Simulation of Fluid-Structure Interaction between injection medium and balloon catheter using ICFD

Lisa Wiesent[1], Marcus Wagner[1], Markus Geith[1]

[1]Ostbayerische Technische Hochschule[2]

1 Introduction

1.1 Medical Background – Arteriosclerosis and its treatment

Cardiovascular disease is one of the leading causes of death worldwide [1]. It is commonly caused by arteriosclerosis, the accumulation of plaque at the inner wall of the blood vessel. Arteriosclerosis induces stenosis, the hardening and narrowing of the vessels, resulting in a disruption of the blood supply to the heart. The fatal consequence of this may be a stroke. Nowadays, coronary stenosis is mainly treated by the implantation of a stent, a small wire meshed tube. During surgery, a stent crimped onto a balloon catheter is inserted minimally invasive into the stenotic artery using a guide wire. Once the balloon is accurately positioned at the stenosis, a fluid is injected into the balloon to inflate it up to a certain target diameter. The deployment of the balloon leads to the compression of the plaque and the widening of the blood vessel's lumen as well as to the expansion of the stent. Thereupon, the balloon is deflated and removed from the vessel whereas the expanded stent remains at the vessel wall to keep it open. In this way the blood flow through the vessel is restored.

A major long term issue of this procedure is restenosis, the re-narrowing of the blood vessel caused by injuries of the vessel wall during stent implantation. To overcome the phenomenon, the optimization of the stents regarding its design as well as its deployment characteristics is of great interest. As experimental. As the implantation of stents directly affects the patients' health, experimental data on stent design and deployment *in vivo* is difficult to obtain and in many cases not feasible on ethical grounds. Numerical simulations, however, enable investigations on stent design and deployment, as well as material composition and loading conditions without endangering patients' lives. Therefore, several computational finite element method (FEM) models of balloon-expandable stents have been developed.

1.2 State of the Art-Numerical Simulation of balloon-expandable Stents

Stent deployment is a complex procedure. Thereby, contact interactions between balloon, stent and arterial wall arise. The balloon deployment further occurs in a high non-linear manner due to large displacements as well as the unfolding of the folded and pleated balloon configuration. Typcial deployment characteristics are foreshortening and dogboning. Depending on the stent morphology and strut design, the original stent length differs from the one obtained after deployment due to changes in strut angles. The percentage change of stent length pre- and post-deployment is referred to as foreshortening. Foreshortening is associated with the misalignment of struts at the intima of the vessel causing alteration in blood flow and wall shear stress [2]. Stent deployment is observed to be a non-uniform procedure starting at the distal ends of the stent. Dog-boning is defined as the percentage change in diameter between the distal end of the stent and the central part, referred to as dog-boning. Slow-motion capturing of the stent deployment further showed the unsymmetrical characteristics of the dog-boning. After the initial fluid injection into the balloon, the typical dog-bone effect is noticeable. Thereupon, the stent is expanding faster at the balloon's far end compared to the inlet. This leads to an unsymmetrical deployment of the stent. After the final expansion, the stent has a cylindrical shape.

As stent deployment is assumed to be causative of in-stent restenosis, the simulation of this process should be performed as realistic as possible. Therefore, a folded and pleated balloon configuration has to be considered. Stent expansion is initiated by the injection of a fluid. With regard to the non-symmetrical deployment characteristics (forshortening dogboning), the interaction of the injection medium and the balloon structure should be taken into account implying a fluid-structure interaction (FSI) analysis.

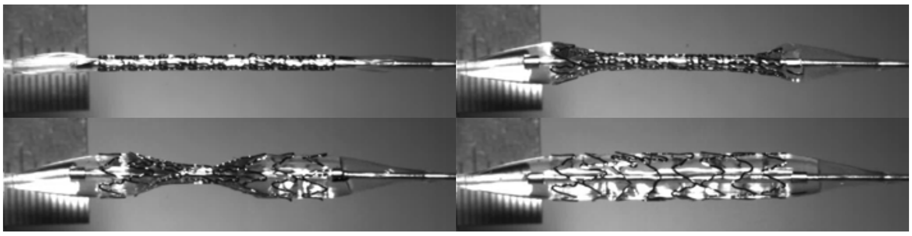

Fig.1: Slow motion track of the deployment of a balloon expandable stent demonstrating the non-symmetrical dog-bone effect

2 Physical Principles of Fluid-Structure Interaction

Fluid-structure interaction (FSI) is a multi-physics problem describing the mutual influence of solids and fluids. Thereby, internal or surrounding fluids apply forces to the structure causing it to deform. This in return, leads to changes in the fluid field, e.g. alterations of the fluid velocity and pressure. PTCA is a typical example of FSI problem. During the balloon inflation, an injection medium applies forces on the structure of the balloon causing it to expand. The resulting deformation of the balloon in return affects the fluid field. FSI consists of three domains, - the fluid domain Ω_f, the structure domain Ω_s and the FSI coupling interface Γ_I.

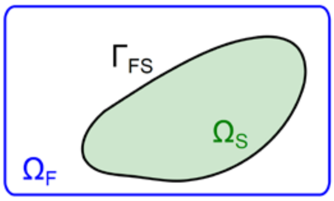

Fig.2: Schematic illustration of FSI

The superscripts ‘s’, ‘f’, and ‘I’ indicate the structure, the fluid subsystem and the fluid-structure interface, respectively. The interface is defined as the part of the structure which is in direct contact with the fluid. The governing equations describing the FSI is based on continuum mechanics. Hence the laws of conservations describe the fluid and the structure domain as well as the fluid-structure interface.

2.1 Governing equations of incompressible fluid dynamics

An isothermal fluid is completely described using four partial differential equations: the conservation of mass and three equations obtained from the conservation of momentum, also known as the Navier Stokes equations. Further, an incompressible fluid is considered stating that the density ρ remains constant over time.

Within the FSI problem, boundary and initial conditions are assigned to the fluid domain. For the inlet, a prescribed fluid velocity is commonly used. If there is no information concerning the flow rate or velocity profile at the inlet, a pressure boundary condition can be defined. The pressure boundary condition can further be used to define a static reference pressure at the outlet. This reference pressure is commonly set to 0. Further, a set of initial condition can be defined to determine the fluid velocity and the pressure at the beginning of the simulation.

Wall boundary conditions are used to bound the fluid and structure domains. It is mainly distinguished between *free-slip* and *non-slip* boundary conditions. The *non-slip condition* states that the fluid velocity at the wall is zero relative to the wall due to viscous effect. This condition is used in most of practical applications. Figuratively speaking, the non-slip condition implies that the fluid *sticks* to the surface of the solid boundary. This condition is essential for the manifestation of a boundary layer at the wall. Mathematically, this condition can be described as

$$v_f = v_I \tag{1}$$

with v_f being the fluid velocity and v_I the velocity of the fluid-structure interface. The *free-slip condition* states that only the normal fluid velocity relative to the wall equals zero, whereas the tangential velocity is not zero. This condition is used if the viscous effects at the wall are negligible.

2.2 Governing equations of Structure mechanics

The inflation of a balloon-expandable stent can be classified as a *geometric non-linear* problem due to the large displacement during balloon deployment. Structure mechanics can be described within continuum mechanics. The three fundamental relations of structure mechanics are: kinematics, material laws and the equilibrium condition.

An overview of the governing equations of the fluid and structure domain is given in figure 3.

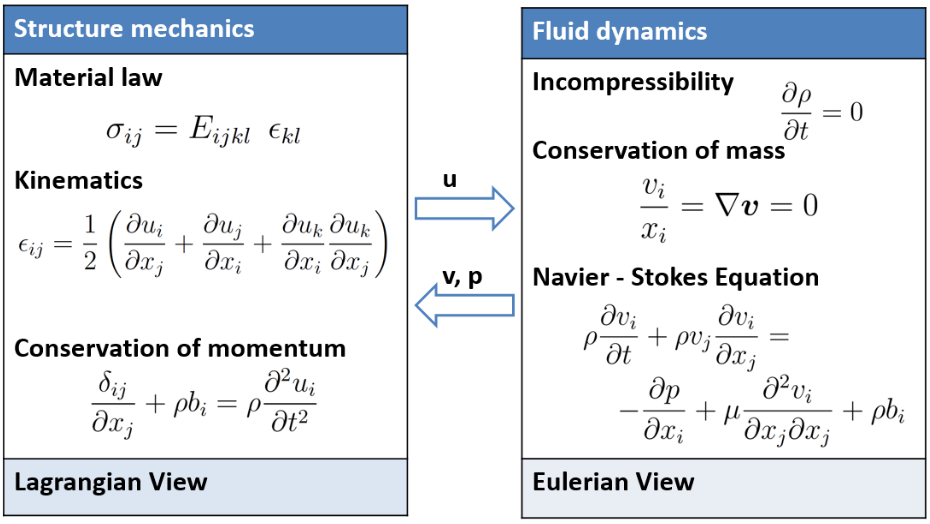

Fig.3: Overview of the governing equations of FSI

2.3 Governing equations of Fluid-Strucutre Interaction

FSI is achieved by coupling conditions at the fluid-structure interface. These conditions guarantee the maintenance of the dynamic and kinematic equilibrium.

The kinematic constraint ensure an identical fluid velocity v_f and structural change of displacement $\frac{\partial u_s}{\partial t} = v_s$ at the fluid-structure interface.

$$v_s - v_f = 0 \qquad \text{, at } \Gamma_I \tag{2}$$

The dynamic constraints ensure the equilibrium of the acting forces at the fluid-structure interface. This constraint is expressed by the stress equilibrium at the interface.

$$\tau_s n - \tau_f n = 0 \qquad \text{, at } \Gamma_I \tag{3}$$

3 Fluid-Structure Interaction – Modelling Approaches

There exist different approaches to solve the system of equations for a FSI simulation. These are mainly classified into two solution approaches - namely a *monolithic* and a *partitioned* approach. Within the partitioned approach it can be further differentiated in terms of the coupling direction (one-/two-way coupling) and the intensity of the coupling. An overview of the FSI modelling approaches is given in figure 4.

Within the monolithic coupling approach, the FSI problem is considered within one mathematical framework Therefore, the equations describing the fluid and the structure problem as well as their interaction are set up within one large system of differential equations. In contrast to the monolithic approach, the partitioned approach divides the multi-physics FSI problem into a fluid and a structure subsystem using individual solvers and discretization schemes. Hence, within the partitioned approach, solution methods can be chosen to best suit the individual physical problem. The interaction of the fluid and solid domain is determined by a coupling algorithm at a determined fluid-structure interface. Using a mapping algorithm, non-conforming meshes at the interface are supported.

Within the partitioned coupling approach, it is differentiated between a *weak* and a *strong coupling* scheme. Within a weak coupling, only one solution for the fluid and the structure domain is obtained once per time-step. The data exchange at the fluid-structure occurs also once per time-step. Therefore, no convergence check between the fluid and structure domain is performed. The time-steps for the fluid and structure domain do not need to match. In contrast to the weak coupling, the strong coupling represents an implicit coupling method. During the solution procedure, data is iteratively exchanged between the fluid and solid solver until a certain convergence criteria is reached at each time-step.

The FSI method can further be characterized by its coupling direction distinguishing a *one-* and a *two-way* fluid structure coupling. Within a one-way coupling either the fluid or the solid solver transfers information to the other domain, but does not receive information in return from it. A two-way fluid-structure coupling is characterized by a large structural deformation resulting in considerable changes in the fluid velocity and pressure field which in return impact the structure domain. Therefore, the mutual interaction the structure and the fluid has to be considered by transferring loads and displacements across the fluid-structure interface.

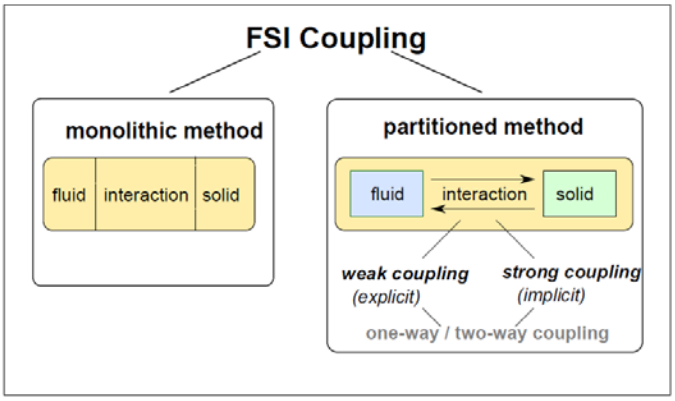

Fig.4: Overview of the FSI modelling approaches

4 FSI – Simulation Model Set-up

The deployment of a PTCA balloon is characterized by large structural displacements caused by the injection medium. Hence, a non-linear FEM simulation is considered. Due to the similar density of the fluid and structure domain ($\rho_s / \rho_f = 0.91$), the FSI model is likely to be highly susceptible to the added-mass effect resulting in numerical instabilities. Considering these characteristics, a partitioned strongly coupled two-way FSI approach was chosen. Within a partitioned approach, a specialized solver can be used for the fluid and structure domain, respectively. Within LS-Dyna, a mechanical and an ICFD solver, are provided. The ICFD solver is fully coupled to the mechanical solver enabling FSI. The choice of the structural solver determines type of coupling. An implicit approach results in a strong FSI coupling ad an explicit approach in a weak FSI coupling. Within a strong coupling, convergence between the fluid and structure domain is achieved. Further, due to additional stabilization techniques within the strong coupling, numerical instabilities due to the added mass effect might be decreased [3]. The implementation of a two-way FSI coupling finally determines for the mutual influence between the fluid and the structure.

4.1 Simulation overview

Within this paper a partitioned FSI approach for the expansion of a balloon by an injection medium is using the ICFD method is presented. The initial mesh was obtained from preliminary simulation including the folding and pleating of a balloon. Two FEA were performed:

1. FSI feasibility analysis: shortened, folded balloon configuration
2. Final FSI simulation: original folded and pleated balloon configuration

4.2 Simulation overview

In general, the model setup of the feasibility and the final FSI simulations only differ in terms of the geometric balloon configuration. However, due to numerical instabilities regarding the final FSI simulation the balloon shell thickness was increased to reach a convergent solution. The fluid pressure used for the balloon deployment had to be increased correspondingly. The key data of the simulations performed is presented in Table 1 The geometric Setup as well as the general model setup is described in the following.

Model	**Length [mm]**	**Shell thickness [mm]**	**Fluid pressure [bar]**	**Termination time [s]**
Feasibility analysis	11	0.025	1	2
Final FSI simulation	7.5	0.06	5	2

Table 1: Simulation overview

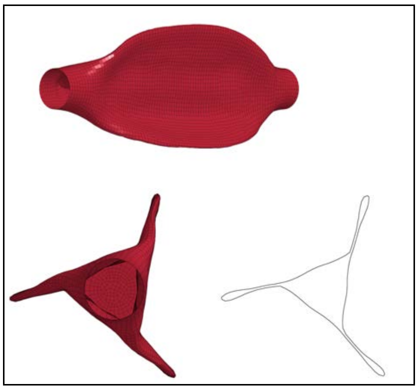

Fig.5: Feasibility Simulation

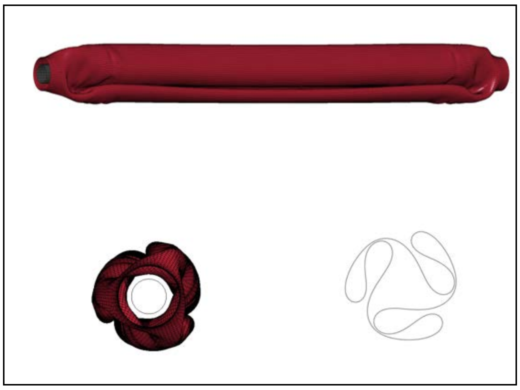

Fig.6: Final FSI Simulation

The simulations time was 2 s. The fluid volume flow is initiated by a fluid boundary condition at the inlet. The balloon wall was defined as the the fluid-strucutre interface. The specific characteristic of this FSI simulation is the consideration of an ecnlosed volume.

4.3 File structure

For the sake of clarity the FSI Analysis was seperated in 5 separate files containing the fluid domain, the fuid mesh, the solid domain, the solid mesh and a main file within which the FSI coupling is realized.

4.4 Structure domain

The structure domain describes the properties of the balloon. The shell type 16 regarding fast, fully integrated elements was used. Further a shell thickness of 0.025 mm was assigned. An elastic material model was chosen and the material properties of the high viscous polyamide Grilamid L25 assigned to the balloon (ρ= 1.01 g/cm³, young's modoulus E=1100 MPa)

The bearing of the balloon was implemented by constraining all translational and rotational degrees of freedom at the two ends of the balloons. At the far end, the translation in the z-direction was enabled to allow for the shortening of the balloon during its deployment. The application of gravity was renounced within the simulation as it does not have an huge impact on the simulations outcome despite the loss of numerical stability and added computational cost. To achieve a strongly coupled

FSI, control cards for an implicit solution were defined. Regarding the large displacements during balloon deployment, a non-linear solution method with BFGS updates was chosen for the implicit analysis. For the structure domain, *solver 5*, a SMP parallel multi-frontal sparse solver working in double precision was selected. This solver enables the coupling to the fluid domain which requires double precision computations.

4.5 Fluid domain

4.5.1 Fluid MS mesh set up

In contrast to the structure domain, the fluid mesh could not directly be obtained from the preliminary simulations as these models consist of shell elements. The ICFD solver, however, only accepts multi-physics (MS) meshes consisting of tetrahedrons. To provide an adequate mesh, the automatic volume mesher was used to generate a volume mesh on the grounds of the balloon shell mesh.

The automatic volume mesher requires a watertight MS surface mesh that defines the fluid boundaries in order to generate a volume mesh. This implies matching nodes at the interfaces as well as no gaps or duplicate nodes [1]. To provide an adequate surface mesh, the initial balloon mesh was duplicated with an off-set of 0.0125 mm considering the balloon-thickness. This ensures that only the lumen of the balloon will be filled with the fluid. In order to generate a watertight surface, the inlet was sealed with a fill plane surface, subsequently meshed and the whole model checked for duplicated nodes. ICFD parts were assigned to the balloon inlet, the wall and the far end.

Thereupon, the mesh was converted into the MS mesh format. Therefore, the keywords ***NODES** and ***ELEMENTS** were changed to ***MESH_SURFACE_NODE** and ***MESH_SURFACE_ELEMENT**, respectively. The desired fluid volume mesh is completely determined by the keyword ***MESH_VOLUME**. Within this keyword, the ICFD Parts constraining the fluid domain and their corresponding MS meshes are assigned to a resulting volume ID. Within the first solution step, a volume mesh will be generated on the grounds of this surface mesh.

Fig.7: Fluid MS mesh set up

4.5.2 Fluid Mesh Refinement

Viscous effects are significant at the walls bounding the fluid domain. To get a better resolution of these near-wall effects, a boundary layer with a thickness of three elements using ***MESH_BL** was defined at the balloon wall. Further, a symmetry condition was applied in order to guarantee that the boundary follows the surface tangent of the surface using ***MESH_BL_SYM**. The balloon deployment processes are characterized by large displacement. In order to maintain a reasonable fluid mesh throughout the whole simulation, an adaptive re-meshing was implemented using ***ICFD_CONTROL_ADAPT_SIZE**. In contrast to the default re-meshing algorithm which only checks for inverted elements, this keyword further considers the element quality and distortion. It was set to run 40 iterations before a re-mesh is forced.

4.5.3 Model set up fluid domain

The fluid domain is bound by the balloon inlet, the wall and the far end represented as ICFD-parts. These are characterized by their material properties and their section. The ICFD-parts further define the boundaries of the fluid domain and are used for the generation of the MS volume mesh as described before. So far physical properties were only assigned to the fluid surface, but not to its volume. To assign properties to the nodes enclosed within the fluid surfaces, a fluid part volume was created using the keyword ***ICFD_PART_VOL**. The part volume is completely defined by the fluid boundary surface determined by the ICFD-parts, the fluid section and material properties. The material properties of water are assigned to the fluid. As an incompressible fluid is considered, no further material data is required. A non-slip condition was applied to the balloon wall and far end using ***ICFD_BOUNDARY_NONSLIP**. This condition ensures that the fluid velocity is assumed to be zero relative to a structural boundary surface. Further, this boundary condition allows for the generation of a near wall boundary layer.

The inflation process of the balloon is implemented by a prescribed pressure condition on the inlet of the balloon using **`*ICFD_BOUNDARY_PRESCRIBED_PRE`**. The pressure is set to rise linearly to a target pressure of 1 bar. A reference pressure of 0 is applied to the whole system.

4.6 Fluid-Structure-Interaction

In this paper, a partitioned strongly coupled two-way FSI approach was considered. A partitioned FSI simulation requires the definition of a fluid-structure interface, the choice of a coupling algorithm (strong/ weak coupling) as well as the definition of the coupling direction (one-/ two-way coupling). The FSI approach is determined by the solution algorithm of the structural subsystem. As mentioned before, an explicit method results in a weak coupling, whereas an implicit methods leads to a strong coupling. Within this modelling attempt, an implicit solution and hence a strong coupling was chosen. The coupling direction is determined within the keyword **`*ICFD_CONTROL_FSI`** by the variable **`OWC.`** Within this modelling attempt a two-way coupling was chosen to represent the mutual influence of the structure and the fluid. The fluid structure interface is defined by the keyword **`*ICFD_BOUNDARY_FSI`**. For the deployment of the balloon, the wall and the far end of the balloon were defined as the FSI interface. At these interfaces the kinetic and dynamic coupling conditions have to be fulfilled. Within an FSI simulation there exist two time steps, one referring to the structure domain and one to the fluid domain. Within a strong coupling, however, only one time step is used for the solution process. Therefore, the smallest time step obtainable from the fluid and structural domain is chosen for the whole FSI solution procedure.

5 Results

The main objective of this thesis was the investigation of the practicability of the FSI approach for the expansion of a balloon using the ICFD method, which is novel. The feasibility analysis showed promising results. However, numerical instabilities arose when this approach is used within the more complex geometric setup that includes completely folded and pleated balloon configuration. Hence, the detailed results obtained from the feasibility analysis are only of secondary importance. The specific characteristics of this FSI problem and its limitations will be discussed in Chapter 6.
By increasing the shell thickness of the balloon to 0.06 mm the successful calculation time of the final FSI simulation could be prolonged. It has to be pointed out that the initial FE mesh still resulted from preliminary folding and pleating simulations of a balloon with a shell thickness of 0.025 mm. Using a shell thickness of 0.06 mm lead to initial penetrations.

5.1 Feasibility Simulation – Small folded balloon

Upon fluid injection, the folded balloon shows an evenly deployment in the course of the FSI simulation (see Figure 8). After reaching the target pressure of 1 bar, the balloon has not reached its final cylindrical shape. Therefore, some edges remain at the initial folds.

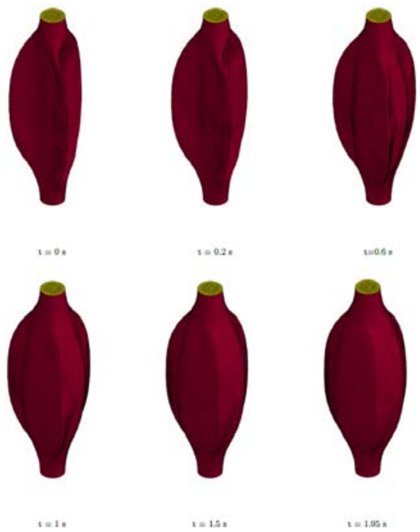

Fig.8: Feasibility Analysis – balloon expansion

5.1.1 Fluid domain

The fluid pressure field within the lumen of the balloon corresponds to the prescribed pressure condition at the inlet. Therefore, the pressure linear increases to a target pressure of 1 bar within 2 seconds. During balloon deployment, the fluid pressure over the entire fluid domain is approximately constant at each time step.

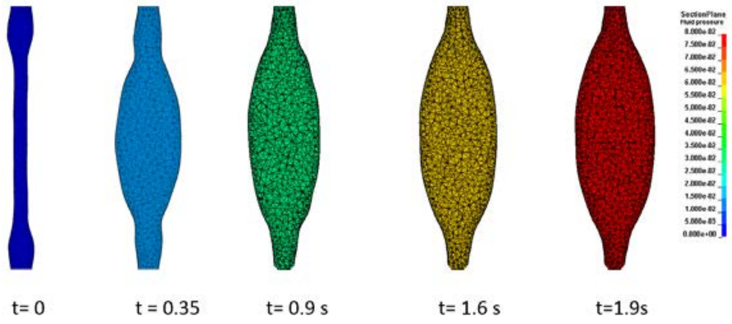

Fig.9: Feasibility Analysis – pressure distribution during balloon deployment

The fluid velocity field is well pronounced. The velocity at the balloon wall equals zero due to the non-slip boundary conditions. The fluid velocity is the greatest at the balloon inlet. This is attributable to the geometry of the balloon. The smallest diameter of the balloon exists at the inlet. Therefore, a diffuser-effect occurs in the transition area of the inlet to the central balloon area implying a reduction in fluid velocity.

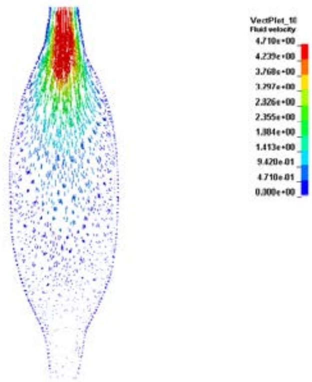

Fig.10: Feasibility Analysis – fluid velocity field

5.1.2 Comparison of the FSI Simulation and the balloon expansion by a mere pressure application

For the reference simulation, a linear rising pressure corresponding to the fluid pressure used within the FSI simulation was applied to the inside of the balloon structure. The other model set up is left unchanged the structural domain of the FSI simulation for comparability.

In general, the stress distribution shows reasonable results within the FSI simulation. The highest stress occurs as the balloon folds (see Figure 11) comparable to the pressure application. A detailed comparison of the results of the FSI simulation and of a mere pressure application is beyond the scope due to time constraints. Therefore, only a brief comparison of the maximum displacements and the order of magnitude of the stress distribution regarding the van Mises stress is considered. At the initial stage of balloon deployment, both simulations show similar results. Proceeding in time, however, larger van Mises stress and therefore larger deformation within the FSI simulation can be observed (see Table 2). As gravity is neglected within the FSI simulations, this phenomenon cannot be related to forces induced by fluid mass. Further investigation is left as future work.

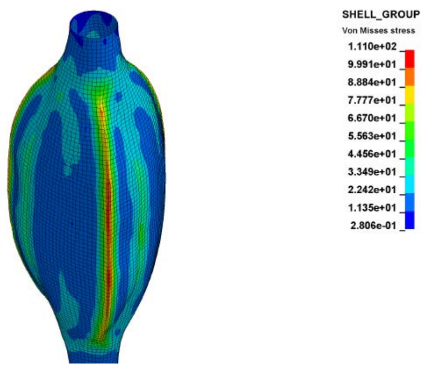

Fig.11: Van Mises Stress distribution during balloon deployment

	FSI Simulation	**Pressure Application Simulation**
$x_{\max}$	0.34	0.29
$y_{\max}$	0.32	0.28
$z_{\min}$	-0.01	-0.01
$\sigma_{v.Mises}$	28.12	24.38

	FSI Simulation	**Pressure Application Simulation**
$x_{\max}$	0.84	0.76
$y_{\max}$	0.75	0.7
$z_{\min}$	-0.03	-0.05
$\sigma_{v.Mises}$	108.21	84,20

Table 2: Coparison of the results from the FSI simulation and the mere pressure application at t= 0.2 (left) and t=1.5 s (right)

5.2 Final FSI simulation- completely folded and pleated ballon configuration

The FSI simulation of the completely folded and pleated balloon configuration led to a non-convergent solution that terminated with an error. During balloon deployment, negative eigenvalues and Jacobi matrices arose implying inverted elements or huge element deformation. The visual representation of the results further showed the local penetration of fluid elements through the structure (see Figure 12). Further, the balloon shows some compressed areas resulting in a rippled surface (see Figure 12). These compressed areas partially preexist within the initial folded and pleated FE mesh and are not related to the FSI simulation.

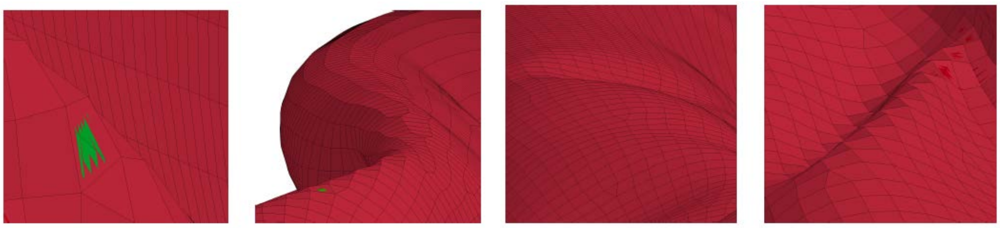

Fig. 12: Final FSI Simulation – Structural mesh and convergence issues

5.3 Mesh development during balloon deployment

5.3.1 Structure Mesh

At the state of the simulation, the initial mesh obtained from preliminary simulations already contains a non-negligible amount of elements that exceed an unfavorable wrapage angle of > 20 degrees. During balloon deployment, the structure mesh is characterized by large displacement and deformation. In the course of the simulation, negative eigenvalues as well as negative Jacobi matrices were observed. This implies large element deformation or even distortion. Mesh deformation especially occurs at the folds of the balloon due to their unfolding. In this area, the mesh resolution is possibly too low to allow for good results. Comparing with metal forming, it is recommended to use at least 4 elements around a 90 degree angle.

5.3.2 Fluid Mesh

Within the FSI simulation, the automatic volume mesher created a reasonable volume mesh on the grounds of the surface mesh of the original geometry. During balloon deployment, the boundaries of the fluid mesh frequently changed. Due to the implemented ALE approach, the mesh was moving 'arbitrarily' in accordance with its structural boundaries. This approach allows for an adaptive fluid volume mesh. In this way, the lumen of the balloon was constantly filled with the fluid not leaving any gaps or holes. Further, the frequent re-mesh of the fluid mesh guaranteed the maintenance of a reasonable mesh quality despite large balloon deformation. Further, additional MS volume elements were created in the course of the adaptive re-meshing of the feasibility analysis but not within the final FSI simulation (see Table 3) The fluid mesh development during balloon deployment is illustrated in Figure 13.

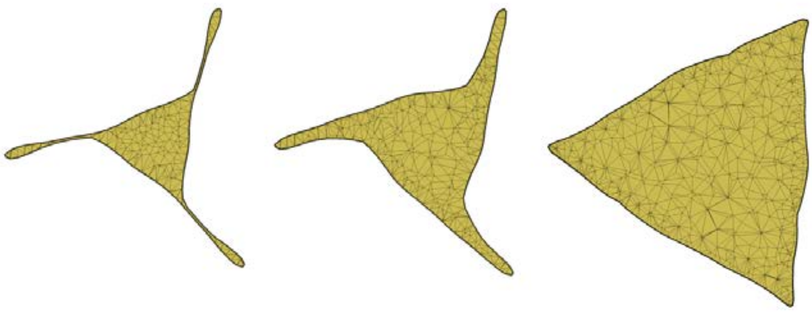

Fig.13: Fluid mesh development during balloon expansion

Mesh refinement was successfully implemented using the adaptive re-meshing as well as the implementation of a boundary layer. The boundary layer led to a higher mesh resolution at the wall of the balloon. This area however consists of acute tetrahedrons.

	FSI Simulation		**Final FSI Simulation**	
Time [s]	0	1.95	0	0.95
MS surface elements	7666	7666	88949	88949
MS nodes	48982	51541	567251	567251
MS volume elements	225970	242137	2621556	2621556+

Table 3: Initial and final MS mesh of the feasibility analysis and the final FSI simulatio

6 Discussion of the convergence issues

Within this approach the ICFD solver of LS-DYNA was used to simulate the FSI between an injection medium and a balloon catheter. The results obtained from the feasibility analysis indicate that this approach is promising for further FSI simulations thanks the continuous balloon deployment as well as the reasonable development of the fluid velocity and pressure field. However, various problems arose when this FSI approach was applied to a more complex geometry of a completely folded and pleated balloon.

The FSI simulation of an enclosed volume using the ICFD solver of LS-DYNA is an innovative approach. Especially considering the flexible, thin walled structure of the balloon, some problem specific issues arose. As the main focus of this paper lies on the practicability of this approach for a further use within the simulation of a balloon-expandable stents, the results obtained from the simplified geometry are of secondary importance. In this manner, it is focused on the discussion of major problems during the simulations and the suggestion of possible solution approaches. The main problems thereby appeared to be the large mesh deformation during balloon deployment, the balloon material as well as the ICFD solver itself.

Initial FE mesh from preliminary simulations:
The initial FE mesh resulting from preliminary simulation shows an unrealistic rippled surface due to local constraints during the folding and pleating process. This surface might lead to numerical instabilities, penetrations and hence to an unrealistic balloon deployment. Some elements have already exceeded a critical wrapage angel of 20 degrees even before the FSI simulation.

Structure shell thickness of 0.025 mm :
Within FEM simulation, a shell thickness of < 0.05 mm is commonly considered to be critical resulting in numerical instabilities. By increasing the balloon thickness from 0.025 mm to 0.06 mm and adjusting the fluid pressure accordingly to 8 bar the calculation time could be increased before error termination resulting in a partially inflated balloon.

Structural deformation during balloon deployment:
During balloon deployment large deformation occurs resulting deformed or even distorted elements.

Density ratio of structure and fluid domain
A density ratio of $\rho_s / \rho_f \leq 1$ known to be susceptible to numerical instabilities due to the added mass effect within a FSI simulation.

Treatment of flexible thin walled structures within the ICFD solver:
The ICFD solver was found to have accuracy problems in case of a fluid pushing at one side of a thin walled flexible structure.

FSI simulation within an enclosed fluid volume
The ICFD solver was further found to have accuracy issues regarding an enclosed volume. This leads to numerical instabilities and an incorrect solution of the FSI condition

7 Summary

In this paper, a strongly-coupled FSI approach using the ICFD method within LS-DYNA was presented in regards to the deployment of a PTCA balloon. The balloon deployment results in a complex FSI model including non stationary fluid boundaries and large structural deformation. The main focus was the examination of the practicability of this FSI approach for the deployment of a balloon and the possible application for the simulation of a balloon-expandable stent in future work.

Thereby, two balloon configurations were examined: a simplified configuration of a short folded balloon and a more complex one of a completely folded and pleated balloon. These balloon configurations were obtained from preliminary simulation work including the folding and pleating procedure of a balloon.

The initial feasibility analysis was successfully performed using a shortened and only folded balloon configuration. The balloon deployment was initiated by an injection medium. Thereby, the fluid pressure was linearly increased to 1 bar within 2 second. The expansion of the balloon occurred continuously. The fluid pressure and fluid field development showed reasonable results. The comparison of the balloon deployment using a FSI method and a pure pressure application showed a similar overall van Mises stress distribution at the balloon structure with the highest value at the folds. However, at a later stage of balloon deployment greater stress and structural displacement could be found within the FSI simulation.

The application of this FSI approach to the more complex geometry of a completely folded and pleated balloon configuration led to convergence problems. Investigations on the non-convergence of the FSI simulation lead to the following explanatory approaches:

Besides the convergence issues the ICFD method proposes a promising FSI approach.

8 Literature

[1] World Health Organization *Global status report on noncommunicable diseases 2014.* World Health Organization, 2014

[2] LaDisa, J. F., L. E. Olson, D. A. Hettrick, D. C. Warltier, J. R. Kersten, & P. S. Pagel. Axial stent strut angle influences wall shear stress after stent implantation: analysis using 3d computational fluid dynamics models of stent foreshortening. *Biomedical engineering online 4* (1), 2005, 1.

[3] Caldichoury, I. ICFD theory manual - incompressible fluid solver in ls-dyna. *Livermore Software Technology Corporation*. (2016).