Particle Methods in Fluid Dynamics *

Jens Struckmeier
Department of Mathematics
University of Kaiserslautern
P.O. Box 3049
67653 Kaiserslautern
Germany

Abstract

Particle methods are numerical simulation techniques for the description of complex flow problems in fluid dynamics. They are applied to simulate rarefied gas flows based on the Boltzmann equation as well as to problems from computational fluid dynamics described by Euler or Navier–Stokes equations.
In the case of classical fluid problems, particle schemes are in particular efficient, if the problem includes complex geometries or one has to handle fluid–structure interaction. In the lecture, we will focus on some typical schemes for Euler equations, like the SPH (Smoothed Particle Hydrdynamics) method or kinetic particle schemes, which are derived from the relation between kinetic and macroscopic equations. In particular, we will demonstrate, how one may relate particle methods to classical Finite–Volume schemes and give some results on the numerical simulation including a fluid–structure interaction.

1 Introduction

Particle methods still present some kind of "exotic" applications in computational fluid dynamics, mainly for continuum flow problems. On the other hand, they are the most popular numerical schemes – even the only possible – to solve kinetic equations, like the Boltzmann transport equation in rarefied gas dynamics. In continuum flows, like the compressible Euler equations, they are not able to compete concerning accuracy with standard schemes from classical computational fluid dynamics. Nevertheless, they become a real alternative, if one has to handle complex geometries or fluid–structure interaction. Here, particle methods are mainly suited, because they use – in general – a Langrangian description of the flow and therefore are formulated without using a spatial grid. Moreover, they are straightforward to implement and very flexible concerning different types of modellizations, like multi–phase or free–surface flows.
The history of particle methods starts with the famous paper by Metropolis and Ulam [11] titled "The Monte Carlo method". Here, the authors considered a numerical method –

*to appear in the Proceedings of the 3rd summer conference on "Numerical Modelling in Continuum Mechanics", Prague 1997

based on a Monte Carlo integration – for the neutron transport equation in nuclear reactors. Later on, Harlow [6] developed the so–called Particle–in–Cell (PIC) method for the Vlasov equation in plasma physics, as well as for compressible continuum flows. The PIC method uses a Lagrangian description for the particle trajectories, where the velocities of the particles are derived by standard discretization techniques. In 1968, Bird [1] published the first paper on the Direct Simulation Monte Carlo method (DSMC) to simulate rarefied gas flows and this schemes became the most popular one in computational rarefied gas dynamics. Vortex methods for incompressible flows, as developed by Chorin [2] in the 1960's, present some other popular schemes, at least in two–dimensional simulations and moreover, they belong to a small number of particle schemes, which are directly used for incompressible equations.

In the present paper we will focuse one some particle methods for the compressible Euler equations, namely the Smoothed Particle Hydrodynamics (SPH) method originally discussed by Lucy [10] and Gingold/Monaghan [5] in the seventies, as well as on the kinetic approach for conservation laws developed first by Sanders/Prendergast [14], later by Kaniel [9] and Deshpande [3]. The SPH method was originally applied to problems from astrophysics, Sanders and Prendergast used the kinetic approach for problems from hydrodynamics.

We consider in the following the compressible Euler equations

$$\frac{\partial \Phi}{\partial t} + \nabla \cdot F(\Phi) = 0 \tag{1.1}$$

formulated in terms of the conservative quantities

$$\Phi = (\rho, \rho u, \rho e)$$

where the flux vector $F(\Phi)$ is given by

$$F(\Phi) = \begin{pmatrix} \rho u \\ \rho(u \otimes u) + p \\ (\rho e + p)u \end{pmatrix}$$

Eq. (1.1) is considered on a spatial domain $\Omega \subset I\!R^3$, where we assume that some appropriate boundary conditions are given at the boundary $\partial\Omega$.

2 Smoothed Particle Hydrodynamics (SPH)

In the following we discuss a popular particle method for compressible Euler equations (mainly used in astrophysics), namely the Smoothed Particle Hydrodynamics method (SPH) as discussed by Monaghan in Ref. The basic idea behind the SPH method is, that each flow quantity $A(t, \mathbf{x})$, like the velocity or pressure field, is smoothed by a smoothing kernel $W'(\mathbf{x}, \mathbf{x}_*)$ with respect to a measure associated to the mass density $\rho(t, \mathbf{x})$ of the flow,

$$A_s(t, \mathbf{x}) = \int_{I\!R^3} \frac{A(t, \mathbf{x}_*)}{\rho(t, \mathbf{x})} W'(\mathbf{x}, \mathbf{x}_*) \rho(t, \mathbf{x}_*) \mathrm{d}\mathbf{x}_* \tag{2.1}$$

Then, the smoothed quantities are approximated by a set of Lagrangian particles, similar to a Monte–Carlo integration of the integral term appearing in (2.1),

$$A_s(t,\mathbf{x}) \approx \sum_{i=1}^{n} m_i \frac{A_i}{\rho_i} W'(\mathbf{x},\mathbf{x}_i) \tag{2.2}$$

Here, the particles are located at the spatial positions $\mathbf{x}_i(t), i = 1,\ldots,n$ and, as Lagrangian particles, they are moving along the trajectories $\dot{\mathbf{x}}_i = \mathbf{v}_i$, where $\mathbf{v}_i(t)$ denotes the velocity of the i–th particle.
The smoothing kernel $W'(\mathbf{x},\mathbf{x}_*)$ is in general radial symmetric, because there are no preferred directions in the smoothing procedure, i.e.

$$W'(\mathbf{x},\mathbf{x}_*) = W(\|\mathbf{x}-\mathbf{x}_*\|,h) = \frac{1}{h^d} f\left(\frac{r}{h}\right) \tag{2.3}$$

where h denotes the so–called smoothing length, defining the decay of the kernel with increasing radial distance $r = \|\mathbf{x}-\mathbf{x}_*\|$. The function f in (2.3) denotes a shape function (d is the spatial dimension) and from a theoretical point of view, the best choice for the shape function seems to be a Gaussian kernel. This kernel is reproduced by differentiation and the Fourier transform itself is again Gaussian, which simplifies the stability analysis of the SPH method. In practice, it is more appropriate to use kernels with compact support – otherwise one has to consider a long–range interactions between single particles. For a detailed investigation on various SPH–smoothing kernels we refer the reader to Ref. [4].
In the case of an isentropic flow, i.e. $P = P(\rho)$, the equations of motion in the SPH method are given by

$$\dot{\mathbf{x}}_i = \mathbf{v}_i \tag{2.4}$$

$$\dot{\mathbf{v}}_i = -\sum_{j=1}^{n} m_j \left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2}\right) \nabla W_{ij} \tag{2.5}$$

where each particle carries the information $(m_i, \mathbf{x}_i(t), \mathbf{v}_i(t))$. Moreover, the density field is obtained via the interpolation formula defined by (2.2), i.e.

$$\rho_i = \sum_{i=1}^{n} m_j W_{ij} \tag{2.6}$$

The right hand side of (2.5) defines the particle interactions by a force term F_{ij}

$$F_{ij} = m_i m_j \left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2}\right) \nabla W_{ij}$$

between the i–th and j–th particle. This is the reason, why – in practice – the Gaussian kernel is substituted by a smoothing kernel with compact support: to reduce the number of particle interactions to the order $O(n)$.
Eq. (2.5) as well as (2.6) are not unique in SPH, in the sense that there exist some other representations for the equations of motion in SPH [12]. E.g., the calculation of a particle

density ρ_i by the interpolation rule (2.6) is referred as "density by summation" and an alternative way to obtain the density of a single particle is the so–called "density by continuity equation": this approach uses an additional evolution equation of the form

$$\frac{\mathrm{d}\rho_i}{\mathrm{d}t} = \sum_{i=1}^{n} m_j(\mathbf{v}_i - \mathbf{v}_j)\nabla W_{ij} \tag{2.7}$$

where ρ_i denotes the density of the i–th particle. Formally, both methods are equal, if the initial condition for Eq. (2.7) is consistent with the interpolation rule, but due to numerical integration of (2.7), the results will vary using (2.7) instead of 2.6).
In the general case, i.e. non–isentropic flows, the system given by Eqs. (2.4) and (2.5) is enlarged by an evolution equation for the thermal energy u_i of a particle and this equation is given by

$$\dot{u}_i = \frac{P_i}{\rho_i} \sum_{j=1}^{n} m_j(\mathbf{v}_i - \mathbf{v}_j)\nabla W_{ij} \tag{2.8}$$

The equations of motion of the SPH method are numerically integrated by standard time–integration schemes for ordinary differential equations. The most popular scheme – typically used in SPH applications – is the so–called modified Euler scheme (often referred as a predictor–corrector scheme): denoting by $\mathbf{X}(t)$ the vector consisting of the discrete quantities in the SPH formulation, the equations of motion may be written in the form

$$\dot{X} = F(X, t) \tag{2.9}$$

where f denotes the right hand sides of (2.4), (2.5) and (2.7), (2.8), respectively.
Then, the modified Euler scheme reads

$$\mathbf{X}_{j+1/2} = X_j + \frac{\Delta t}{2} F(\mathbf{X}_j, t_j) \tag{2.10}$$

$$\mathbf{X}_{j+1} = X_j + \Delta t F(\mathbf{X}_{j+1/2}, t_{j+1/2}) \tag{2.11}$$

Here, the indices j and $j+1$ denote the states at the discrete times t_j and $t_{j+1} = t_j + \Delta t$ and $j + 1/2$ corresponds to a common predictor step. In practice, the evaluation of the right hand side of (2.10) is substituted by the term $F(\mathbf{X}_{j-1/2}, t_{j-1/2})$, which is mor efficient, because one needs to evaluate the right hand side of (2.9) only ones, but this scheme yields the same order of approximation as the original one.
One can show, that the stability of the scheme given above is nearly identical to that of a more sophisticated Leapfrog or high–order Runge–Kutta scheme. The time integration yields a typical CFL–condition for explicit schemes applied to hyperbolic systems. This CFL–condition needs to be modified by the artificial viscosity term, which is introduced to handle shock solutions.
Like other numerical schemes for the compressible Euler/Navier–Stokes equations, the SPH method gets into problems in the low–Mach number limit. The equations of motions (2.4), (2.5) become a stiff system, because in the equations for the velocities of the SPH particles appears the inverse Mach number on the right hand side of the equation. Applying an explicit time integration yields extremely small time steps and there is still a need for better integration techniques (even some modified SPH formulations) in the low–Mach

number limit.
In order to treat shocks with the SPH method, it is necessary to add an artificial viscosity term in the evolution equations for the particle velocities, i.e.

$$\dot{\mathbf{v}}_i = -\sum_{j=1}^{n} m_j \left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2} + \Pi_{ij} \right) \nabla W_{ij}$$

where Π_{ij} denotes the artificial viscosity. Numerical experiments showed, that the standard viscosity terms like the bulk or the von Neumann–Richtmyer viscosity are not appropriate viscosity terms to be used in the SPH method. Hence, Monaghan proposed a so–called viscous "pressure" term, which is based on a model for real molecular interactions,

$$\Pi_{ij} = \begin{cases} \frac{1}{\rho_{ij}}(-\alpha h c_{ij} \mu_{ij} + \beta h^2 \mu_{ij}^2) & : \text{ if } \mu_{ij} \leq 0 \\ 0 & : \text{ else} \end{cases} \tag{2.12}$$

and

$$\mu_{ij} = \frac{(\mathbf{v}_i - \mathbf{v}_j)(\mathbf{x}_i - \mathbf{x}_j)}{\|\mathbf{x}_i - \mathbf{x}_j\|^2 + \varepsilon h^2}$$

The additional term εh^2 in the denominator prevents the term μ_{ij} to be singular as $\mathbf{x}_j \to \mathbf{x}_i$. Moreover, the artificial viscosity term (2.12) contains two more fit parameter a and b, which are determined (as well as the value for ε) by numerical experiments, e.g., $(\alpha, \beta) = (1, 2)$ and $\varepsilon = 0.01$ for problems from gas dynamics (to treat high Mach number shocks) and $(\alpha, \beta) = (0.1, 0)$ and $\varepsilon = 0.01$ for problems from hydrodynamics.

3 A Finite–Volume Particle Method

In the last section we discussed the SPH method and one of the drawbacks of this scheme lies in the artificial viscosity term, which is added in the momentum equation, in order to treat shocks. The SPH viscosity term is derived heuristically, trying to model real molecular interactions, and includes some fit parameter obtained from numerical experiments.
One possibility to get rid of the artificial viscosity term is to reformulate the SPH method as a Finite–Volume method, where the Lagrangian particles play the role of a (moving) "grid structure" and to replace the artificial viscosity by some known numerical flux functions from classical CFD. This approach is presented in the following.
We consider a general system of conservation laws written as

$$\partial_t \Phi + \nabla \cdot F(\Phi) = 0, \tag{3.1}$$

where the vector $\Phi = \Phi(t, \mathbf{x})$ denotes the conservative quantities and $F(\Phi)(t, \mathbf{x})$ the corresponding flux function. Eq. (3.1) is considered on some spatial domain $\Omega \subset I\!R^3$, where we assume for the following, that some appropriate boundary conditions are given at the boundary $\partial\Omega$. Applying a classical Finite–Volume scheme to the system given in (3.1), one considers a set of discrete conservation laws in the form

$$V_i \frac{\mathrm{d}\Phi_i}{\mathrm{d}t} = -\sum_{j \in N(i)} \beta_{ij} H(\Phi_i, \Phi_j, n_{ij}) \tag{3.2}$$

where the quantities Φ_i denote the averages of the conservative variables Φ over a cell ν_i with volume V_i. Moreover, the function $H(\Phi_i, \Phi_j, n_{ij})$ denotes the numerical flux function, which is used instead of the exact flux vector $F(\Phi)$. The coefficients β_{ij} on the right hand side denote some grid structure coefficients, typically $\beta_{ij} = |S_{ij}|n_{ij}$, where $|S_{ij}|$ denotes the length of a common edge of the two cells ν_i and ν_j and n_{ij} the corresponding normal vector along the edge ($N(j)$ denotes the index set for the neighbouring cells of ν_i). Formally, the discrete conservation laws are obtained by testing the conservation laws (3.1) against a special set of test functions $\Psi_i(\mathbf{x})$, where $\Psi_i(\mathbf{x})$ is the characteristic function over the cell ν_i, i.e.

$$\Psi_i(\mathbf{x}) = \mathbb{1}_{\nu_i}(\mathbf{x})$$

Hence, the grid structure coefficients may even be expressed in the form

$$\begin{aligned} \beta_{ij} &= \int (\Psi_i \nabla \Psi_j - \Psi_j \nabla \Psi_i) \mathrm{d}\mathbf{x} \\ n_{ij} &= \beta_{ij}/|\beta_{ij}| \end{aligned}$$

With the same idea, i.e. testing the conservation laws against a special class of test functions, one may derive a new class of equations of motion for (moving) Lagrangian particles. This is achieved by substituting the characteristic function $\mathbb{1}_{\nu_i}$ over a cell ν_i by some smoothing function $\Psi_i(t, \mathbf{x})$ related with the spatial positions $\mathbf{x}_i(t)$ of the Lagrangian particles.
In particular, we define a new set of test functions $\Psi_i(t, \mathbf{x})$ by

$$\begin{aligned} \Psi_i(t, \mathbf{x}) &= \frac{W_i(t, \mathbf{x})}{\sigma} \\ W_i(t, x) &= m_i W(\mathbf{x} - \mathbf{x}_i(t)) \\ \sigma(t, x) &= \sum_{j=1}^{n} W_j(t, \mathbf{x}) \end{aligned}$$

where the shape function $W(\mathbf{x} - \mathbf{x}_i(t))$ corresponds, e.g., to a generic smoothing kernel used in the SPH formulation. Introducing this kind of smoothing functions, each particle has a (time–dependent) volume $V_i(t)$, given by

$$V_i(t) = \int \Psi_i(t, \mathbf{x}) \mathrm{d}\mathbf{x}$$

Moreover, it is straightforward, that the Ψ_i–functions give a partition of the unity for each $t \geq 0$ and $\mathbf{x} \in \Omega$,

$$\sum_{i=1}^{n} \Psi_i(t, \mathbf{x}) = 1 \quad \forall\, t \geq 0, \mathbf{x} \in \Omega$$

and we have (formally) the normalization condition

$$\sum_{i=1}^{n} V_i = |\Omega|$$

Now we integrate the system of conservation laws with respect to the new set of test functions, i.e.

$$\int_\Omega \Psi_i(\partial_t \Phi + \nabla \cdot F(\Phi))\mathrm{d}\mathbf{x} = 0, \quad i = 1, \ldots, n$$

which yields the time evolution

$$\frac{\mathrm{d}}{\mathrm{d}t}(V_i \Phi_i) = \int \Psi_i \nabla \cdot F(\Phi)\mathrm{d}\mathbf{x} - \int \frac{\partial \Psi_i}{\partial t}\Phi \mathrm{d}\mathbf{x} \tag{3.3}$$

where the discrete quantities are defined by

$$\begin{aligned} \Phi_i &= \frac{1}{V_i}\int_\Omega \Phi \Psi_i(t, \mathbf{x})\mathrm{d}\mathbf{x} \\ V_i &= \int \Psi_i(t, \mathbf{x})\mathrm{d}\mathbf{x} \end{aligned}$$

Remark 3.1 The second term on the right hand side of Eq. (3.3) denotes the additional term due to (moving) particles.

Straightforward computations yield the formula

$$\frac{\partial \Psi_i(t, \mathbf{x})}{\partial t} = \sum_{k=1}^{n} \left(\dot{\mathbf{x}}_k(t)\Psi_i(t, \mathbf{x})\frac{\nabla W_k(\mathbf{x})}{\sigma(t, \mathbf{x})} - \dot{\mathbf{x}}_i(t)\Psi_k(t, \mathbf{x})\frac{\nabla W_i(\mathbf{x})}{\sigma(t, \mathbf{x})} \right)$$

which shows that

- this approach coincides with the classical Finite–Volume method, if the particles are not moving, i.e. $\dot{\mathbf{x}}_i = 0$ and $\Psi_i = \mathbb{1}_{\nu_i}$,
- for Lagrangian particles, i.e. $\dot{\mathbf{x}}_i = \mathbf{v}_i$, $i = 1, \ldots, n$, the additional term in the Finite–Volume method is given by

$$\int \frac{\partial \Psi_i}{\partial t}\Phi \mathrm{d}\mathbf{x} = \sum_j (v_i \int \Phi \mathrm{d}\mathbf{x} - v_j \int \Phi \mathrm{d}\mathbf{x})$$

Applying Lagrangian particles, yields a time evolution for the discrete quantities, which includes the standard Finite–Volume term, i.e.

$$\int \Psi_i \nabla \cdot F(\Phi)\mathrm{d}\mathbf{x} \approx -\sum_j \beta_{ij} H(\Phi_i, \Phi_j, n_{ij})$$

(including a numerical flux function) and some additional term given by

$$\int \frac{\partial \Psi_i}{\partial t}\Phi \mathrm{d}\mathbf{x} \approx \sum_j (\gamma_{ji} v_i \Phi_j - \gamma_{ij} v_j \Phi_i)$$

Now, we approximate both terms like in the classical Finite–Volume method to obtain the equations of motion of the Finite–Volume particle method in the form

$$\frac{\mathrm{d}}{\mathrm{d}t}(V_i \Phi_i) = -\sum_j \beta_{ij} H(\Phi_i, \Phi_j, n_{ij}) + \sum_j (\gamma_{ij} v_j \Phi_i - \gamma_{ji} v_i \Phi_j)$$

Here, the (particle) geometry coefficients are defined by

$$\gamma_{ij} = \int \frac{W_i \nabla W_j}{\sigma^2} \mathrm{d}\mathbf{x}, \quad \beta_{ij} = \gamma_{ij} - \gamma_{ji} \tag{3.4}$$

Remark 3.2 In order to compute these coefficients, it is obvious that in practice one should use appropriate quadrature formula.

Summarizing the previous results, we obtained equations of motions for a combined Finite–Volume particle method in the form

$$\frac{\mathrm{d}}{\mathrm{d}t}(V_i \Phi_i) = -\sum_j \beta_{ij} H(\Phi_i, \Phi_j, n_{ij}) + \sum_j (\gamma_{ij} v_j \Phi_i - \gamma_{ji} v_i \Phi_j) \tag{3.5}$$

together with coefficients β_{ij} and γ_{ij} given by 3.4.
Eq. (3.5) describes the dynamic behaviour for the discrete products $V_i \Phi_i$, whereas on the left hand side appear the averaged quantities Φ_i separately. Hence, we need to close Eq. (3.5) by an additional evolution equation for the volumes $V_i(t)$ of the particles. This is achieved by introducing the trivial conservation law $\partial(1)/\partial t = 0$, which yields in the weak form (i.e. testing against the Ψ_i–functions) a simple evolution equation for the volumes $V_i(t)$.
The main advantage of this new approach is that the artificial viscosity in the original SPH method is substitued by a numerical flux function, like in classical Finite–Volume methods. In Chapter 5, we will give some preliminary results on a shock tube problem, where the van Leer flux is used as numerical flux function. Finally it should be mentioned, that one might even develop a mixed Euler–Lagrange particle method using the Finite–Volume formulation presented above. Some further details and results on this new approach will be published elsewhere [7].

4 Kinetic Particle Methods

The idea behind Kinetic Particle Methods (KPM) is to use a microscopic description of fluid flows for the calculation of macroscopic quantities describing a continuum fluid. In transitional flows, there are two different possibilities to describe the time evolution of fluids: in low–density regions, one uses a kinetic density function $f = f(t, \mathbf{x}, \mathbf{v})$ for a microscopic picture of a (gas) flow and the time evolution is given by the nonlinear Boltzmann transport equation

$$\partial_t f + \mathbf{v} \cdot \nabla_x f = \frac{1}{\varepsilon} Q(f), \quad f = f(t, \mathbf{x}, \mathbf{v}) \tag{4.1}$$

where ε denotes a small parameter proportional to the mean free path of the gas. If the parameter ε in front of the collision operator $Q(f)$ tends to zero, i.e. in the limit of a vanishing mean free path, one obtains equations for the conservative quantities Φ of the collision operator

$$\partial_t \Phi + \nabla_x \cdot F(\Phi) = 0, \tag{4.2}$$

which yields the Euler equations for continuum flows. The conservative quantities are related with the kinetic density function via the relation

$$\Phi = \int (1, \mathbf{v}, \mathbf{v}^2/2) f(t, x, \mathbf{v}) \mathrm{d}\mathbf{v} \tag{4.3}$$

Formally, the continuum limit is obtained applying an asymptotic expansion of the kinetic density,

$$f(t,x,v) = \sum_{n=0}^{\infty} \varepsilon^n f_n(t,x,v)$$

as ε tends to zero and the zeroth order term yields the local Maxwellian function

$$f_0(t,x,v) = f_M(\mathbf{v}, \Phi)$$

and substituting the Maxwellian distribution into (4.3) exactly yields the conservative variables described by the Euler equations.
Hence, with the help of well–defined particle schemes for the Boltzmann transport equation, it is straightforward to derive a corresponding particle scheme for the compressible Euler equations.
Particle methods for the Boltzmann transport equation use an approximation of the kinetic density function by a discrete set of particles located in the phase space, i.e.

$$f(t,\mathbf{xv}) \sim \frac{1}{n} \sum_{i=1}^{n} \delta(\mathbf{x} - \mathbf{x}_i(t)) \delta(\mathbf{v} - \mathbf{v}_i(t))$$

Remark 4.1 Concerning the theoretical foundation of particle schemes for the Boltzmann equation as well as some applications, we refer the reader to Refs. [1] and [13].

To derive the time evolution of the kinetic particles, one applies a splitting method over small time increments, in order to separate the free–streaming of particles from binary collisions defined by the collision operator $Q(f)$. Macroscopic quantities, like the density and pressure field, are recovered at each discrete time step of the splitting method, using a typical interpolation formula, e.g.

$$\rho(t,\mathbf{x}) = \frac{1}{n} \sum_{i=1}^{n} W(\mathbf{x}, \mathbf{x}_i(t)) \tag{4.4}$$

where W denotes an appropriate smoothing kernel.
The first step in the splitting method, the free–streaming of particles, is simply governed by the equations

$$\mathbf{x}_i(t+\Delta t) = \mathbf{x}_i(t) + \Delta t \mathbf{v}_i(t), \quad \bar{\mathbf{v}}_i(t) = \mathbf{v}_i(t) \tag{4.5}$$

whereas in the second step, one resamples the velocities according to binary collisions,

$$(\mathbf{v}_i(t+\Delta t))_{i=1,\ldots,n} = T((\bar{\mathbf{v}}_i(t))_{i=1,\ldots,n})$$

Here, the operator T is derived applying a standard time–discretization to the space–homogeneous Boltzmann equation

$$\frac{\partial f}{\partial t} = \frac{1}{\varepsilon} Q(f) \tag{4.6}$$

As mentioned above, it is straightforward to derive a similar kinetic particle method for the compressible Euler equations and this may be seen directly from Eq. (4.6). In the limit $\varepsilon \to 0$, the solution of (4.6) will instantaneously relax to a local Maxwellian distribution

with macroscopic quantities obtained from the interpolation rule (4.4) applied to the particle positions and velocities defined by the free–streaming step (4.5). Hence, the collision process in a rarefied gas flow is substituted ba a so–called projection phase (particles are resampled according to a given local Maxwellian distribution).
This kind of Kinetic Particle Method for the compressible Euler equations may be applied to more general systems of conservation laws and is not restricted to the compressible Euler equations only. Moreover, in the case of the Euler equations, one may even substitute the local Maxwellian distribution used in the projection phase by some "artificial" equilibrium functions, which are much more easy to handle in numerical simulations. For a detailed description we refer the reader to the work done by Junk [8]. In the case of a general system of conservation law (for simplicity in 1D),

$$\frac{\partial \Phi}{\partial t} + \frac{\partial}{\partial x} F(\Phi) = 0$$

one has to solve to problem of "realization of moments", i.e. to find a function $G : I\!R^n \times I\!R \to I\!R^n$, $G = G(\Phi, v)$, which satifies the two integral identities (consistency relation)

$$\int G(\Phi, v) \mathrm{d}v = \Phi$$

and

$$\int v G(\Phi, v) \mathrm{d}v = F(\Phi)$$

Returning again to the Euler equations, the problem of "realization of moments" may be solved using the Ansatz

$$G(\Phi, v) = (1, v, v^2/2) f(\Phi, v)$$

for the scalar function $f(\Phi, v)$ and

$$\Phi = (\rho, \rho u, E), \quad F(\Phi) = (\rho u, \rho u^2 + p, (E + p)u)$$

with

$$p = \rho T, \quad T = 2\left(\frac{E}{\rho} - \frac{1}{2}u^2\right)$$

If we further introduce a shape function f^* by

$$f(\Phi, v) = \frac{\rho}{\sqrt{T}} f^*\left(\frac{v - u}{\sqrt{T}}\right)$$

then the Maxwellian distribution is expressed in the form

$$f^*(v) = \frac{1}{\sqrt{2\pi}} \mathrm{e}^{-v^2/2}$$

and this function trivially fulfills the consistency relations defined above. Kaniel [9] proposed a shape function, which is a special interest because it has a compact support and moreover very easy to handle in numerical simulations,

$$f^*(v) = \frac{1}{2\sqrt{3}} \mathbb{1}_{[-\sqrt{3}, \sqrt{3}]}(v)$$

Finally, Sanders and Prendergast [14] constructed q weighted sum of Dirac deltas,

$$f^*(v) = a\delta(v - v_0) + b\delta(v) + a\delta(v - v_0)$$

which fulfill the consistency relations using appropriate values for a, b and v_0.