Refined radial basis function-generated finite difference analysis of non-Newtonian natural convection

Miha Rot¹,² and Gregor Kosec²

¹Jozef Stefan International Postgraduate School, Jamova cesta 39, 1000 Ljubljana, Slovenia

²Jožef Stefan Institute, Parallel and Distributed Systems Laboratory, Jamova cesta 39, 1000 Ljubljana, Slovenia

(Dated: February 24, 2025)
In this paper we present a refined Radial Basis Function-generated Finite Difference (RBF-FD) solution for a non-Newtonian fluid in a closed differentially heated cavity. The non-Newtonian behaviour is modelled with the Ostwald-de Waele power law and the buoyancy with the Boussinesq approximation. The problem domain is discretised with scattered nodes without any requirement for a topological relation between them. This allows a trivial generalisation of the solution procedure to complex irregular three dimensional (3D) domains, which is also demonstrated by solving the problem in a two dimensional (2D) and 3D geometry mimicking a porous filter. The results in 2D are compared with two reference solutions that use the Finite volume method in a conjunction with two different stabilisation techniques, where we achieved good agreement with the reference data. The refinement is implemented on top of a dedicated meshless node positioning algorithm using piecewise linear node density function that ensures sufficient node density in the centre of the domain while maximising the node density in a boundary layer where the most intense dynamic is expected. The results show that with a refined approach, more than 5 times fewer nodes are required to obtain the results with the same accuracy compared to the regular discretisation. The paper also discusses the convergence with refined discretisation for different scenarios for up to $2 \cdot 10^5$ nodes, the impact of method parameters, the behaviour of the flow in the boundary layer, the behaviour of the viscosity and the geometric flexibility of the proposed solution procedure.

Keywords: meshless method, dimension independent, refined discretisation, Navier-Stokes, non-Newtonian fluid, power-law fluid, natural convection, heat transport

---

a) Electronic mail: miha.rot@ijs.si
I. INTRODUCTION

Natural convection, a type of flow driven by the temperature-dependent density of a fluid, is a cornerstone of many natural and industrial processes. The most vivid examples in nature where it plays a crucial role are probably weather systems, e.g. sea and land breezes. In industry, it is of paramount importance in metal casting, various heating systems, food processing, etc. The minimal model describing natural convection involves coupled heat transfer and fluid dynamics, where simple fluids such as air and water are usually modelled as Newtonian fluids, i.e. the viscosity of the fluid is constant. However, such an approximation becomes insufficient when dealing with more complex fluids like melts of large and complicated molecules, polymer suspensions, foams, biological fluids such as blood, food, etc. In such fluids, also called non-Newtonian fluids, the relationship between stress and strain is no longer linear and the viscous stress becomes related to the shear rate. The behaviour of purely viscous non-Newtonian fluids can generally be divided into two groups based on how their viscosity changes with increasing shear rate, namely shear-thinning, where viscosity decreases, and shear-thickening, where viscosity increases.

Because of its importance in industry and in understanding nature, the study of Non-Newtonian Natural Convection (NNC) has attracted much research attention. Since closed-form solutions are rare and limited to extreme simplifications, the problem is usually treated numerically. The dynamics of non-Newtonian fluids have been studied in various fields, from non-Newtonian blood flow in arteries to injection moulding of plastics. In particular, natural convection in cavities has been thoroughly analysed due to its direct application in industry.

From the numerical analysis point of view, the most commonly reported approaches to solving NNC in cavities are based on the Finite Volume Method (FVM) for the discretisation of the relevant partial differential operators and Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) for pressure-velocity coupling. A comprehensive study of NNC solved with FVM and Quadratic Upstream Interpolation for Convective Kinematics (QUICK) to treat the convective terms was presented in, and with upwind stabilisation of convective terms in. The FVM with QUICK was further investigated in in solving NNC with internal rotating heater and cooler. Moraga et al. demonstrated the multigrid FVM with a fifth power differentiation scheme for convective transport in the solution of NNC with
phase change. A similar numerical approach was also used in the solution of NNC in three dimensions (3D)\(^{16}\). In conjunction with the Finite Difference Method (FDM) and upwind stabilisation, the solution of NNC with internal heat source was recently demonstrated in\(^{17}\). The NNC was also solved with a Finite Element Method (FEM)\(^{18,19}\) as well as with the Lattice Boltzmann Method (LBM)\(^{20}\).

All the aforementioned solution methods are mesh-based, i.e., the nodes are structured into polygons that completely cover the computational domain, a process also known as meshing. In FDM, FVM and LBM, a regular grid is often used, making meshing a trivial task, but at the cost of complications with irregular geometries and potential refinement. At FEM, meshing is mandatory and often also the most time-consuming part of the entire solution process, especially for realistic 3D geometries, which generally cannot be automatically meshed and therefore often require the user’s help. An alternative to the mesh-based methods is the meshless approach\(^ {21}\). The conceptual difference between mesh-based and meshless methods is that in the latter all relationships between nodes are defined solely by inter-nodal distances. An important implication of this distinction is that the meshless methods can work with scattered nodes, which greatly facilitates the consideration of complex 3D geometries and adaptivity.

In this paper, we introduce a novel refined Radial Basis Function-generated Finite Difference (RBF-FD)\(^ {22}\) meshless solution of NNC in two dimensional (2D) and 3D irregular domains computed on automatically generated scattered nodes\(^ {23}\), using the artificial compressibility method (ACM) for pressure velocity coupling\(^ {24–26}\), and explicit time stepping. No stabilisation of the convective terms is used to minimise the impact of numerical diffusion in the results. We present a unified NNC solution procedure for 2D and 3D that can be easily extended to arbitrary geometries and inherently supports \(hp\)-adaptivity\(^ {27,28}\).

Results for a reference 2D case are compared with data from Turan et al.\(^ {13}\) and Kim et al.\(^ {12}\), showing that our results are in the range of the comparative data. Therefore, in addition to a novel solution procedure, we also extend the range of available numerical solutions for the given problem with a completely different numerical approach. Note that the reference solutions differ only in the stabilisation of the convective terms (Upwind vs QUICK), otherwise in both papers results are computed with FVM and SIMPLE coupling.

In section I\(\text{I}\) a mathematical model of NNC is discussed, followed by a presentation of the meshless numerical method and solution procedure in section I\(\text{II}\). The analysis of results
for the reference 2D case and a showcase of the method’s versatility on more complex cases are shown in sections IV and V.

II. PROBLEM FORMULATION

The dynamics of natural convection in non-Newtonian fluids are governed by a system of three partial differential equations describing the continuity of mass, the conservation of momentum, and the conservation of energy

\[ \nabla \cdot \mathbf{v} = 0, \]

\[ \rho \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\nabla p + \nabla \cdot \left( \eta \left( \nabla \mathbf{v} + (\nabla \mathbf{v})^T \right) \right) - g \rho \beta T \Delta, \]

\[ \rho c_p \left( \frac{\partial T}{\partial t} + \mathbf{v} \cdot \nabla T \right) = \nabla \cdot (\lambda \nabla T), \]

\[ \eta = \eta_0 \left( \frac{1}{2} \left\| \nabla \mathbf{v} + (\nabla \mathbf{v})^T \right\| \right)^{\frac{n-1}{2}}, \]

with \( \mathbf{v}, T, p, \rho, \eta, g, \beta, T_\Delta, c_p, \eta_0, n \) representing the flow velocity field, temperature field, pressure field, viscosity field, density, gravity, thermal expansion coefficient, temperature offset, heat capacity, viscosity constant and non-Newtonian exponent, respectively.

The buoyancy force that drives the natural convection dynamics is relatively weak and ensures that the maximum velocity remains well below the speed of sound. This allows us to model the fluid as incompressible and reduce the continuity equation to \( \nabla \cdot \mathbf{v} = 0 \). The fluid motion is described by the Navier-Stokes equation, which is modified from its usual form by the addition of a force term describing the buoyancy caused by the thermal expansion. This force is approximated by the Boussinesq approximation, which is based on the assumption that the acceleration of a fluid driven by natural convection remains insignificant compared to gravity and consequently the small thermal fluctuations of the density only play a role when amplified by the strong gravity in the buoyancy term. The Boussinesq approximation couples the fluid motion described by the Navier-Stokes equation with the temperature described by the energy equation and thus establishes the model for natural convection driven by the presence of a temperature gradient.

The constant viscosity \( \eta \) is replaced by the Ostwald-de Waele power law model defined in Eq. (4). The shear dependence is captured by the exponentiated tensor norm of the shear rate tensor and controlled by the exponent \( n \), which controls the extent of the non-
Newtonian behaviour, and $\eta_0$, which is used as a scaling factor.

The model reduces to a Newtonian fluid when $n = 1$ and can be used to describe both shear thickening behaviour with $n > 1$ and shear thinning behaviour with $n < 1$. We focus on the latter as it leads to stronger convection, i.e. more interesting flow behaviour, and is more common in realistic fluids. A visualisation of the viscosity dependence for an arbitrarily chosen range of shear rate norms can be found on the right-hand panel of Figure 1 for a range of exponents that we will use in further analysis. It can be seen from the figure that while $n$ has a dramatic impact on the effective viscosity, there is also a problem with the divergent viscosity at low shear rates in the shear thinning regime. This is mainly a problem for the initial, transient part of the simulation, where a flow pattern emerges from the initially stationary fluid and can be solved by downward bounding the shear rate norm used in the power law to $10^{-10}$.

Model parameters that reflect the behaviour of a realistic fluid are typically determined by fitting the model to experimental data. Ostwald-de Waele power law model is the most basic approach to characterizing shear-dependent behaviour, and it is only appropriate for the intermediate shear rate regime. To accurately capture the asymptotic behaviour of realistic shear-thinning fluids at both low and high shear rates, the specialized models with more intricate algebraic structure due to additional parameters are used. Nevertheless, from a numerical standpoint, these more complex models do not significantly affect the solution methodology nor do they introduce any additional numerical challenges.

The natural convection described with the system of equations described above is applied to the De Vahl Davis case, a differentially heated square cavity with height and width $L = 1$, shown schematically in the left graph of Figure 1. The left wall is kept at a constant temperature $T_C = -1$, while the right wall is kept at a higher constant temperature $T_H = 1$, inducing the heat transfer that drives the dynamics of the system. The top and bottom boundaries are insulated. No-slip boundary conditions for velocity are imposed on all walls.

The difference in boundary temperatures creates a temperature gradient within the cavity that leads to natural convection. The convective flow appears due to the temperature-dependent changes in fluid density and the resulting buoyancy forces, which are described by the Boussinesq approximation term in Eq. (2). The fluid cools and becomes denser at the left wall, so it falls and moves to the right, where it is heated by the hot wall, rises and completes the circular flow.
The circular flow caused by natural convection significantly increases the heat transfer between the differentially heated walls compared to conduction alone. The ratio between the two is known as the Nusselt number and provides a convenient reduction that expresses the convective heat transfer in a single scalar value. Such reduction can be further analysed to determine the temporal behaviour, as shown in Figure 5, from which we can deduce when the steady state was reached. The Nusselt number used in the following analysis is calculated as the average of the values in the cold wall nodes

\[ \text{Nu} = \frac{L}{T_H - T_C} \left| \frac{\partial T}{\partial x} \right|_{x=0}. \]  

The problem is further characterised by two dimensionless numbers. The Prandtl number (Pr) is a material property that expresses the ratio between the heat and momentum transport properties of the fluid. The Rayleigh number (Ra) can be interpreted as the ratio of buoyancy and thermal diffusivity, and is the product of the Grashof and Prandtl numbers. For the purposes of this study it can be interpreted as an analogue of the Reynolds number for natural convection, with larger values implying wilder dynamics. Both Ra and Pr are a
function of viscosity and must be modified to account for non-Newtonian viscosity:

\[ Pr = \frac{\eta_0 \alpha^{n-2} L^{2-2n}}{\rho} \]

\[ Ra = \frac{\rho g \beta \Delta T L^{2n+1}}{\alpha^n \eta_0} \]

with \( \alpha = \frac{\lambda}{c_p \rho} \) as the thermal diffusivity. The definitions for the dimensionless numbers match the reference solution\(^{13}\) to facilitate comparison, as does the dimensionless time:

\[ \hat{t} = \frac{\alpha}{L^2} t \]

and velocity:

\[ \hat{v} = \frac{L c_p \rho}{\lambda} v. \]

We omit the caret notation, since dimensionless values are used wherever velocity or time are referred to in the following discussion.

### III. NUMERICAL SOLUTION PROCEDURE

Our goal is to solve the problem (1)-(3) using an approach that is as general as possible, including the generality of the number of dimensions, the order of the method, and the shape of the considered domain, while supporting a spatially variable discretisation. To achieve this, we employ the RBF-FD method with augmenting monomials. A meshless numerical technique operating on scattered nodes.

The first step is to discretise the domain, which in the meshless context means populating with scattered nodes. Although in the early stages of meshless development some authors used even randomly generated nodes\(^{21}\), it is now generally accepted that despite the apparent robustness of meshless methods regarding the node positioning, nodes still need to be generated according to certain rules\(^{23}\), i.e. nodes have to "uniformly" cover the domain with minimal empty spaces and satisfy the minimum distance requirement to avoid ill-conditioning of the approximation. There are several specially designed algorithms for meshless discretisation, ranging from expensive iterative\(^{34}\) to advancing front\(^{35}\) approaches. In this paper we use a Poisson disk sampling based advancing front approach\(^{23,36-38}\). The core of the algorithm is the iteration, where candidate nodes are sampled around the already positioned nodes and only those that do not violate the minimal distance requirement are
Figure 2. A comparison between node distributions in domains populated with a constant node density on the left, refined density in the middle and the obstructed domain with refined narrow channels on the right.

We begin the analyses with a constant density node distribution, shown in the left panel of Figure 2, which, as we will see in section IV C, is a sub-optimal strategy for the problem at hand with intense dynamics in the boundary layer (see section IV A). An improved node placement strategy, shown in the central panel of Figure 2 and discussed in section IV C, uses prior knowledge of the system to devise a target node density that significantly reduces computational cost without sacrificing accuracy. This solution is still not ideal, as it requires physical intuition and manual input, but it demonstrates the potential for the eventual goal, which is an $h$-adaptive solution with an appropriate error indicator. In a rightmost plot of Figure 2 we demonstrate discretisation of an irregular domain that will be used in section V.

Once the computational nodes are generated we identify the approximation stencil, i.e. the nodes used in local approximation of differential operators. Here, we will resort to simplest stencil strategy, where for each node $i$, $s$ neighbouring nodes are identified as...
a stencil $S_i$. Besides basic closest-node stencils, recent studies\cite{39} suggest that advanced symmetric stencils might significantly enhance the stability of meshless approximations. However, most research still employs nearest neighbours stencils; thus, we will also use it in this paper.

In the next step we construct a generalised finite difference approximation to numerically evaluate the linear differential operator $L$ in the central node from the approximated function’s values in stencil nodes

$$(Lu)_i \approx \sum_{j=1}^{s} w_{i,j} u_{S_i(j)}, \quad (10)$$

where $u_k$ denotes the value of the arbitrary approximated function $u$ at the position of the $k$-th node $p_k$. The weights $w$ are determined through the demand that the Eq. (10) is exact for a set of basis functions, in our case radial basis functions (RBF)

$$\phi(j,k) = \phi \left( \frac{\| p_k - p_j \|}{\delta_j} \right), \quad (11)$$

where we introduced $\delta_j$ as a local scaling factor that decouples the approximation from the choice of coordinate system. It is set to an arbitrary local measure of distance, e.g., the distance to the closest stencil node, and is of the utmost importance when using RBFs that use a scaling parameter. Finally, we get a local linear system $Aw_i = b$

$$
\begin{bmatrix}
\phi(S_i(1), S_i(1)) & \cdots & \phi(S_i(1), S_i(s)) \\
\vdots & \ddots & \vdots \\
\phi(S_i(s), S_i(1)) & \cdots & \phi(S_i(s), S_i(s))
\end{bmatrix}
\begin{bmatrix}
w_{i,1} \\
\vdots \\
w_{i,s}
\end{bmatrix}
= 
\begin{bmatrix}
(\mathcal{L}\phi)(i, S_i(1)) \\
\vdots \\
(\mathcal{L}\phi)(i, S_i(s))
\end{bmatrix}, \quad (12)
$$

for each node with the solution yielding stencil weights $w_i$. The right hand side vector $b$ is formed by applying the linear operator $L$ to the basis function and evaluating the result with an argument analogous to Eq. (11). We use polyharmonic splines (PHS) basis

$$\phi(r) = r^k, \quad (13)$$

with odd order $k$, additionally augmented with polynomials for ensuring the polynomial reproduction and positive definiteness\cite{40}. This setup is widely recognised as an RBF-FD method in meshless community.

The system for stencil weights is expanded with $N_p = \binom{m+d}{m} q$, where $m$ denotes the monomial order and $d$ the spatial dimension. The monomials are scaled with a
similar argumentation as RBFs in Eq. (11)

\[ q_l(j, k) = q_l \left( \frac{p_k - p_j}{\delta_j} \right). \]  

(14)

The \( A w_i = b \) system for approximation weights from Eq. (12) is augmented with monomials

\[
\begin{bmatrix}
A & Q \\
Q^T & 0
\end{bmatrix}
\begin{bmatrix}
w_i \\
\lambda
\end{bmatrix}
= \begin{bmatrix}
b \\
c
\end{bmatrix},
\]

(15)

\[ Q = 
\begin{bmatrix}
q_1(S_i(1), S_i(1)) \cdots q_{N_p}(S_i(1), S_i(1)) \\
\vdots & \ddots & \vdots \\
q_1(S_i(s), S_i(1)) \cdots q_{N_p}(S_i(s), S_i(1))
\end{bmatrix},
\]

\[ c = 
\begin{bmatrix}
(\mathcal{L}q_1)(S_i(1), S_i(1)) \\
\vdots \\
(\mathcal{L}q_{N_p})(S_i(1), S_i(1))
\end{bmatrix},
\]

with the additional weights \( \lambda \) treated as Lagrange multipliers and discarded after computation.

Augmentation with an order of at least \( m = \frac{k-1}{2} \) is required to guarantee the positive definiteness for a PHS with order \( k \). Higher orders of augmentation \( m \) provide better convergence characteristics\cite{12,13}, with the order of convergence

\[ \mathcal{O}(h^{m+1-\ell}) \]  

(16)

for \( m \geq l \), where \( \ell \) is the order of the linear operator \( \mathcal{L} \). The higher accuracy comes at the cost of increased computational complexity, since the required stencil size is \( s \geq N_p \), with \( s > 2N_p \) as the often recommended value\cite{13}. Increased stencil size affects both the pre-computation of the approximation weights, with complexity \( \mathcal{O}((s + N_p)^3) \), and the eventual scalar product \( \mathcal{O}(s) \) evaluation. We will use \( k = 3 \), which is the minimum odd value that can be used to approximate second order derivatives, for all numerical results presented in this paper and compare the results for monomial orders \( m = 2 \) and \( m = 4 \). The support size is chosen conservatively as \( s = 2N_p + 1 \) unless otherwise specified. Nevertheless, augmenting monomials open an opportunity for \( hp \)-adaptivity, since their order directly controls the order of the method\cite{15}.

Now that we have the RBF-FD approximation for derivatives, next step is to formulate a solution procedure for the problem at hand. The pressure-velocity coupling in the incompressible Navier-Stokes system is performed using the artificial compressibility method (ACM)\cite{24,25} that was first introduced by Chorin\cite{46} in 1967 but is now experiencing resurgence due to its explicit and local nature that allows for easy parallelisation and GPU usage\cite{47,48}.
The method works by artificially introducing a slight compressibility into the otherwise incompressible system in order to calculate the pressure field.

Since the focus of this study lies in the spatial discretisation we use the simple first-order explicit Euler method for the temporal discretisation of the Navier-Stokes equation. The intermediate velocity

\[
v' = v + \Delta t \left( \nabla \cdot \left( \eta \left( \nabla v + (\nabla v)^T \right) \right) - v \cdot \nabla v - g \rho \beta T \right) \tag{17}\]

is calculated by using the viscosity field \( \eta \) calculated from the previous step velocity according to \[4\] and the offset \( T_\Delta \) in Boussinessq term based on the previous step temperature field. Note that the pressure term is omitted at this step. The time-step

\[
\Delta t = \min \left( \min_i \left( c_1 \frac{h_i}{\|v_i\|_2} \right), \min_i \left( c_2 \frac{\rho h_i^2}{2\eta_i} \right) \right) \tag{18}
\]

is calculated dynamically during each step of the iteration to adapt to the changing velocity and viscosity in computational nodes with a wide range of inter-nodal distances \( h_i \). The time-step constants are chosen as \( c_1 = 0.05 \) and \( c_2 = 0.15 \) based on trial and error. Values of the dynamic timestep during the simulation with a constant discretisation density using \( h = 0.005 \) are shown on the right graphs of Figure 5. The variable time-step is beneficial for most of the displayed cases as the most stringent requirements only occur during the relatively short period of initial flow formation. Note that the \( h^2 \) term in the viscous limit is likely to dominate the timestep selection for very dense discretisations and might prove to be a limiting factor.

We omitted the pressure term from the initial intermediate velocity \( v' \) because the subsequent pressure-velocity coupling is done iteratively. First the intermediate velocity is corrected with the previous pressure field

\[
v = v' - \frac{\Delta t}{\rho} \nabla p, \tag{19}\]

then the pressure field is corrected to counteract any divergence present in the velocity

\[
p \leftarrow p - \Delta t C^2 \rho (\nabla \cdot v), \tag{20}\]

while enforcing

\[
\frac{\partial p}{\partial n} = 0, \tag{21}\]
on the boundary, with \( \hat{n} \) being the corresponding surface normal vector. The strength of the artificial compressibility effects is recomputed at every step and governed with the artificial speed of sound \( C \)

\[
C = \gamma \max_i (\|v_i\|_2, ||v_{ref}||_2),
\]

(22)

where \( \gamma \) is the compressibility parameter, and \( v_{ref} \) a reference velocity introduced to prevent potential issues caused by \( C \) reaching zero. We use \( \gamma = 5 \) for all computations presented in this paper. If a time-accurate solution was required, the pressure-velocity coupling iteration would have to be repeated until the maximum divergence of the velocity field dropped below a desired level. In steady state cases like the DVD in examined regime a single iteration is sufficient as we are not really interested in the transient regime.

The new velocity is then used in the advection part of the energy transfer equation, again discretised with the explicit Euler method, which is then used to calculate the new temperature field. The same dynamic timestep \( \Delta t \) calculated for the momentum equation can also be used for the energy equation as the latter is well within stability limits of the former when dealing with \( Pr = 100 \gg 1 \) case.

IV. RESULTS

A. General flow behaviour

We will study the impact of non-Newtonian behaviour on three flow cases specified with Rayleigh and Prandtl dimensionless numbers as defined in section II. Rayleigh numbers \( Ra = \{10^4, 10^5, 10^6\} \) are chosen to capture changes in the upper range of steady natural convection regime, while \( Pr = 100 \) remains constant and is chosen to facilitate comparison with the existing reference solution. The non-Newtonian behaviour is examined with five values for the non-Newtonian exponent \( n = \{0.6, 0.7, 0.8, 0.9, 1\} \) progressing from the \( n = 0.6 \) case that exhibits the strongest shear-thinning behaviour to the Newtonian \( n = 1 \) case. Note that the most extreme case with \( Ra = 10^6 \) and \( n = 0.6 \) is close to the edge of the steady regime. Further increases in \( Ra \) or decreases in \( n \) would result in an oscillatory flow.

A sample of the resulting flow profiles can be seen in Figure 3 where the velocity magnitude is displayed as a heat map and overlaid with temperature contours. All cases exhibit the previously described circulation caused by natural convection, but there are drastic
differences in the maximum velocity and the thickness of boundary layers as Ra and shear-thinning increase.

Both effects are expected and can be explained by the definition of varied parameters. The Rayleigh dimensionless number is defined as the product of Prandtl, which we keep constant, and Grashof dimensionless numbers. The latter expresses a ratio between buoyant and viscous forces and explains why increased Ra results in cases where fluid in the boundary layer convects away before conducting much heat to the neighbouring fluid. Similarly reduced \( n \) decreases viscous penalty for high velocity gradients close to the constant temperature edges, leading to a further reduction in boundary layer thickness.

Velocity fields also offer the first opportunity to verify the results. We compare the vertical velocity cross-section at \( y = 0.5 \) with the reference solution in Figure 4. The position and value of the largest vertical velocity in the reference solution is added with an estimated error caused by the process of extracting the data from a similar cross-section plot. Our results match the reference solution across the whole range of the considered parameters with the exception of the \( Ra = 10^6, n = 0.6 \) case with problematic convergence that is discussed and solved in the following sections.

In the subsequent analysis we utilise the Nusselt number as a scalar observable to simplify the description of the system’s behaviour, facilitating temporal observation and comparison between different cases. One such example is shown in Figure 5 where we track the evolution of the Nusselt number to determine when we reach a steady state. We can also observe the effects of stronger shear-thinning. Cases with lower \( n \) are faster to reach the stationary state as it is easier for convection to start, with the lower effective viscosity, and to play a bigger part in heat transfer as reflected in higher Nusselt values. The initial Nusselt number is high due to the high temperature gradient at the constant temperature boundary when starting from a zero-temperature zero-velocity initial condition. The value then decreases as the importance of conductive heat transport increases until circulation is established. Alternatively we could start with the diffusive field as the initial condition for temperature but this would only lead to a more violent flow formation with little benefit to accelerating the convergence to the convection dominated steady state.
Figure 3. Flow profiles for a selection of cases. Velocity magnitude is visualised with a heat-map while the overlaid contours display the changes in temperature. Each sub-figure has a distinct velocity range specified by the colourbar above.

B. Convergence under $h$ refinement

The node density convergence analysis is performed to analyse node independence. We repeatedly run the same cases with decreasing internodal distance $h$ and examine how the average Nusselt number on the cold boundary changes with increasing number of computa-
Figure 4. Cross-section of vertical velocity $v_y$ close to the right wall at $y = 0.5$ for a range of different $Ra$ and $n$, calculated with $Pr = 100$ and $m = 4$. The error bars show the locations and values of maximum vertical velocity in corresponding plots from a reference solution.\textsuperscript{13}

Figure 5. \textit{left}: Time evolution of the average Nusselt number on the cold wall that is used as a scalar observable for the system’s dynamics. \textit{right}: Dimensionless values of the dynamic timestep throughout the simulation. Cases shown in this Figure use a constant discretisation density with internodal distance $h = 0.005$ corresponding to $N = 35222$ computational nodes.
tional nodes. The velocity and temperature fields are initially set to zero and the simulation is ran to $t = 0.1$ which is determined to be sufficient to ensure that a stationary state has been reached as seen in Figure 5.

The convergence behaviour for Newtonian ($n = 1$) cases shown in the bottom row of Figure 6 is very tame, as the coarsest considered discretisation already provides results with less than 10% discrepancy compared to the finest. Unfortunately, this is no longer the case as we progress towards non-Newtonian cases with a thinner boundary layer. The variation of the observed values increases and the convergence rate decreases. This culminates in the most extreme case with $Ra = 10^6$ and $n = 0.6$, shown in the upper right graph of Figure 6, where adequate convergence is not achieved with the considered node counts. This can be further corroborated with the reference mismatch already observed for this case in Figure 4.

Proceeding to even higher node densities, requiring longer computational times, is wasteful, especially as the utilised numerical method allows for an elegant optimisation described in following sections.

In next step we compare different monomial augmentation orders $m = 2$ and $m = 4$, shown as different colours in Figure 6. We consider the finest available discretisation to produce an "accurate" solution and study the rate of convergence relative to that in the left graph of Figure 7. From the log-log plot of the average Nusselt number offset against the inter-nodal distance $h$, we can determine that even though the higher augmentation order leads to a smaller initial error, the rate of convergence is similar for both. The discrepancy from the expected order, expressed with Eq. (16), are most likely caused by non-linearities in the system.

There is an interesting transition in convergence curves shown in Figure 6 as we move towards the upper right corner, corresponding with cases that exhibit a thinner boundary layer with higher velocities. The calculated Nusselt numbers initially rise as we increase node density before falling towards the value they eventually converge to. This behaviour can be better understood by examining the velocity cross-section convergence, shown in the middle and right graph of Figure 7 for two of the $n = 0.6$ cases where it is most apparent. The velocity profiles show that the boundary layer initially becomes narrower and faster until a sufficient number of nodes is present to adequately capture the dynamics. This behaviour provides us with additional motivation for refining the discretisation close to the heated and cooled boundaries. Furthermore the hypothesized issues with derivative approximation
Figure 6. Convergence of the average Nusselt number on the cold boundary with columns for all of the considered Ra and rows for different non-Newtonian exponents $n$. The two different considered monomial augmentation orders $m$ are shown with different plot colours. Node count is proportional to the inter-nodal distance as the density is constant throughout the domain.

appear to also have a connection with the stencil size and/or approximation order as the Nusselt number peaks visible in Figure 6 move towards higher densities when augmented with a higher monomial order.

Furthermore, we examine the violation of symmetry

$$u(x, y) = -u(1 - x, 1 - y)$$  \hspace{1cm} (23)

as an alternative method for assessing the fitness of the solution. This is additionally motivated by the fact that the $Ra = 10^6$, $n = 0.6$ case with problematic convergence exhibits clearly visible asymmetry in velocity and temperature profiles shown in Figure 3. First, we
Figure 7. left: Convergence of average Nusselt number on the cold boundary towards the value calculated with the finest discretisation that was used as the ground truth. middle & right: Changes in the interpolated vertical velocity profiles close to the boundary at $y = 0.6$ with increasing number of uniformly positioned computational nodes. Cases shown are parametrised with $Pr = 100$ and $n = 0.6$ and computed with $m = 2$. Dots display the horizontal position and vertical velocity in computational nodes that lie less than $\frac{h}{2}$ from the interpolation $y$.

introduce a measure of symmetry violation by interpolating the vertical velocity $v_y$ at an arbitrarily chosen $y$ and $1 - y$ and calculating the relative error as

$$\epsilon = \frac{\max_{x \in [0,1]}(|v_y(x, y) + v_y(1 - x, 1 - y)|)}{\max_{x \in [0,1]}(|v_y(x, y)|)}$$

(24)

where we normalise the maximum offset between a vertical velocity and its symmetric value with the maximum vertical velocity for a given case and selected $y$. This choice of denominator is preferable as it focuses on the symmetry errors in the relevant high velocity part of the domain while remaining relative for comparison between different cases. The resulting symmetry errors as a function of the number of nodes are shown in Figure 8 and corroborate the previous discussion that was based on convergences in Figure 6. Additionally, we can also confirm the observation from the left graph of Figure 7 as both $m = 2$ and $m = 4$ exhibit similar convergence behaviour. Note that in addition to confirming the convergent behaviour of the method, the introduced symmetry violation can also be interpreted as a lower bound for the error.
Figure 8. Convergence of the relative symmetry error at $y = 0.75$ with columns for all of the considered Ra and rows for different non-Newtonian exponents $n$. The two different considered monomial augmentation orders $m$ are shown with different plot colours. Node count is proportional to the inter-nodal distance as the density is constant throughout the domain.

C. Refinement

Small flow structures and large velocity gradients (boundary layer) only occur in a relatively small part of the domain as seen in the right graph of Figure 7 whose $x$-axis only covers about 10% of the domain’s width. Needlessly covering the entire domain with the high node density, that is only required close to the cold and hot boundary, drastically increases the number of operations required at every time-step without improving the results.

We introduce a variable node density expressed as the inter-nodal distance. In general, the
inter-nodal distance would be a function of position but we use a symmetric configuration, shown schematically in the left graph of Figure 9, that only depends on the distance to the closest boundary \( d \). The idea is to position nodes with inter-nodal distance \( h_1 \) in a band within \( w \) of the boundary to ensure a sufficient discretisation in this intense region, while using much coarser discretisation \( h_2 \) in the centre of the domain, with a linear transition between the two

\[
    h = \begin{cases} 
        h_1, & \text{where } d < w \\ 
        h_1 + \frac{d-w}{w} (h_2 - h_1), & \text{otherwise}
    \end{cases}
\]

(25)

The value of the central inter-nodal distance \( h_2 \), expressed in terms of minimal distance \( h_1 \) to prevent excessive density gradients that would require further analysis in terms of method stability and accuracy. Furthermore \( h_2 \) is bounded with \( h_{\text{max}} \) to prevent instabilities. This relatively simple refinement scheme leads to drastic savings in node count shown in the right graph of Figure 9. Even with the relatively conservative refinement parameters, there are almost an order of magnitude fewer nodes for the same boundary inter-nodal distances \( h_1 \). The slight non-linearity in the node count is caused by \( h_{\text{max}} \) limited \( h_2 \) when \( h_1 \) is large. The savings appear to be even more dramatic when a refined discretisation is visually compared to an unrefined one as seen in Figure 2.

We present an additional observation regarding the numerical method parameterisation that can only be made when dealing with aggressively refined discretisations. The recommended support size \( s > 2N_p \) for the approximation, corresponding to \( s = 12 \) in the analysed case, is not sufficient to reach a stable solution for higher refinement ratios \( \frac{h_2}{h_1} \) as shown in the left graph of Figure 10. The results for the \( \text{Ra} = 10^5 \), \( \text{Pr} = 100 \), \( n = 0.6 \) case calculated with \( m = 2 \) and \( h_1 = 0.0025 \) show that the range of stable refinement ratios raises with increasing support size until reaching \( \frac{h_2}{h_1} = 25 \) with \( s = 15 \). Based on this we conservatively choose \( s = 2.5N_p + 1 \) rounded to the closest integer, corresponding to \( s = 16 \) in Figure 10 as a rule for support size selection in the subsequent refined cases. The Nusselt number for the selected support size remained practically the same for the entire range of refinement ratios resulting in discretisations ranging from 140134 nodes for \( \frac{h_2}{h_1} = 1 \) to 12438 nodes for \( \frac{h_2}{h_1} = 25 \). Note that the \( h_{\text{max}} \) was not enforced for the discretisation and that the dense band
Figure 9. *left:* A schematic representation of the inter-nodal distance as a function of distance from the boundary. *right:* Computational node count as a function of minimum inter-nodal distance for a constant density and a refined case.

width was set to $w = 0$, further amplifying the impact of the changing node density on the boundary flow layer. The discrepancy between the different considered support sizes seen in Nusselt numbers on the left is also apparent from the vertical velocity cross-sections shown on the right graph of Figure 10 with the larger support sizes appearing to smoothen the velocity peak.

The refinement parameters, used in Figure 9 and in other refined results with unspecified values, were chosen based on a convergence analysis. The analysis performed at $Ra = 10^6$, $Pr = 100$ and $n = 0.6$ would need to be repeated for other cases with a significant differences in boundary layer thickness and other flow characteristics. Refinement parameters have been individually varied for different border densities with the results shown in Figure 11.

We assume that a further refinement in parameters would not lead to significant changes and therefore we norm the Nusselt number, shown in the upper graphs, to the best present value. Normalized values allow for comparison between cases with a different maximum density and conveniently show the relative offset in value.

The refinement ratio $\frac{h_2}{h_1}$ sweep is performed with a band width $w = 0.05$ that is wide enough to not have a meaningful impact on the results as established in the following paragraph. The resulting Nusselt numbers, shown in the upper left graph of Figure 11, are relatively unaffected by the refinement ratio, as long as the resulting $h_2$ is small enough to
Figure 10. left: The average Nusselt number on the cold wall as a function of refinement ratio $h_2/h_1$ for different support sizes $s$. right: Vertical velocity cross-section close to the right wall at $y = 0.5$ corresponding to the rightmost cases from the left graph with $h_2/h_1 = 25$ and different support sizes $s$. The cross-section shows values in nodes closer than $5h_1$ in $y$ coordinate. Support sizes are chosen as $s = k_sN_p + 1$, rounded to the closest integer for $k_s \in \{1.75, 2, 2.25, 2.5, 2.75, 3\}$.

ensure numerical stability. Results are stable, with variations within 1%. We chose $h_2/h_1 = 20$ as a conservative choice for refinement aggressiveness, corroborated with Figure 10 for the selected support size. The conservative choice of refinement ratio is justified by the graph of computational node count dependence, shown in the lower left graph of Figure 11, where we can see that further increasing refinement ratio leads to diminishing reduction in the number of computational nodes. This analysis as also used to set the $h_{\text{max}} = 0.05$ based on refinement ratios where stable solution was no longer achieved.

The dense boundary width $w$ sweep is calculated with a refinement ratio of $h_2/h_1 = 10$, that has been determined to be adequate in the previous paragraph, and the results are shown in the upper right graph of Figure 11. The main goal of the dense boundary band is providing a sufficient discretisation for the high velocity flow and the corresponding gradients in the boundary layer. We can use Figure 7 to estimate the required width to 0.02-0.03, based on the distance from the boundary where flow velocity reduces to half its maximum value. The estimate is confirmed with numerical results that show no improvement when increasing the boundary band width $w$ beyond a point where it covers the boundary layer flow. The effect of $w$ is less noticeable when using smaller $h_1$ as the linearly decreasing density beyond the edge of the dense band still provides a sufficient density as long as the peak of the flow.
Figure 11. Impact of refinement scheme parameter variation on the Nusselt number shown in the top and node count in the bottom graphs. Nusselt values are normed to the value attained with the best considered parameter value. *left:* Variation of the refinement ratio $h_2/h_1$ with the dense boundary band held constant at $w = 0.05$. *right:* Variation of the dense boundary band $w$ with the refinement ratio held constant at $h_2/h_1 = 10$.

is covered. We chose $w = 0.025$ as the refinement parameter for further use as it provides practically the same results as $w = 0.1$, especially at smaller $h_1$ that we are mainly interested in, while providing a significant reduction in node count as seen from the lower right graph of Figure 11.

The symmetric refinement approach is not ideal, but is suitable as a proof-of-concept due to the small number of density function parameters, which simplifies their analysis and selection. A simple improvement would be to treat the non-insulated and insulated boundaries differently, since the latter do not exhibit the sharp convective flow layer and can be discretised with a lower density. The real improvements to refinement can become arbitrarily complex and focus on deriving an error indicator based either on a previous, less refined solution or on the properties of the numerical method itself.

We use the refined density to recalculate the convergence for $Ra = 10^6$, $n = 0.6$ case that did not converge with the constant density discretisation. The new convergence results
Figure 12. *left:* Convergence study for the problematic Ra = 10^6, Pr = 100, n = 0.6 case repeated with a refined density. Red and blue horizontal lines show Nusset values derived from correlations defined in Eq. (28) and (29) while the black shows the best value from the convergence study presented in Turan et al. 13. *right:* The relative symmetry error for the repeated convergence study.

are shown in Figure 12 for the two different considered augmentation orders m. Not only is convergence achieved, but it is also achieved with a significantly smaller number of nodes and with a drastically reduced symmetry error. The computational times are shown in Table I to further emphasise that the reduction in the number of computational nodes translates into savings in computational time required. Even with the increased support size the refined discretisation still provides a solution in more than 10 times shorter time that the unrefined. The timing was performed on 4 cores of Intel(R) Xeon(R) E5520 CPU with the frequency fixed to 2.27GHz.

D. Comparison with reference data

Non-Newtonian fluid dynamics in a differentially heated cavity have already been tackled previously, and we have performed our study on a matching case for verification. We use the results provided by Turan et al. 13 and Kim et al. 12, who used FVM with SIMPLE coupling and Upwind or QUICK stabilisation. We compare those results with a refined RBF-FD and ACM, without stabilisation of convective terms to minimise the effect of numerical diffusion.
Both publications provide an empirical fit for the average Nusselt number as a function of Rayleigh number $Ra$, Prandtl number $Pr$ and non-Newtonian exponent $n$

$$\overline{Nu}_{Turan} = 0.162 Ra^{0.043} Pr^{0.341} \left( \frac{Ra^{(2-n)}}{Pr^n} \right)^{\frac{1}{2(n+1)}} \exp(C(n-1)), \quad (27)$$

$$C = \begin{cases} 
1.343 Ra^{0.065} Pr^{0.036} & \text{where } n \leq 1 \\
0.858 Ra^{0.071} Pr^{0.034} & \text{otherwise} \end{cases}, \quad (28)$$

$$\overline{Nu}_{Kim} = 0.3 n^{0.4} Ra^{\frac{1}{n+1}}, \quad (29)$$

that we can use to compare against our results.

The Nusselt values calculated by the provided correlations are added to Figure 12, supplemented by the exact numerical value from the convergence study performed by Turan et al. on this case. All results agree well, with small deviations that are normal due to discretisation errors and different numerical methods. We must also keep in mind that the provided correlations, in all their complexity, are still only empirical fits over a wide range of flow regimes and are a rather crude approximation, as can be seen from the relatively large discrepancy between the Turan et al. fit and the convergence value from the corresponding study.

We extend the fit comparison to all cases considered with the results shown in Figure 13. All computed values fall within the range of the provided correlation functions, again confirming that the meshless RBF-FD method yields valid results. The relative difference between the fits increases significantly for calmer cases with a smaller Nusselt number, casting further doubt on the accuracy of the empirical fits for a wide range of parameters. The
Figure 13. Comparison between the calculated Nusselt values and correlations provided in existing literature. Solid lines represent correlations defined in equations (28) and (29).

case-dependent discrepancy between our numerical results and the provided fits is similar to that in the source publication.

We also tested different monomial orders for the entire range of considered cases shown in Figure 13. The difference between the results computed with augmentation orders is in all cases smaller than the difference between Turan et al. and Kim et al., which means that the lower order $m = 2$ is adequate for a stable and accurate computation when the discretisation is sufficiently dense. Whether using higher order method is beneficial is case dependent as can be seen from Figure 6. Cases with lower Ra, where resolution of the boundary layer is not problematic, appear to benefit from the higher order by reaching node density independent result at a drastically lower node count. The opposite is true for the high Ra and low $n$ cases where increasing the order is at times counterproductive. Using a higher order method appears beneficial for the refined discretisation cases shown in Figure 12 but the final benefits of increasing the augmentation order are again case dependent and require additional analysis to determine optimal compromise between the lower total number of computational nodes and the increased computational time required per computational node.

We report the average Nusselt value at the cold boundary calculated with the finest discretisation for the full range of considered parameters at Pr = 100 in Table II.
Table II. A table of the best obtained average Nusselt values for all of the considered parameters. Presented values are calculated with $m = 2$. Refined density discretisation is used for the problematic $Ra = 10^6$, $n \in \{0.6, 0.7\}$ cases.

V. GEOMETRICAL FLEXIBILITY

Finally, we apply the described solution procedure to more complex domains in order to demonstrate the geometrical flexibility that we touted as one of the main benefits. In the first example, shown in Figure 14 we add circular obstructions to the central part of the De Vahl Davis case to simulate how the non-Newtonian behaviour would impact the convective flow passing through a porous filter. The computational node distribution used in this case, shown in the right panel of Figure 2, utilises a modified refinement strategy that increases node density in narrow channels and on nearby boundaries to ensure that at least two computational nodes discretise the channel’s width. Even though the dimensionless numbers defined in section II are no longer suitable for this case we stick with them to enable comparison with unobstructed flow profiles shown in Figure 3. We use the same random filter configuration for the shear-thinning non-Newtonian fluid on the left and the Newtonian fluid on the right graph of Figure 14. The results are unsurprising with both the change in temperature profile and the reduction of maximum velocity confirming that the filter’s narrow channels present a far greater hindrance for the Newtonian fluid. This can further be confirmed by comparing the average Nusselt number on the cold wall against the unobstructed case with the same $h = 0.008$ discretisation. The $Nu$ value decreases in both cases, indicating weaker convective heat transfer, but the change in the non-Newtonian $n = 0.6$ case from 35.8 to 35 is almost negligible while the $Nu$ value practically halves from 9.5 to 4.8 in the Newtonian $n = 1$ case.

The shear-thinning filter case presents a suitable opportunity to examine the spatial variation in viscosity shown in Figure 15. We show the inverse of viscosity to better highlight
Figure 14. Flow profiles for $Ra = 10^6$, $Pr = 100$ case with a porous central section. Velocity magnitude is visualised with a heat-map while the overlaid contours display the changes in temperature. The strongest shear-thinning $n = 0.6$ case is shown on the left sub-figure while the right displays the Newtonian $n = 1$ case.

the shear-thinning aspect with the most affected areas corresponding to edges of the high velocity layer next to the vertical walls and in high velocity channels through the filter as seen in Figure 14. Viscosity behaviour next to the vertical walls is identical to the non filter cases.

We formulated the method in a dimensionally agnostic manner, which can be with some effort directly transformed into dimensionally agnostic object-oriented code design, using powerful C++ template systems. The filtered 2D case can thus be quickly transformed into its 3D variant shown in Figure 16 with minimal interference in the core code that is limited only to the parameters (refer to the supplied repository for the actual code). For better visual representation plane intersections of the vertical velocity, temperature and viscosity are presented in Figure 17. The purpose of presenting this 3D results is twofold. First, we want to demonstrate the ability of the presented method to address a complex physical problem on an irregular 3D domain. Second, we want to demonstrate the powerful concept
VI. CONCLUSIONS

In this paper, we have proposed a dimension independent refined meshless solution procedure for NNC. The main advantage of the proposed approach is that it can operate on scattered nodes, which greatly facilitates the consideration of complex 3D domains and the implementation of refined discretisation, both of which are demonstrated in this paper. Moreover, the approximation weights are computed individually, allowing for a variation in the stencil size $s$, monomial augmentation order $m$, the type and order of the RBFs used, etc. The ease of adaptation on all mentioned levels allows the use of a sophisticated but
Figure 16. Natural convection in a 3D differentially heated cavity with obstructions. This case differs from the previous as the temperature differential and resulting flow is now vertical. The glyphs are sized according to the velocity magnitude in the computational node and coloured according to the temperature complying with the colourbar in Figure 17.

slow approximation only where it is truly needed with faster alternatives elsewhere, which is often the majority of the domain when dealing with realistic cases.

From the meshless method point of view, we have shown that the method is appropriate for the considered problem given that the discretisation is sufficiently dense to describe the present flow structures. We have shown how the problems with inadequate discretisation manifest themselves and how they can be resolved by refinement. We have shown that the proposed solution procedure allows for an aggressively refined discretisation that can be pushed even further if the support size is slightly increased from the usual recommendation, leading to an order of magnitude lower computational times.

From the non-Newtonian fluid dynamics point of view, we presented new results for the benchmark case that is already solved with two similar FVM approaches\cite{12,13} with a fundamentally different numerical approach and, most importantly, without stabilisation, that effectively introduces numerical diffusion in the solution.

In future work, we will expand the analysis of how the interplay between node density and stencil size influences sharp flow structures, like the boundary layer shown in this paper, and attempt to determine what are the requirements for an accurate reproduction. We will use this knowledge to develop an h-adaptive solution, which requires an adequate error indicator and a refinement logic that constructs the target node density function from the
Figure 17. Plane intersections of the 3D case shown in Figure 16. The first row displays vertical velocity, the second row the temperature and the third row the inverse of the shear-thinning viscosity. Values and positions shown on the scatter plot correspond to computational nodes within \( \frac{h}{2} \) of the intersecting plane.

error indicator data and refines the nodes accordingly.\(^{27}\)

In addition, we also plan to analyse how hyperviscosity\(^{51}\) and adaptive upwind\(^{52}\) stabilisations, and using different approximation approaches\(^{53}\) affect the flow structures.

ACKNOWLEDGMENTS

The authors would like to acknowledge the financial support of the Slovenian Research and Innovation Agency (ARIS) research core funding No. P2-0095, Young Researcher programme
PR-10468, and project funding N2-0275. Funded by National Science Centre, Poland under the OPUS call in the Weave programme 2021/43/I/ST3/00228. This research was funded in whole or in part by National Science Centre (2021/43/I/ST3/00228). For the purpose of Open Access, the author has applied a CC-BY public copyright licence to any Author Accepted Manuscript (AAM) version arising from this submission.

**DATA AVAILABILITY STATEMENT**

The code for the presented numerical methodology, plotting scripts, and data that support the findings of this study are openly available on Zenodo at [http://doi.org/10.5281/zenodo.14901942](http://doi.org/10.5281/zenodo.14901942) reference number. For ease of access, the code and data are also available in the GitLab repository, reference number.

**REFERENCES**

1. A. Rahimi, A. D. Saee, A. Kasaeipoor, and E. H. Malekshah, International Journal of Numerical Methods for Heat & Fluid Flow (2018).
2. A. Bejan, *Convection heat transfer*, John Wiley & Sons, 2013.
3. R. P. Chhabra, *Non-Newtonian Fluids: An Introduction*, pages 3–34, Springer New York, New York, NY, 2010.
4. X. Wang and X. Li, Finite Elements in Analysis and Design **46**, 551 (2010).
5. S. Charm and G. Kurland, Nature **206**, 617 (1965).
6. A. Gratão, V. Silveira, and J. Telis-Romero, Journal of Food Engineering **78**, 1343 (2007).
7. J. Welti-Chanes, F. Vergara-Balderas, and D. Bermúdez-Aguirre, Journal of Food Engineering **67**, 113 (2005), IV Iberoamerican Congress of Food Engineering (CIBIA IV).
8. E. C. Bingham, *An investigation of the laws of plastic flow*, Number 278, US Government Printing Office, 1917.
9. L. Yang and K. Du, Journal of Thermal Analysis and Calorimetry, **1** (2019).
10. J. Kwack and A. Masud, Computational Mechanics **53**, 751 (2014).
11. S. Patankar, Numerical heat transfer and fluid flow, taylor & francis london, 1980.
12. G. B. Kim, J. M. Hyun, and H. S. Kwak, International Journal of Heat and Mass Transfer **46**, 3605 (2003).
13. O. Turan, A. Sachdeva, N. Chakraborty, and R. J. Poole, Journal of Non-Newtonian Fluid Mechanics 166, 1049 (2011).

14. M. V. Bozorg and M. Siavashi, International Journal of Mechanical Sciences 151, 842 (2019).

15. N. O. Moraga, M. A. Marambio, and R. C. Cabrales, International Communications in Heat and Mass Transfer 88, 108 (2017).

16. D. A. Vasco, N. O. Moraga, and G. Haase, Numerical Heat Transfer, Part A: Applications 66, 990 (2014). 

17. D. S. Loenko, A. Shenoy, and M. A. Sheremet, Energies 12, 2149 (2019).

18. A. Alsabery, A. Chamkha, H. Saleh, and I. Hashim, Powder Technology 308, 214 (2017).

19. L. Mishra and R. P. Chhabra, Heat Transfer Engineering 39, 819 (2018).

20. G. R. Kefayati, International Communications in Heat and Mass Transfer 53, 139 (2014).

21. G.-R. Liu, Mesh free methods: moving beyond the finite element method, CRC press, 2002.

22. A. I. Tolstykh and D. A. Shirobokov, Computational Mechanics 33, 68 (2003).

23. J. Slak and G. Kosec, SIAM Journal on Scientific Computing 41, A3202 (2019).

24. W. Trojak, N. Vadlamani, J. Tyacke, F. Witherden, and A. Jameson, Computers & Fluids 247, 105634 (2022).

25. G. Kosec, Advances in Engineering Software 120, 36 (2018), Civil-Comp - Part 2.

26. T. Yasuda, I. Tanno, T. Hashimoto, K. Morinishi, and N. Satofuka, Computers & Fluids 258, 105885 (2023).

27. J. Slak and G. Kosec, International Journal for Numerical Methods in Engineering 119, 661 (2019).

28. M. Jančič, J. Slak, and G. Kosec, p-refined rbf-fd solution of a poisson problem, in 2021 6th International Conference on Smart and Sustainable Technologies (SpliTech), pages 01–06, IEEE, 2021.

29. J. D. Anderson Jr, Fundamentals of aerodynamics, Tata McGraw-Hill Education, 2010.

30. D. J. Tritton, Physical Fluid Dynamics, Oxford Science Publ, Clarendon Press, 1988.

31. The shear rate tensor for incompressible fluids is the same as the strain rate tensor, which can be expressed as the symmetric part of the velocity gradient.

32. The limit is arbitrarily chosen and has a negligible effect on the result.

33. G. De Vahl Davis, International Journal for Numerical Methods in Fluids 3, 249 (1983).

34. D. P. Hardin and E. B. Saff, Notices of the AMS 51, 1186 (2004).
The 6 monomials in 2-D case with \( m = 2 \) would be \( q = \{ 1, x, y, x^2, xy, y^2 \} \).