with turbulent flow does not come to play in this case. The case will yield a steady state solution at which point the coefficients are calculated and compared with the reference solutions. The spectral element method applied in Nek is known to be an accurate

FIGURE 5.2: *Computational domain and boundary conditions.*

solution method and is expected to provide a very good result in a test-case like this.

The drag and lift forces on an surface S are given as

$$F_D = \int_S (\rho\nu \frac{\partial v_t}{\partial n} n_y - p n_x) dS \quad , \quad F_L = - \int_S (\rho\nu \frac{\partial v_t}{\partial n} n_x + p n_y) dS. \quad (5.1)$$

Nevne hvordan integralet løses i Nek.

Dersom det var noen problemer med rammeverket, nevnt det kort!

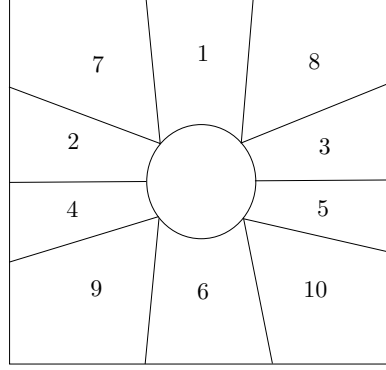
konsisten eq. bruk!

The coefficients corresponding to these forces known as the drag and lift coefficients are given by the formulas

$$c_D = \frac{2F_D}{\rho U^2 D H} \quad , \quad c_L = \frac{2F_L}{\rho U^2 D H}. \quad (5.2)$$

Nek provides functions for calculating lift and drag on any user-specified object. The function is called `drag_calc(scale)`, with the input parameter defined by the user, for this case `scale = 2/(\rho U^2 D H)`. Apart from this the function `set_obj()` has to be modified in order to create an object which consists of all the faces on the cylinder. Let x, y be points in the computational domain, x_c, y_c be the coordinates to the center line in the cylinder and $0 < tol \ll 1$ be some user defined tolerance. The faces that belong to the cylinder can then be found by looping over all elements and their faces evaluating $\epsilon = \sqrt{(x - x_c)^2 + (y - y_c)^2}$. If $\epsilon < tol$ for an entire face then this face is known to belong to the cylinder and is added to the object. Nek also allows the user to specify multiple objects assigning the faces of interest to object 1, object 2 etc. The geometry and mesh for this case was generated in ICEM, and the total number of elements are 2070. For the final calculation polynomial degree $P = 11$ was applied leading to a total of $N = 2755170$.

Initially this case was solved using a second degree polynomial to describe the circle segments corresponding to each element. The mesh around the cylinder is illustrated in figure 5.3. Note that these elements was split in three, in order to obtain a finer mesh in the region of interest. Of the elements numbered in 5.3 only the first six contains edges on the cylinder. Hence the second order polynomials describing the curved edges describe approximately $\Theta = 360^\circ / (6 \cdot 3) = 20^\circ$ of the complete circle. There is no computational reason for not incorporating an order $n = P$ approximation of the circle segments eliminating this approximation error. An algorithm was therefore implemented as an extension to the currently available features regarding curvature on edges. The full description of the algorithm is presented in chapter 5.3. The importance of the error

FIGURE 5.3: *Initial mesh around cylinder.*

resulting from the second degree approximation of the circle segments are presented in chapter 6.

5.3 ADVANCES IN THE MESH-GENERATION ROUTINE

The routine `xyzarc()`:

The Gordon Hall algorithm was already implemented as a function in Nek, with the GLL-points, polynomial degree and some initial coordinates to the element. The algorithm creates a distribution of the internal GLL-points in each element. If the element consists of linear edges the only necessary input are the vertices, but by specifying the points on edges and faces the algorithm creates a logical distribution of the internal GLL-points to a deformed element.

The curved edge is specified in the .rea file and the routine `genxyz()` processes the input of each edge. By specifying the radius and the circle center `genxyz()` calls the routine `xyzarc()` which performs the following algorithm;

a, b will be the two end nodes of the edge c will be the mid node, s will be the arc length, θ will be the full angle of the circle sector, cc is the center coordinates, g will be the vector containing the GLL-points in $[-1, 1]$ and r will be the radius.

```

l = a-b                                ! vector between the corner nodes
c = (a+b)/2                            ! midpoint location
h = c-cc                                ! height of the framed triangle
θ = arctan(abs(l)/2*abs(h))             ! half the angle of the circle sector
s = r*θ                                 ! arc length
g = g*θ                                 ! angles to the GLL-points on the circle-sector
!----- Finding the intersecting points -----!
!---- x on the line l, and extend x-cc to the arc ----!
do k=1,lx1                             ! for the number of nodes in one direction
  α = h*tan(g[k])                       ! offset from the midpoint on l
  x = c-α*l/abs(l)                      ! actual coordinate on l
  m = x-cc                              ! hypotenuse of the imposed triangle
  edge(k) = cc+r*m/abs(m)               ! final coordinate on the arc
enddo

```

These lines creates the wanted edge curved as a circle sector corresponding to the radius and circle center given. The remaining operation is to call the Gordon Hall algorithm and create the internal GLL-points defined by the edges provided. The figure 5.4 illustrates the geometry on which the algorithm is performed. Notice that the algorithm assumes that the center c is somewhere on the plane defined as all the points with equal distance to both a and b .

- initial script
- changes and modifications
- performance testing
- pitfalls

5.4 ADDITIONAL PROJECTION ALGORITHM

The routine `xyzarc()` enables the user to represent circular edges with the same order as the Lagrange polynomials of each element. For more complex geometry such as actual terrain and other surfaces without any analytic expression the large element sizes makes the geometrical representation difficult. There is however no reason why the GLL-points on a face can be projected onto a non-analytical surface. Although there are no support for automatically reading additional information for distributing the GLL-points exactly

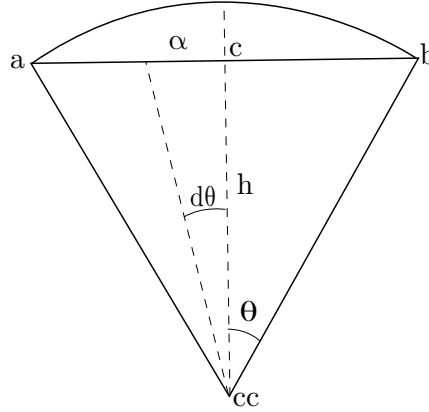


FIGURE 5.4: *A sketch of the curved edge and the variables necessary to calculate the projection*

on a given surface a routine was made which demands a minimum of manual work. The method can be summed up in these simple steps

1. Create initial Mesh and convert to .rea applying nmshconvert.
2. Create refined surface mesh on the non-regular surface.
3. Enable projection by setting `param(33) = 1`.
4. Choose number of interpolation points by modifying `param(34) = (1,2,3)`

In addition to the standard Nek library the file `surfpro.f` needs to be added to the `/trunk/nek/` folder along with all the other scripts applied by Nek.

The algorithm relies on two external files generated by the modified `nmshconvert` script. `surf.i` contains all the coordinated to the points on the refined surface. `bdry.i` contains the element, and face number to all the faces to be projected onto the surface. When generated they automatically alter the local `SIZE` file to contain some key variables applied in the algorithm in `surfpro.f`. The algorithm is best explained through a simple

box with a non-regular floor. As an initial test-problem the hill of Ekeberg was applied. Before describing the algorithm let $E_{tot} = n_x n_y n_z$ be the total number of elements, N is the polynomial degree and let us for simplicity assume that $n_x = n_y = n_z$ such that $E = E_{tot}^{2/3}$ is the number of elements containing a face on the non-regular surface. The number of points on the refined surface N_s should be somewhat larger than EN^2 in order to describe the surface for all the GLL-points that belong to the boundary. The pseudo code for the algorithm is listed below.

```

do e,f in bdry.i
  wrk = create_working_surface(e,f)
  do i in GLL-nodes
    interp = init_interpolation_array()
    do j in wrk
      update_int_array(interp,wrk(j))
    enddo
    set_new_point(interp,wrk,i,e,f)
  enddo
  fix_GLL()
enddo
fix_geom()

```

In order to understand the algorithm a short description of the auxiliary functions is given in the list below

- `create_working_surface(e,f)` – Loops through all the nodes in `surf.i` and adds the surface-coordinates within a certain radius to the center GLL-node to the array `wrk`. This saves time in the search for interpolation points for each GLL-node.
- `init_interpolation_array()` – initializing the array containing the closest points on the surface for the current GLL-node.
- `update_int_array(interp,wrk(j))` – compares the current surface point to the already existing interpolation points and adds it to the list if it is found to be closer to the initial GLL-node.
- `set_new_point(interp,wrk,i,e,f)` – updating the new GLL-point determined by the surface points in `interp`.

- `fix_GLL()` – There is a risk after distributing the GLL-points on the surface that some of the internal GLL-points falls outside the element. `fix_GLL()` distributes all internal GLL-points correctly between the newly projected face and the opposite.
- `fix_geom()` – An already existing Nek routine which redistributes the GLL-points to assure that the distance between them on the new surface are correct.

Although this routine is only called once, and therefore will not contribute significantly to the total runtime of the program it is desirable to have a fast algorithm. Another analysis important to be made is the amount of extra storage space needed for this algorithm. By analysing the pseudo code the time of the algorithm should be of order $O(EN^2(E+N^2))$ and the amount of additional storage space will be of order $O(EN^2+E+N^2) = O(EN^2)$.

The routine attempts to be as automatic as possible and the only implementation necessary is a call from `usrdat2` with 3 input variables.

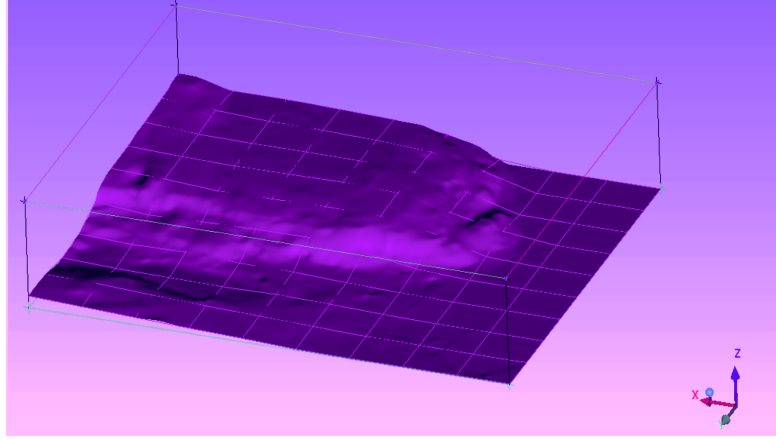
Now an illustrative example of how this method is applied by the user. Say you have a project called "myFlow", and the mesh and surface mesh created in ICEM are named `mesh_myFlow` and `surfmesh_myFlow`. The following commands are then executed

```
>> /nmshconvert --mesh mesh_myFlow
      --reafile init.rea --outfile myFlow.rea
      --tol 1e3 --temperature True --curvetype A

>> ./nmshconvert --mesh surfmesh_myFlow
      --mesh_format surface
```

5.4.1 APPLICATION ON THE HILL OF EKEBERG

As an initial test of the algorithm a mesh was created based on the hill of Ekeberg. The surface was loaded as a `.tin` file in ICEM and a simple box was created with the described terrain as floor. A simple sketch of the domain with the initial mesh in given in figure 5.5. This geometry was chosen because it resembles a typical problem with spectral elements. Since the initial element-mesh is relatively coarse it does not capture all the details in the geometry and the GLL-nodes distributed on the faces corresponding to the unstructured surface will be misplaced. It is however no theoretical problem to reconstruct the surface with with a polynomial of order P . With the routine described

FIGURE 5.5: *The initial mesh of the hill*

in chapter 5.4 The surface was approximated accurately by higher order polynomials. The algorithm restricts itself to relatively smooth surfaces.

5.5 SPATIAL AVERAGING ROUTINE

A dynamic Smagorinsky model has previously been implemented in Nek5000 for flow in a channel. The method as described in chapter 2 depends on an averaging routine to calculate the dynamic Smagorinsky constant. The implementation in Nek applies an average routine in the plane, assuming that the Smagorinsky constant is the same for all points with equal distance to the walls of the channel. This is a rather specific averaging routine only applicable to channel flows.

When applying dynamical Smagorinsky to Case 1 a new spatial mean routine had to be applied. It was first attempted to average only in time, but this proved not to be sufficient. It was therefore implemented a simple routine for taking the average within each element, let c_{num}^e, c_{den}^e denote the numerator and the denominator in equation 2.28. The means are then calculated as

$$c_{num}^e = \frac{1}{V} \int_{\Omega_e} c_{num}^e d\Omega = \frac{1}{V} \sum_{i=1}^{N^3} \rho_{i,e} c_{num,i}^e. \quad (5.3)$$

And similarly for c_{den}^e . The coefficients $\rho_{i,e}$ are found in the array `BM1(1x1,1y1,1z1)` in the file `MASS`.

CHAPTER 6

RESULTS

6.1 DRAG AND LIFT ON A CYLINDER

The effect of the implemented algorithm in Nek5000 explained in Chapter 5 is illustrated by solving a laminar flow test problem. The solution is compared with previously benchmark computations performed by a number of contributors [19]. The test problem considered is steady flow with $Re=20$ in a rectangular channel past a cylinder. The drag and lift coefficients on the cylinder are calculated and compared to a pair of reference values. The results can be found in table 6.1. As the results clearly show the treatment of the geometry is crucial, both coefficients are computed with significantly better accuracy. The polynomial degree used in the calculations with Nek5000 was chosen as $p = 11$ in all directions. The explicit number of cells is therefore $2070 \cdot 11^3 = 2755170$, approximately 88% of the number of cells used for the benchmark simulations. Compared with the results from the other softwares applied in [19] Nek5000 performs just as well or better in most cases. It should be mentioned that the division of the grid is

# of Cells	Software	c_D	c_L	% Err c_D	% Err c_L
2070	Nek5000 (mid)	6.18349	0.008939	0.030	4.19
2070	Nek5000 (arc)	6.18498	0.009413	0.006	0.13
3145728	CFX	6.18287	0.009387	0.04	0.15
3145728	OF	6.18931	0.00973	0.06	3.5
3145728	FEATFLOW	6.18465	0.009397	0.01	0.05

TABLE 6.1: *Results for the drag and lift coefficients with reference values $c_D = 6.18533$ and $c_L = 0.009401$.*

done differently for Nek5000 so the comparison is not as direct as it may seem from the table.

add more info about the mesh for this case??

6.1.1 PARAMETER ADJUSTMENTS IN NEK5000

As discussed in chapter 4 there are many adjustments available in Nek. In order to enlighten the actual effect on the results, several different settings have been investigated and the results are presented. All simulations are performed as transient flows and the data collected in table 6.2 are gathered when the flow has reached a steady state solution.

Perform the different tests and show the difference.

#	Settings			% Error		Data		
	Aliasing	IOFS	Filter	c_D	c_L	DT	CFL	T/Tstep
1	No	No	No	1.0	2.3	1e-04	2.03	2.1e-02

TABLE 6.2: *Performance data for different settings in Nek5000*

6.2 GAS DISPERSION IN A SIMPLIFIED URBAN AREA

This case is a part of a larger project designed to evaluate different solvers ability to perform simulations of gas dispersion. An initial attempt to simulate the dispersion of gas was performed in Nek5000 with fairly coarse element sizes, but without any subgrid-scale model. Only applying filtering on the last 3 nodes made the solution sufficiently stable and the results can be seen in figures 6.1 and 6.1. Discuss the plot...

redo these simulation in case they were started to early.

6.2.1 DYNAMIC SMAGORINSKY

Applying a subgrid scale model to the problem should imply a more accurate result since the grid is not sufficiently fine to resolve the finest eddies.

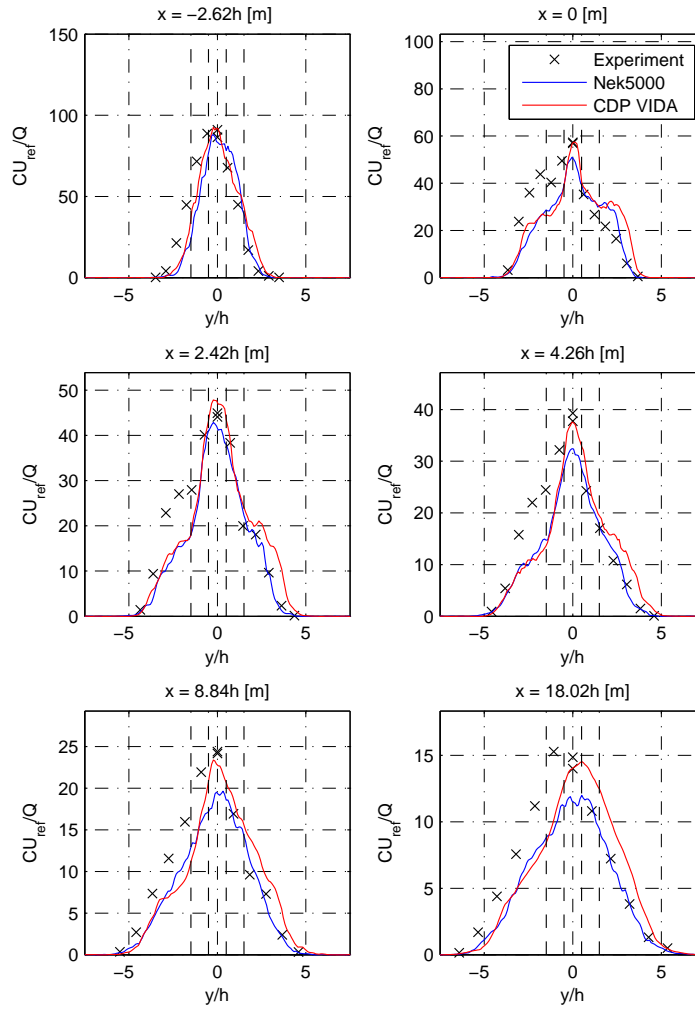


FIGURE 6.1: Time-averaged concentration with a sample time of 18.00 s at $z/H = 0.025$ plotted horizontally and scaled with the free-stream velocity and emission rate. Compared against wind tunnel data. Two dashed lines on either side of the centerline represent the canyon.

6.3 DISCUSSION AND CONCLUSION

How did Nek perform overall, user-friendly ?,correctness,speed etc.

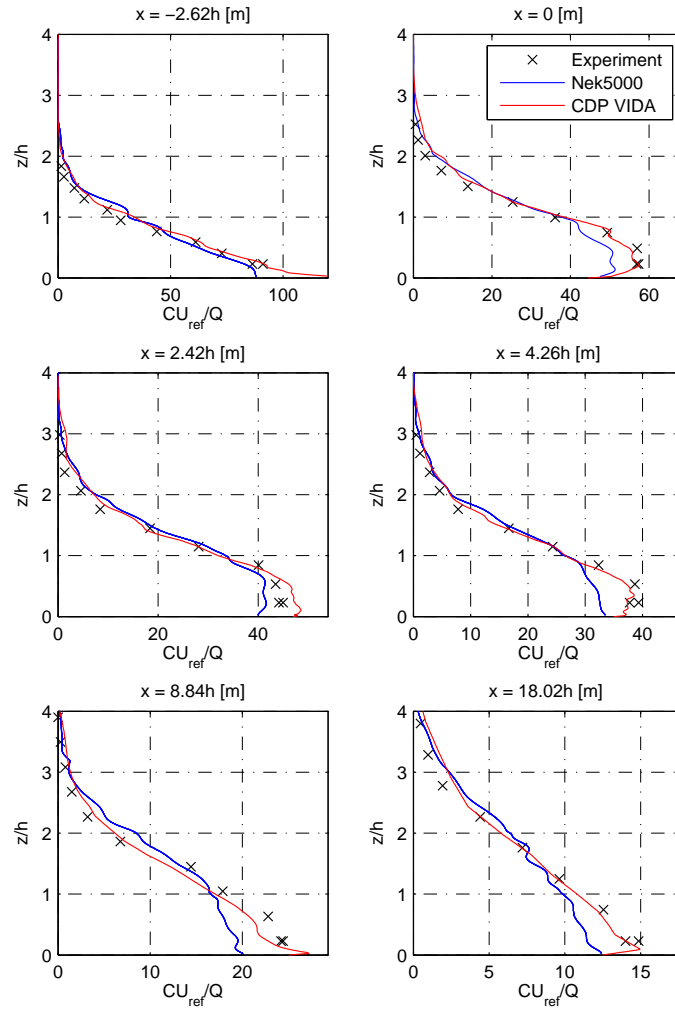


FIGURE 6.2: Time-averaged concentration with a sample time of 18.00 s at $y = 0$ plotted vertically and scaled with the free-stream velocity and emission rate. Compared against wind tunnel data. Two dashed lines on either side of the centerline represent the canyon.

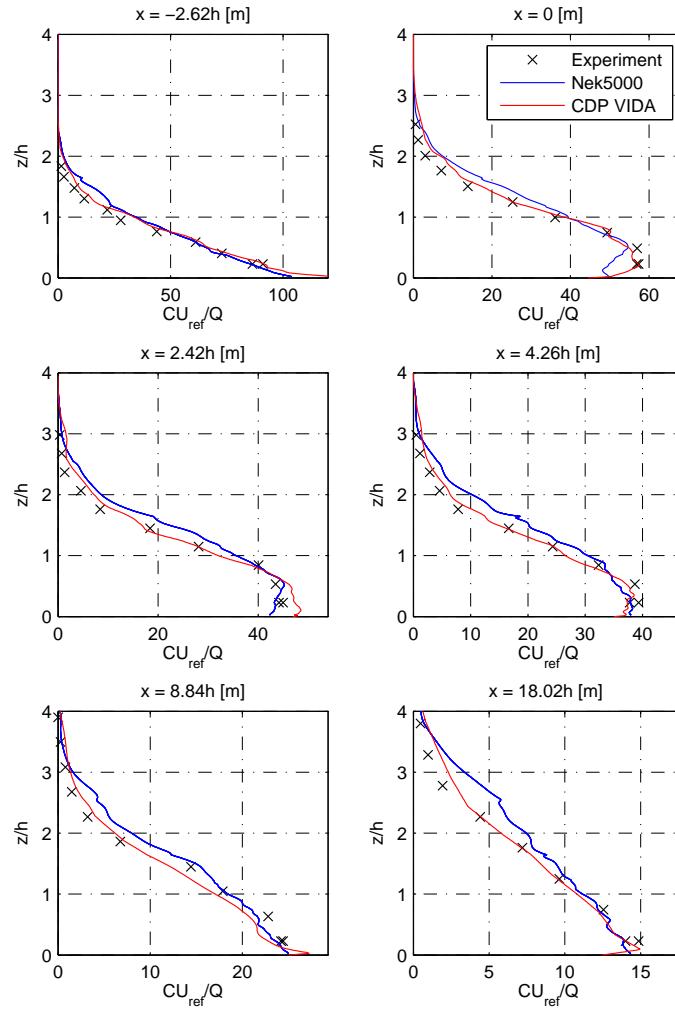


FIGURE 6.3: Time-averaged concentration with a sample time of 22.00 s at $y = 0$ plotted vertically and scaled with the free-stream velocity and emission rate. Compared against wind tunnel data. Two dashed lines on either side of the centerline represent the canyon.

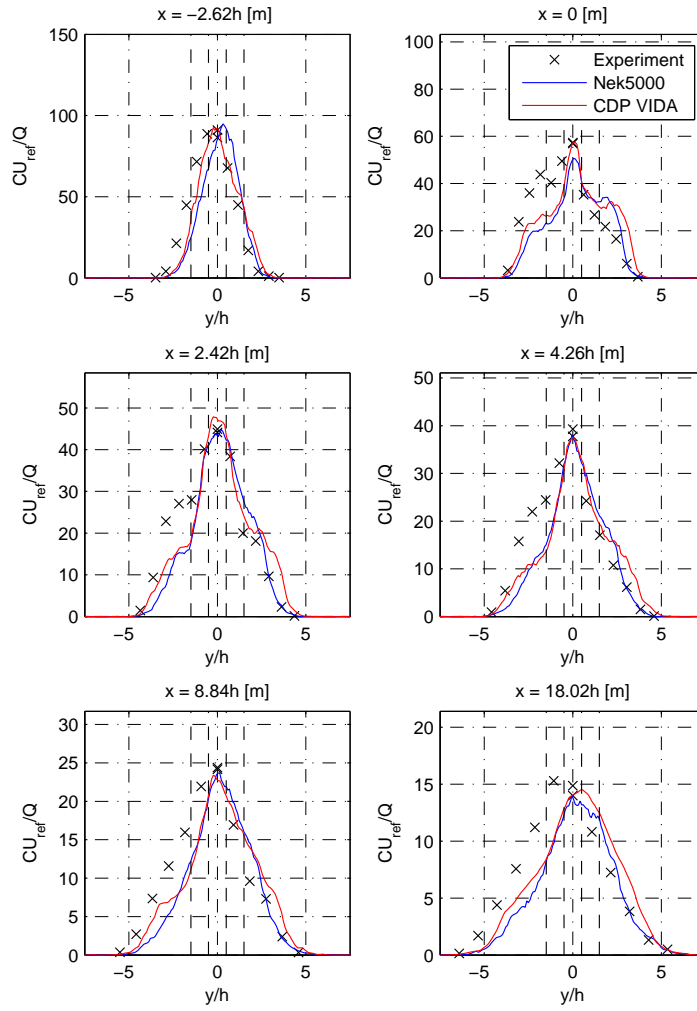


FIGURE 6.4: Time-averaged concentration with a sample time of 22.00 s at $y = 0$ plotted vertically and scaled with the free-stream velocity and emission rate. Compared against wind tunnel data. Two dashed lines on either side of the centerline represent the canyon.

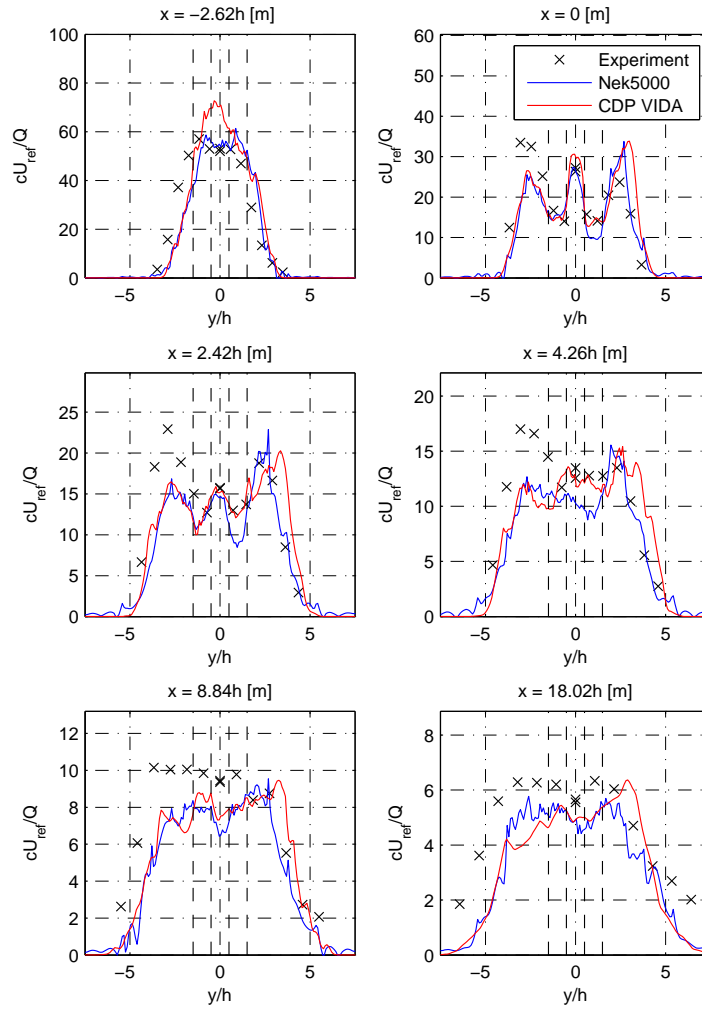


FIGURE 6.5: Time-averaged concentration with a sample time of 22.00 s at $y = 0$ plotted vertically and scaled with the free-stream velocity and emission rate. Compared against wind tunnel data. Two dashed lines on either side of the centerline represent the canyon.

APPENDIX A

FUNDAMENTAL BASICS OF NUMERICAL ANALYSIS

A.1 GLL-QUADRATURE

A.2 ESSENTIAL POLYNOMIALS

1. Legendre polynomials
2. Lagrange basis

A.3 PRELIMINARY CONCEPTS

1. coersiveness
2. Bounded

A.4 LAX-MILGRAM THEOREM

Stating the theroem without proof

APPENDIX B

VARIABLES AND FUNCTIONS IN

NEK5000

B.1 VARIABLES

The Nek manual provides information on many of the variables given in the .rea and SIZE file. It is however no list of useful variables defined in other files. Below is a list of some of the variables that have been frequently used in .usr subroutines which initially are defined outside of both SIZE, .rea and .usr.

B.2 FUNCTIONS

B.2.1 STANDARD CALCULATIONS FOUND IN MATH.F OR NAVIER1.F

nekasgn(ix,iy,iz,ie) Assigns the coordinates of node (ix,iy,iz) in element ie to the common variables x,y,z

facind(kx1,kx2,ky1,ky2,kz1,kz2,nx1,ny1,nz1,f) Assigns the index limits of a face f with nx1,ny1,nz1 points in each spatial direction.

zwgll(zg,wg,nx1) Get the nx1 GLL-points and weights to zg and wg.

cadd(zg,c,nx1) Adding a constant c to a vector zg of length nx1.

AVG	
uavg(ax1,ay1,az1,lelt)	Averaged values of u, similar for v,w,p
urms(ax1,ay1,az1,lelt)	Variance of u, similar for v,w,p
vwms(ax1,ay1,az1,lelt)	Covariance of vw
tavg(ax1,ay1,az1,lelt,ldimt)	Averaged values of t and all passive scalars
GEOM	
xm1(lx1,ly1,lz1,lelt)	X-coordinates for the velocity mesh
xm2(lx2,ly2,lz2,lelv)	X-coordinates for the pressure mesh
unx(lx1,lz1,6,lelt)	Surface normals
INPUT	
cbc(6,lelt,0:ldimt1)	Boundary conditions of each face
ccurve(12,lelt)	Curved side character
curve(12,6,lelt)	Curved side information
PARALLEL	
lglel(lelt)	Mapping from local to global element index
gllel(lelg)	Mapping from global to local element index
SOLN	
vx(lx1,ly1,lz1,lelv)	X-velocity
t(lx1,ly1,lz1,lelv,ldimt)	Temperature and passive scalars
vtrans(lx1,ly1,lz1,lelt,ldimt1)	Diffusive constant to additional scalars
vdiffl(lx1,ly1,lz1,lelt,ldimt1)	Convective constants to additional scalars
TSTEP	
istep	Current iteration step
iostep	Output step frequency
time	Current time
tstep	Current timestep
dt	Timestep
dtlag(10)	The preevious 10 timesteps
bd(10)	Max 10 backward difference coeffs
ab(10)	Max 10 extrapolation coeffs (Adam-Bashforth)
WZ	
zgm1(lx1,3)	GLL points for x,y and z directions
WZF	
zgl(lx1)	Gauss lobatto points
wgl(lx1)	Gauss lobatto weights
OTHER	
x,y,z	Local coordinates assigned by nekasgn()
ux,uy,uz	Local velocities assigned by nekasgn()
temp	Local temperature assigned by nekasgn()
nio	Processor node number
ndim	Number of dimensions
nelv	Number of elements for velocity mesh
nelt	Number of elements for the t-mesh
pm1 (lx1,ly1,lz1,lelv)	Pressure mapped to mesh 1

TABLE B.1: *useful variables in Nek, the bold capital sections denote the seperate files in /trunk/nek/.*

cmult(zg,c,nx1) Multiplying every element of vector zg of length nx1 with c.

chsign(wrk,nx1) change the sign of every element in vector wrk of length nx1.

cfill(zg,c,nx1) Fill vector zg of length nx1 with the constant c.

rzero(zg,nx1) Fill vector zg of length nx1 with zeroes.

rcopy(zg,zg2,nx1) copy all elements from vector zg2 to vector zg, both of length nx1.

B.2.2 FUNCTIONS REGARDING MESH AND DISTRIBUTION OF GLL-POINTS

gh_face_extend(x,zg,n,type,e,v) The Gordon hall algorithm described in chapter 3, the type variable denotes whether the algorithm should use vertices, edges or faces to distribute the inner GLL-points.

xyzlin(xl,yl,zl,nxl,nxl,nxl,e,ifaxl) Generate bi- or trilinear mesh.

fix_geom() Routine for re distributing the gll-points correctly on the updated geometry.

B.2.3 ADDITIONAL AUXILIARY FUNCTIONS IMPLEMENTED FOR THIS THESIS

fix_gll(e,f) Redistribute the gll-points between the given face and the opposite to make sure that all points lie within the element.

getface(kx1,kx2,ky1,ky2,kz1,kz2,wrk,n,e) assigning the values of the face in element e corresponding to the index limits kx1,kx2... to the array wrk of size n*n*3.

getsurfnorm(sn,ix,iy,iz,f,ie) Providing the surface normal sn at point ix,iy,iz of element ie and face f

calcerror(error,lambda,sn,wrk,radius) calculate the distance from the initial gll-point to a given point on the surface.

interp_up(iinterp,rinterp,error,k) Update interpolation points

set_new_pt(iinterp,rinterp,ix,iy,iz,e) defining the position of the new gll-point on the surface

getlimits(k,n,kx1,kx2,ky1,ky2,kz1,kz2) Get the index limits kx1,kx2... corresponding to edge k with n gll-points.

setcoords(xq,yq,zq,xedge,yedge,zedge,nxl,k) copy the updated edges xedge to the initial edges xq

getcoords(xq,yq,zq,xedge,yedge,zedge,nxl,k) copy the node information from the initial edge xq to xedge

BIBLIOGRAPHY

- [1] Frank M. White. *Viscous fluid flow*. McGraw-Hill series in mechanical engineering. McGraw-Hill, New York, 1991. ISBN 0-07-069712-4. URL <http://opac.inria.fr/record=b1096847>.
- [2] D. K. Lilly. The representation of small scale turbulence in numerical simulation experiments. In *IBM Scientific Computing Symposium on environmental sciences*, pages 195–210, Yorktown heights, 1967.
- [3] M. Germano, U. Piomelli, P. Moin, and W. H. Cabot. A dynamic subgrid-scale eddy viscosity model. *Phys. Fluids A*, 3:1760–1765, 1991.
- [4] George Karniadakis and Spencer Sherwin. *Spectral/hp element methods for computational fluid dynamics*. Oxford University Press, 2013.
- [5] Alfio Quarteroni. *Numerical Models for Differential Problems, 2. edition*. Springer, 2014.
- [6] J. l. Guermond. Nonstandard nonconforming approximation of the stokes problem, i: Periodic boundary conditions .
- [7] C. Canuto, M.Y Hussaini, and A. Quarteroni et al. *Spectral methods : evolution to complex geometries and applications to fluid dynamics*. Scientific computation. Springer, Berlin, 2007. ISBN 978-3-540-30727-3. URL <http://opac.inria.fr/record=b1127560>.
- [8] Endre Süli and David F. Mayers. *An Introduction to Numerical Analysis*. Cambridge, 2006.
- [9] Yvon Maday and Anthony T Patera. Spectral element methods for the incompressible navier-stokes equations. In *IN: State-of-the-art surveys on computational*

- mechanics (A90-47176 21-64)*. New York, American Society of Mechanical Engineers, 1989, p. 71-143. Research supported by DARPA., volume 1, pages 71–143, 1989.
- [10] Anthony T Patera. A spectral element method for fluid dynamics: Laminar flow in a channel expansion. *Journal of Computational Physics*, 54(3):468–488, Jun 1984. ISSN 0021-9991. doi: 10.1016/0021-9991(84)90128-1. URL [http://dx.doi.org/10.1016/0021-9991\(84\)90128-1](http://dx.doi.org/10.1016/0021-9991(84)90128-1).
- [11] Karol Z. Korczak and Anthony T. Patera. An isoparametric spectral element method for solution of the navier-stokes equations in complex geometry. *Journal of Computational Physics*, 62(2):361–382, Feb 1986. ISSN 0021-9991. doi: 10.1016/0021-9991(86)90134-8. URL [http://dx.doi.org/10.1016/0021-9991\(86\)90134-8](http://dx.doi.org/10.1016/0021-9991(86)90134-8).
- [12] Paul Fischer and Julia Mullen. Filter-based stabilization of spectral element methods. *Comptes Rendus de l'Académie des Sciences - Series I - Mathematics*, 332(3):265 – 270, 2001. ISSN 0764-4442. doi: [http://dx.doi.org/10.1016/S0764-4442\(00\)01763-8](http://dx.doi.org/10.1016/S0764-4442(00)01763-8). URL <http://www.sciencedirect.com/science/article/pii/S0764444200017638>.
- [13] R. Pasquetti and C.J. Xu. Comments on “filter-based stabilization of spectral element methods”. *Journal of Computational Physics*, 182(2):646 – 650, 2002. ISSN 0021-9991. doi: <http://dx.doi.org/10.1006/jcph.2002.7178>. URL <http://www.sciencedirect.com/science/article/pii/S0021999102971780>.
- [14] N. Young. *An Introduction to Hilbert Space*. Cambridge mathematical textbooks. Cambridge University Press, 1988. ISBN 9780521337175. URL https://books.google.no/books?id=_igwFHKwcyYC.
- [15] M. O. Deville, P. F. Fischer, and E. H. Mund. High-order methods for incompressible fluid flow. 2002. doi: 10.1017/cbo9780511546792. URL <http://dx.doi.org/10.1017/CB09780511546792>.
- [16] *NEK5000 documentation*, 2015.
- [17] Y. Maday, Anthony T. Patera, and Einar M. Rønquist. An operator-integration-factor splitting method for time-dependent problems: Application to incompressible

- fluid flow. *Journal of Scientific Computing*, 5(4):263–292, 1990. ISSN 0885-7474. doi: 10.1007/BF01063118. URL <http://dx.doi.org/10.1007/BF01063118>.
- [18] Daniel Eriksson. Neutrally buoyant gas dispersion within an urban street canyon. *MekIT'15.*, 2015.
- [19] S. Turek and M. Schäfer. Recent benchmark computations of laminar flow around a cylinder, 1996.