SUMMARY
Stenting has been becoming a common clinical procedure for the treatment of peripheral vascular districts such as carotid arteries. To assess how stent implantation affects the hemodynamic and mechanical environment, computational modeling provides a useful tool to set up virtual bench tests. In this work we present a virtual stent design platform, combining parametric geometrical design with different finite element analyses, using pyFormex as an open-source interface. In particular we focused on high quality hexahedral stent mesh generation starting from high resolution microCT scanned stent samples.

INTRODUCTION
Several stent designs are available on a dedicated growing market. The stent behavior in the diseased artery depends on many factors such as stent design, material and the target lesion [1] which all influence their deformation ability. In particular the stent design profoundly influences the post procedural hemodynamic and solid mechanical environment of the stented artery. These alterations of the vascular environment affect the long-term patency of stented vessels. As an example, stent implantation can cause wall stress concentrations and a mismatch of elastic properties that provoke a tissue response leading to restenosis [2]. Stent design should, therefore, minimize tissue damage and induced stresses though provide the necessary radial support.

In this context numerical models have become a promising aid to evaluate the properties of stents both as a complement to experimental studies and as their substitute when tests are difficult or impossible to perform. In particular, computational models can be a rapid and economic solution when the investigation of different alternatives is desired to evaluate various combinations of material properties, loading conditions and/or geometries without the need to produce new prototypes or set up real experimental bench tests. Therefore, computational modeling appears to be a great support in stent design during the development phase.

For the above mentioned reasons there is a growing interest in virtual bench testing to predict the mechanical behavior and the biomechanical interaction with the vessel wall in a patient specific modeling environment [1, 3, 4]. In this study we present a virtual stent design platform, combining parametric geometrical design with different finite element analyses, using pyFormex as a dedicated pre-and postprocessor. Even though pyFormex has previously been used for similar purposes, the aim of the present work is to further simplify and automate high quality hexahedral stent mesh generation. The tool is used for meshing a microCT scanned stent samples, though the proposed meshing methodology can also be applied directly to Computer Aided Design (CAD) data.

METHODS
The workflow of the presented computational framework is shown in Figure 1. The geometrical information is obtained from the segmented high resolution microCT images of the stent (acquired either in the expanded or in the crimped configuration in the delivery system) and used for basic measurements of the stent using pyFormex, an open-source software under development at Ghent University [5]. pyFormex is used to generate the high quality hexahedral mesh and to generate the complete input files for the ABAQUS finite element solver. In the input files the following parts are defined (a) the stent mesh; (b) cylindrical sheets used to (i) expand the stent, (ii) form the delivery system, (iii) crimp the stent into the delivery system and (iv) bend the stent inside the delivery system before implantation into patient specific artery models (as proposed in previous studies [1,3]). In case a crimped stent configuration is available from the CT scan, the stent is first expanded, subsequently is annealed to maintain the expanded geometry and to eliminate internal stresses and then crimped again to emulate the stress distribution into the delivery system.

Figure 1 Workflow of the virtual bench test

The present study focuses on high quality stent mesh generation. There are two big advantages in using pyFormex
for this task. On one hand it allows to generate a fully parametric hexahedral mesh of the stent based on high resolution imaging; on the other hand it allows an easy manipulation of the mesh features of the units to be assembled avoiding typical meshing problems [4] occurring with automatic mesh generators such as different element size even for equally sized stent segments.

A database of different shapes has been created. Referring to Figure 2 database includes circular arcs (in red), straight struts (in green), double bended struts (in yellow) and different connectors (in blue) for T and cross connections. These basic parts can be adjusted according to curvature angles, curvature radii, the strut width or the length of the segments measured on the STL data.

The Acculink stent (Abbott Vascular) is used to illustrate the mesh generation.

![Figure 2 Assembling of the different parts. Repeated parts are in dark grey (top). Unrolled STL stent cell to be matched (bottom).](image)

**RESULTS AND DISCUSSION**

Figure 3 shows the mesh of an Acculink stent in its crimped configuration. The different parts of the dataset are combined to create the geometrical repetitive units of the unrolled stent (Figure 2) and then replicated and rolled (Figure3). Thanks to the implemented database of different forms, which are flexible in dimension and geometrical features, the different units can be easily adapted and assembled to match the STL model of the high resolution scan. This approach allows parametric changes of the global features (i.e. stent length, stent diameter, number of rings, etc) or local features of the stent (i.e. strut length, strut bending radius) to compare different configurations and predict various scenarios i.e. for the improvement of the stent design [3].

**CONCLUSIONS**

Computer models have been demonstrated to be a powerful tool in the design of medical devices and the study of their mechanical performance. The proposed methodology will shorten the generation and increase the quality of hexahedral stent meshes and enhance automation of the design process. Future investigation will include the finite element analyses steps and comparison of different commercial stent designs.

![Figure 3 Acculink stent mesh (built with only few repetitive units)](image)

**ACKNOWLEDGEMENTS**

The authors gratefully acknowledge financial support of the Research Foundation – Flanders (FWO) for the financial support of this research activity.

**REFERENCES**


