Available online at www.sciencedirect.com

ScienceDirect

Procedia CIRP 40 (2016) 443 – 448

www.elsevier.com/locate/procedia

13th Global Conference on Sustainable Manufacturing - Decoupling Growth from Resource Use

CFD-Simulations In The Early Product Development

Krotil Stefan[a*], Reinhart Gunther[a]

[a] *Fraunhofer IWU Project Group RMV, Beim Glaspalast 5, 86153 Augsburg, Germany*

* Corresponding author. Tel.: +49-821-56883-76; fax: +49-821-56883-50. *E-mail address:* stefan.krotil@iwu.fraunhofer.de

Abstract

A growing range of products shifts the product development in focus. Especially the early design phase defines the costs and quality. Thus, to ensure the quality of the products, different concepts are designed and evaluated. A tool for the evaluation of the different designs is simulation due to being fast and cost efficient. But simulation in the early phase still remains a problem. This results of the basic and fast changing concepts, which take too much effort to be converted into a simulation model. To get around these circumstances, we propose a method for an online modelling in the design process. Instead of defining the simulation model at once, the method modificates the simulation model during simulation run time. Furthermore, the functionality is shown in a CFD simulation using the meshless Smoothing Hydrodynamics method. Using the GPU to achieve real-time simulation, the method allows a WYSIWYG design-simulation development helping to raise the quality of products.

Peer-review under responsibility of the International Scientific Committee of the 13th Global Conference on Sustainable Manufacturing

CFD; development; lagrangian; Smoothed Particle Hydrodynamics; real-time

1. Advantages of Simulation in Product Development

Shorter product life cycles as well as a growing range of product varieties shift the product development more and more into focus. Furthermore, multidisciplinary in development leads to a gaining dependency between different engineering disciplines like mechanics, electrics and informatics [1].

Nomenclature

a, b	fluid particles
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
CPU	Central Processing Unit
GPU	Graphics Processing Unit
h	smoothing length
m	mass
p	pressure
ρ	density
r	distance
v	velocity
x	position
W	weighting factor
WYSIWYG	What-You-See-Is-What-You-Get

Special attention gets the design phase of a product development, which defines the basic functionality and shape. In addition, in this phase significant decision about quality and cost are made [2]. In order to support these determination a validation and verification of different alternatives can help to reduce the risk of wrong decisions and thus, reduce economic risk and raise sustainably of products [3].

Simulation has been established as a helpful tool. An example is the cheaper setup for experiments compared to real experimental setups take a lot of effort to build and are cost-intensive. In addition, observability of critical processes as well as the breakdown into simpler segments is an advantage of simulation [4].

Peer-review under responsibility of the International Scientific Committee of the 13th Global Conference on Sustainable Manufacturing
doi:10.1016/j.procir.2016.01.090

2. Simulation of Fluids

The simulation of the behavior of fluids is realized by Computational Fluid Dynamics (CFD). A CFD process can be divided into four phases, the construction of the geometry (CAD model), the synthesis of the computational domain, the actual run of the simulation and the analysis of the results after the post-processing [5]. A new simulation run always requires to execution of all steps. As CFD computation takes a lot of effort, the whole process can take hours to days. Thus, most often the use of CFD simulations is limited to special cases like automotive or aerospace dynamics [6].

2.1. Methods in CFD

As CFD is still a challenge, there are many different theories, which can be clustered in eulerian- and lagrangian approaches.

The commonly used and well researched eulerian approach is based on a predefined, fixed mesh, which covers the whole computational domain [7]. The granularity and quality of the mesh discretization is a critical part of the precision of the solution and the computation time [8]. Thus, based on the granularity of discretization different methods are commonly used. Methods used in science are the Direct Numerical Simulation, which needs the finest discretization [9] or the Large Eddy Simulation [10]. As the computation time still remains very long, industry mainly uses the Reynolds-Averaged-Navier-Stokes method [11].

In contrast to the eulerian methods lagrangian approaches discretize the fluid itself with particles, which can move around. This group of approaches experiences a quite new but fast gaining interest in the scientific community [12] and can help to meet today's challenges. A main reason is that some problems appearing in the eulerian discretization can be easily handled. For example, the separation of two fluids in a multiphase simulation can be modelled easier [13]. Another advantage is the lack of the need for regeneration of the mesh quality and the easier modelling of boundary conditions [14, 15] as they can be handled like collisions [16, 13]. Furthermore, each particle in each time step can be computed independently to each other. Therefore, the algorithms of lagrangian approaches can be computed highly parallel [17]. With evolving computer technology towards parallelization [18] this ability can lead to a critical property.

An example for the meshless lagrangian simulation method is the Smoothed Particle Hydrodynamics method, which was initially developed as a solution method for astrophysical problems [19, 20]. The method, based on a kernel density estimation, is a statistical method, having a lower error than pure statistical functions, e. g. Monte-Carlo methods [21]. On the basis of the solution method for astrophysical problems, the method was adapted to fluid dynamic problems with free surfaces. This evolution is known as the weakly compressible SPH [22]. On this basis, for further advancements focusing the precision [23] or the performance up to real-time [24] were made [25]. Furthermore, the method itself is validated [26].

The weakly compressible SPH method is a lagrangian method, in which each virtual particle caries physical parameters like mass m or velocity v and additional information like its position x. Thus, the fluid dynamic problem using the incompressible Navier-Stokes equation can be solved stap by step using kernel approximation. For example, the density of a location a can be evaluatied solving

$$\rho_a = \sum m_b W_{ab}, \tag{1}$$

where W_{ab} denotes

$$W_{ab} = W(r_{ab}, h), \tag{2}$$

with h being the smoothing length and r_{ab} the distance between the particles a and b. Furthemore, the complex pressure gradient term of the Navier Stokes equation can be obtained using

$$-\left(\frac{1}{\rho}\nabla p\right)_a = -\sum m_b \left(\frac{p_a}{{p_a}^2} + \frac{p_b}{{p_b}^2}\right)\nabla_a W_{ab}, \tag{3}$$

where p_a is the pressure at particle a and ∇_a the gradient with respect to the position a. [26]

2.2. Preparation of the computational domain

Either in eulerian or lagrangian methods the preparation and precomputation of the computational domain is quite similar (Figure 1). The first step is the extraction of a CAD model. This is followed by the mesh generation. In the eulerian approaches, the whole fluid domain gets meshed having a set of different strategies and tools [8]. In contrast, the lagrangian approaches triangulate the CAD-model [14] and does not limit the fluid domain to boundaries. In both cases this step is the most critical one. On the one hand a too dense mesh or triangulation leads to high computation times, on the other hand a too coarsely granular discretization leads to errors based on low resolution.

This step is followed by the selection of physics and the description of fluid properties. The last step is the definition of the boundary conditions [18].

3. Simulation of Fluids

Utilizing its advantages, Lagrangian methods can help to resolve the limitation to special cases and help to extend the field of application, for example a simulation of a filling process in an industrial application.

But nowadays, the preparation of the computational domain can take up to 50% of the whole simulation cycle

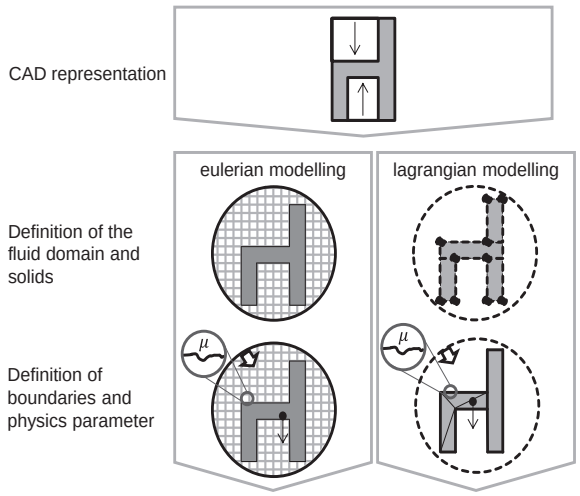

Figure 1: General process of a simulation model generation in eulerian and lagrangian discretisation

[27]. Thus, reducing the modelling time and effort helps to increase the viability.

Therefore, we introduce an online generation of CFD domains with focus on the update of the geometry. The reduction of effort and cost comes along in two dimensions. The first dimension is the decrease of work for the user as a computational domain does not need to be generated totally. In contrast, it can evolve during the simulation run. The second advantage of the online simulation is the optimization of the geometry to reduce the computational effort based on better collision resolution.

3.1. Concept

In order to achieve a fast online model generation for lagrangian CFD simulation we propose four steps. Based on the modelling process of rigid body dynamics [28] the method can be divided in four steps (Figure 2). Beginning with an common known CAD model a triangulated mesh gets prepared. After that a collision model is solidificated an gets paramerized in order to get a phyiscs model. Last but no least the physics model gets enriched with information about its kinematic behavior.

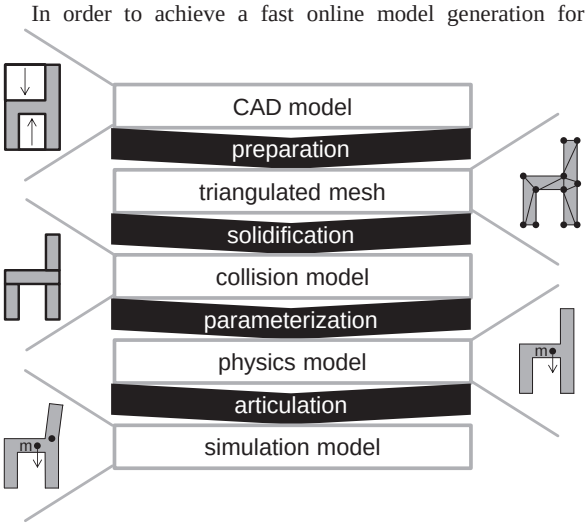

Figure 2: Process of the generation of the geometric simulation model

3.2. Preparation

The first step is the conversion of the CAD input data into a triangulated mesh. The preparation can be divided into two information blocks – the hierarchy of the input model and the master concepts. The hierarchy tree only refers to unique master concepts. A master concept includes the description of different instances, which can differ in location and rotation, but not in scale. The constant scale is necessary as on the second block each master concept gets tessellated into a triangular mesh (and the error would scale). In order to reduce model errors, the discretization error should be smaller than the designated simulation resolution.

3.3. Solidification

On the basis of the model hierarchy and the triangulated master concepts the most significant step for the future computational effort is the solidification. The collision model defines the complexity and thus the effort put into collision resolution following the concept, that convex aggregated objects need less computational effort than checking each triangle.

As the model can change online during a simulation, the first step is to check differences between the existing and new geometry information in the dimensions hierarchy and master concepts. Within the next steps only new or modified master concepts are computed. The other ones can remain constant in the simulation.

Due to convexity having a great influence on computational effort, each master concept convexity gets checked. If the master concept is already convex, the collision model can be computed based on the surface of each triangle. In the other cases the master concept is decomposed into convex parts following known algorithms [28-31]. Furthermore, as online simulation needs a fast adaption of the computational domain, each triangular plane of the master concept's mesh gets evolved into a collision model. These collision models remain in the simulation as an interim solution until the convex decomposed collision model is computed.

3.4. Parameterization

In the next step the generated collision models gets enriched with the simulation-relevant parameters like mass, center of gravity or surface properties [27, 32]. The relevant data can be either extracted out of the CAD model or

extracted out of a database, for example a product data management system.

3.5. Articulation

Last but not least the kinematics of the physics model gets described. Kinematics can be subdivided into two types.

The first type is passive kinematics like links or joints. This context gets defined in the model hierarchy on the basis of the restrictions made in the CAD model or manual adjustments.

The second type is active kinematics like motors or fixed motion. Due to having online adapted simulation models, the generation of motion has to follow the same concepts. Thus, fixed motion gets defined with an interactive motion planer. Thus the motion of parts can be changed online. As the motion sticks on hierarchy parts with an unique identification it still remains constant even if the model changes.

4. Use Case and Test Result

The described method of an online CFD simulation with changing geometry information was carried out successfully. The lagrangian method used in this use case was the Smoothed Particle Hydrodynamics [13, 16, 33], which was executed on the Graphics Processing Unit (GPU). Furthermore, the CFD-domain was prepared on the Central Processing Unit (CPU) and afterwards pushed onto the GPU. This way, the workload was balanced over the whole workstation and a real-time simulation of the fluid processes was possible.

4.1. Performance measurement

Figure 3 shows the test geometry modelled in a CAD-tool and updated online in the simulation. Each part was individually constructed and added successfully to the simulation domain without any restart. On this basis the effect of the derivation step was analyzed. To review the effect of the two-step solidification the exact influence of the convexity was analyzed. Furthermore, the decomposition was executed in an integral way. Thus, the impact of a partial or approximate convex decomposition could be evaluated. The whole process of updating the geometry online takes several milliseconds. Thus, leaving out the convex decomposition, the model can be updated without an interruption of the user. As convexity is crucial for the simulation performance, the derivation performs a convex decomposition.

Figure 4 shows the dependency of the number of fluid particles and the shape (convexity) of the geometry with 1 being fully concave and 12 being fully convex. Thus, the simultaneous convex decomposition of the master concept has a great impact on the computational effort and is needed to accomplish an online simulation for bigger models. Having a full convex decomposed geometry the computational time in our example utilizing a Nvidia Quadro 2000K could be

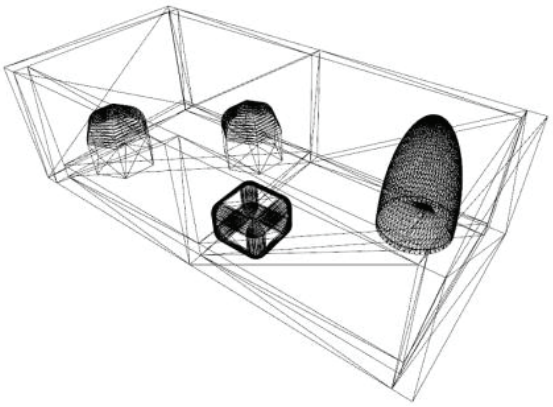

Figure 3: Test geometry of the online simulation

reduced up to 35.6% of the concave mesh without any loss of solution precision in the lagrangian method. Thus, in our example the computation time was redurced from 12.1 ms to 4,3 ms using 1 000 000 fluid particles.

4.2. Use case of WYSIWYG-CFD-Simulation

In addition, Figure 5 shows a possible development process utilizing the WYSIWYG-approach with an online-modification of the geometries. In this case, divided into 10 steps, on the left side the normal CAD-based development process of a concept is shown. In addition, on the right side, the simulation process is shown. In history, after each modification of the geometry, a new simulation run had to be started. Using the WYSIWYG-approach, the simulation model is synchronous to the CAD-concept.

Thus, shown, in step 1, a fluid source has to be created and the simulation gets started. In addition, the design of a simple filling process gets performed with a first CAD-concept of a

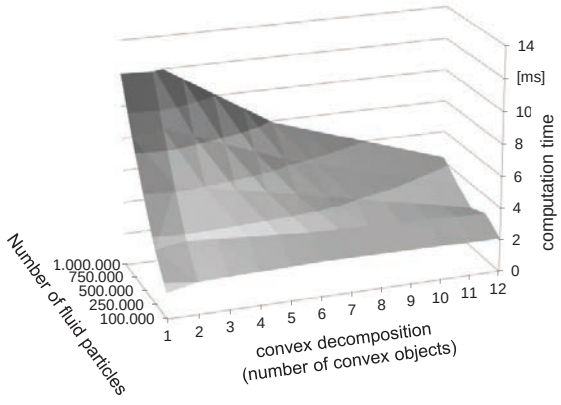

Figure 4: Computation time of the geometry handling with changing number of fluid particles and grade of convex decomposition

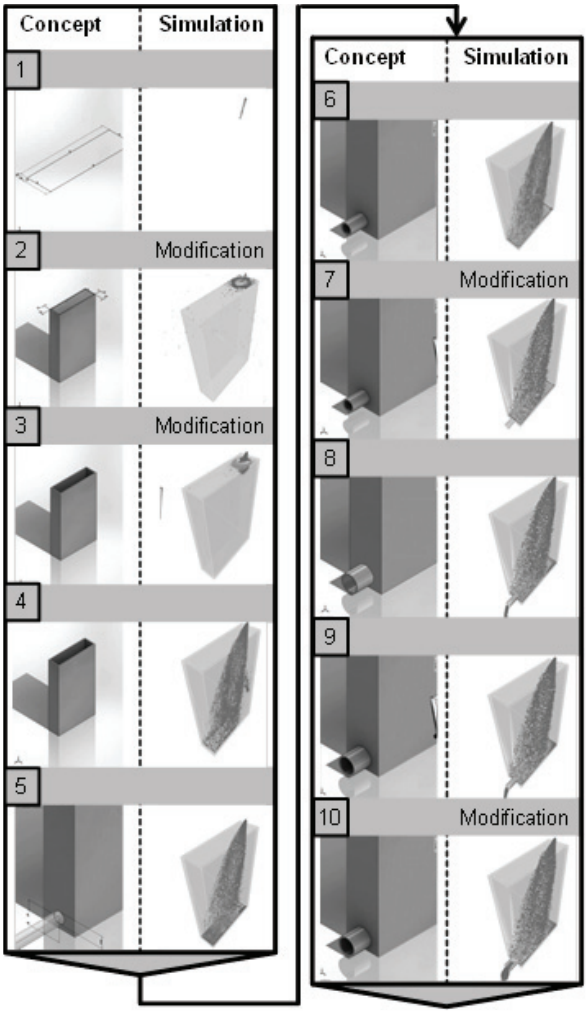

Figure 5: WYSIWYG-online simulation of a filling process

box, which directly gets transferred to the simulation (2). Neither a restart nor a manual effort has to be performed by the user. In step 3, the box gets eroded, which directly influenced the simulation model. As can be seen, the fluid fills the box. Between the steps 3 to 8, an outlet is designed independent of the simulation, which keeps on running. Having finished the design process the size of the outlet can be evaluated directly measuring the flow. Using this easy and fast feedback of design decisions as well as the fast modification of the geometry in the simulation, the user can simply try out the size of the outlet (step 7-10). This way, the user can concentrate on the design and can see the results of design changes directly in the simulation.

5. Summary

A new method for online model generation for lagrangian CFD consisting of four different steps was shown. Furthermore, a use case shows the potential of the method. In addition, first test were made showing the impact of proper convex decomposed geometry information in lagrangian simulations reducing the computational effort to 35.6%. Hence, the online model generation helps to make CFD simulation more economic and therefore allows CFD simulations in areas, which were not simulated yet because of not being economic. Furthermore, new applications like interactive construction are possible.

Our next step is to check different convex decomposition algorithms and the effect of bigger models. Moreover, we plan to investigate multi-user changes as well as further optimizations in the workload of a multi-GPU system.

Acknowledgements

The work presented in this paper was supported and funded by the Bavarian Ministry of Economic Affairs and Media, Energy and Technology and the Free State of Bavaria.

References

[1] Abele R, Reinhart G. Zukunft der Production. Munich: Hanser; 2011.
[2] Eigner M., Daniil R, Radoslav Z. Modellbasierte Virtuelle Produktentwicklung. Berlin: Springer; 2014.
[3] Maropoulos, PG, Ceglarek D. Design verification and validation in product lifecycle. In: CIRP Annals-Manufacturing Technology. 2010, 59(2): 740-759.
[4] Döbler T. Simulation und Visualisierung in der Produktentwicklung. MFG-Stiftung Baden-Württemberg, 2008.
[5] Tu J, Yeoh GH., Liu C., , Computational fluid dynamics: a practical approach. Burlington: Butterworth-Heinemann; 2007.
[6] Hirschel EH, Weiland C. Issues of Multidisciplinary design. In: Hirschel EH, Krause E.100 Vol. of 'Notes on Numerical Fluid Mechanics'. Berlin: Springer; 2009.
[7] Solenthaler B. Incompressible fluid simulation and advanced surface handling with SPH. PhD thesis. Zurich: ETH Zurich; 2005.
[8]] Kortelainen J. Meshing tools for open source CFD-a practical point of view. VTT, Espoo, Finland, 2009.
[9] Georgiadis NJ, Rizzetta DP, Fureby C., Large-Eddy Simulation: Current Capabilities, Recommended Practices, and Future Research. In: AIAA Journal. 2010; 48 (8): 1772-1784.
[10]] Sagaut P, Deck S. Large eddy simulation for aerodynamics: status and perspective. In: Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences . 2009; 367(1899):2849-2860
[11] Davidson DL. The Role of Computational Fluid Dynamics in Process Industries. In; The Bridge. 2002; 32(4):9-14
[12] Crespo AC, Domínguez J M, Rogers BD, Gómez-Gesteira M, Longshaw S, Canelas R, Vacondio R, Barreiro A, García-Feal O. DualSPHysics: Open-source parallel CFD solver based on Smoothed Particle Hydrodynamics (SPH). In: Computer Physics Communications. 2015; 187: 204-216.
[13] Monaghan JJ, Cas RAF, Kos AM, Hallworth M., Gravity currents descending a ramp in a stratified tank. In: Journal of Fluid Mechanics. 1999; 379: 39-69.
[14] Liu GR. Meshless Methods. Boca Raton: CRC press; 2010.
[15] Nguyen VP, Rabczuk T, Bordas S, Duflot M. Meshless methods: a review and computer implementation aspects. In: Mathematics and computers in simulation. 2008; 79(3): 763-813.
[16] Monaghan JJ. Smoothed particle hydrodynamics. In: Reports on progress in physics. 2005; 68(8): 1703-1759.

[17] Crespo AC, Dominguez JM, Barreiro A, Gómez-Gesteira M, Rogers BD. GPUs, a new tool of acceleration in CFD: efficiency and reliability on smoothed particle hydrodynamics methods. PloS one. 2011: 6(6).
[18] Laurien E, Oertel H. Numerische Strömungsmechani. Wiesbaden: Vieweg Verlag; 2011.
[19] Lucy LB. A numerical approach to the testing of the fission hypothesis. In: The astronomical journal. 1997; 82:1013-1024.
[20] Gingold RA, Monaghan JJ. Smoothed particle hydrodynamics: theory and application to non-spherical stars. In: Monthly notices of the royal astronomical society.1977; 181(3):375-389.
[21] Monaghan JJ. Why particle methods work. In: SIAM Journal on Scientific and Statistical Computing. 1982; 3(4): 422-433.
[22] Monaghan JJ. Simulating free surface flows with SPH. In: Journal of computational physics. 1994; 110 (2):399-406.
[23] Vorobyev A. A Smoothed Particle Hydrodynamics Method for the Simulation of Centralized Sloshing Experiments. Karlsruhe: KIT Scientific Publishing; 2012.
[24] Kawai M, Yohei S. Adaptive Parameter Adjustment of a Real-Time SPH Simulation for Interactive Virtual Environments: Application to Parallel Processing. In: Software Engineering, Artificial Intelligence, Networking and Parallel & Distributed Computing (SNPD), 13th ACIS International Conference. IEEE, 2012; 13:692-697.
[25] Morris JP, Fox PJ, Zhu Y. Modeling low Reynolds number incompressible flows using SPH. In: Journal of computational physics. 1997; 136(1):214-226.
[26] Brizzolara S, Savio L, Viviani M, Chen Y, Temarel P, Couty N, Hoflack S, Diebold L, Moirod N, Iglesias AS. Comparison of experimental and numerical sloshing loads in partially filled tanks. In: Ships and Offshore Structures. 2011; 6(1-2):15-43.
[27] Versteeg HK, Malalasekera W. Computational Fluid Dynamics. Edinburgh: Pearson Eductaion; 2007.
[28] Lacour F-FR.. Modellbildung für die physikbasierte Virtuelle Inbetriebnahme materialflussintensiver Produktionsanlagen. PhD thesis, Munich: TU Munich; 2011.
[29] Asafi S, Goren A, Cohen D. Weak Convex Decomposition by Lines - of - sight. In: Computer Graphics Forum. 2013; 32(5): 23-31.
[30] Ren Z, Yuan J, Liu W. Minimum near-convex shape decomposition In: IEEE Transactions on Pattern Analysis and Machine Intelligence. 2013; 35(10): 2546-2552.
[31] Ehmann SA., Lin MC. Accurate and fast proximity queries between polyhedra using convex surface decomposition. In: Computer Graphics Forum. 2001; 20(3): 500-511.
[32] Spitzweg M. Methode und Konzept für den Einsatz eines physikalischen Modells in der Entwicklung von Produktionsanlagen. PhD thesis. Munich: TU Munich , 2009.
[33] Gomez-Gesteira M, Rogers BD, Dalrymple RA, Crespo AJ. State-of-the-art of classical SPH for free-surface flows. In: Journal of Hydraulic Research. 2010; 48(S1): 6-27.