Demonstrating A Linked Data Visualiser for Finite Element Biosimulations

Muntazir Mehdi*, Yasar Khan*, Andre Freitas†, Joao Jares*, Saleem Raza‡, Panagiotis Hasapis§ and Ratnesh Sahay*
*Insight Centre for Data Analytics, National University of Ireland, Galway,
Email: {firstname.lastname}@insight-centre.org
†University of Passau, Germany
‡Bielefeld University, Germany
§INTRASOFT International, Luxembourg

***Abstract*—Healthcare experts have recently turned towards the use of Biosimulation models to understand the multiple or different causative factors that cause impairment in human organs. The applications of biosimulations have been applied in different biological systems ranging from human ear, cardiovascular, to neurovascular research using Finite Element Method (FEM). FEM provide a mathematical framework to simulate these dynamic biological systems. Visualizing and analyzing huge amounts of Finite Element (FE) Biosimulation numerical data is a strenuous task. In this paper, we demonstrate a Linked Data Visualiser–called SIFEM Visualiser–to help domain-experts to Visualise, analyze and compare biosimulation results from heterogeneous, complex, and high volume numerical data. The SIFEM Visualiser aims to help healthcare experts in exploring and analyzing Finite Element (FE) numerical data and simulation results obtained from different aspects of inner-ear (Cochlear) model - such as biological, geometrical, mathematical, and physical models.**

I. Introduction and Motivation

Finite Element Method (FEM) provide a mathematical and computational framework to simulate dynamic biological systems. The applications of FEM in terms of simulating biological systems range from human ear, cardiovascular, to neurovascular research. The Finite Element (FE) biosimulation experiments are performed on biosimulation models, where these models span across multiple, complex and semantically incompatible domains, such as biological models, geometrical structures and mathematical-physical models.

FE biosimulation models are complex and sophisticated systems. Additionally, visualizing and analyzing results of biosimulation experiments performed on these FE models can grow unmanageable, due to volume, heterogeneity, and complexity of numerical data. A biosimulation model is a system of mathematical equations encoded in a computational language, representing different and often heterogeneous mathematical parameters. There are various tasks involved in performing a biosimulation experiment on a biosimulation model, such as defining geometry of the biological system, creating a mathematical mesh, defining input numerical parameters, solver usage, visualisation and result/output interpretation. Most of these tasks are usually performed in isolated environment, hence, a biosimulation experiment takes many hours. Among these tasks, visualisation, analysis, and interpretation of biosimulation experimental results is a challenging task for the domain-experts.

Parameter Type	Parameter Value	Datatype
MESH DIVISIONS	4.0 4.0 3.0	DIV_L, DIV_W, DIV_H
GEOMETRY	0.035 0.001 0.001 3.0E-4 5.0E-5	(DOUBLE)LENGTH (DOUBLE)HEIGHT (DOUBLE)WIDTH
LOAD FREQUENCY	50 1.0	(DOUBLE)FREQUENCY (DOUBLE)VALUE
MATERIAL PROPERTY	1000 1500.0	DENSITY_RHO, (DOUBLE)SPEED_OF_SOUND
YOUNG MODULUS FUNCTION	0.000000 14101716.454877	(DOUBLE)X_AXIS, (DOUBLE)Y_AXIS
EXTERNAL LOAD FREQUENCY	1.000000e-03 1 1 1 0 1 1 1 1 0.000000e+00	DISPLACEMNT_X, DISPLACEMNT_Y, DISPLACEMNT_Z
HEADING CARD	6060 1 1 1 1 0	IFORM, ISOLVER
NODAL POINT DATA	1 0 0 90 0.000000 1.00e-002 2 1 1000.000	FREQUENCY

Fig. 1: Finite Element numerical parameters, terminologies and format

Our work is conducted in context of the SIFEM EU project[1] and driven by the need of clinical organizations and labs conducting biosimulation experiments to understand the exact pathophysiological consequences and risk factors of hearing impairment in humans. One of the core aim of the project is development of an open-source visualiser to better interpret results of Finite Element multi-scale modeling and biosimulation experimental results of human inner-ear cochlear mechanics. The Finite Elements Method (FEM) – a mathematical framework – is assisting researchers in studying the structure-function relationship in normal and pathological cochlear. Figure 1 shows a set of FE parameters, values and their datatypes required to model the cochlea (inner-ear). Usually, hundreds of such parameters and billions of instances (i.e, values) are required to construct a full-fledged cochlea (inner-ear) model. Once a set of parameters and values are collected a numerical solver performs the finite element

[1] http://www.sifem-project.eu/

calculations with simulation results displayed in graphical format.

In this paper, we demonstrate a Linked Data Visualiser that shows biosimulation results along multiple dimensions. The aim is to combine, link and Visualise different biosimulations data (such as biological, geometrical, mathematical, physical) for inner-ear (cochlea) mechanics. The proposed SIFEM Visualiser addresses following challenges in integrating, interpreting and visualizing numerical parameters along with simulation results:

- Visualise and analyze high volume, heterogeneous, and complex numerical data over FE simulation results.
- resolve different data formats by transforming them into standard RDF format. For instance, all the eight (8) parameters and their values shown in the Figure 1 are taken from separate experimental data files stored in different formats (e.g, .dat, unv, .pak, etc.).
- providing links across input parameters/values and simulations results in-order to reuse relevant data analysis in future experiments.
- validating SIFEM Visualiser on a real-life use case including experimental datasets, terminologies, and models provided by clinical organizations.

II. SIFEM VISUALISER

SIFEM Visualiser [2] is an integral part of SIFEM Linked Data platform [1] which aims at improving the automation in interpretation and visualisation of finite element (FE) models for inner ear (cochlea) mechanics in an integrated fashion. Figure 2 represents the systematic work flow of the SIFEM Visualiser.

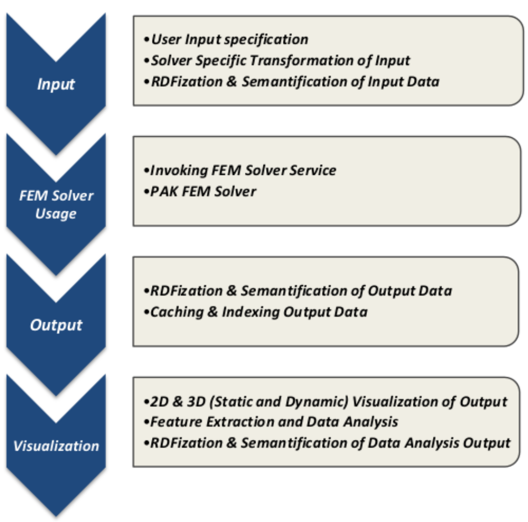

Fig. 2: SIFEM Visualiser Flow

[2]The initial version of the SIFEM Visualiser is released as an open source version, the source code of **beta-version 1** is available at: https://goo.gl/sxePc8. The SIFEM Visualiser can be accessed on http://213.249.38.66:7071/Sifem/ and User-Manual on https://goo.gl/hbTYZt. A video, depicting a sample simulation experiment of the running SIFEM platform can be found on https://goo.gl/wDC0am

To start with a simulation experiment, the experimenter has to provide initial inputs for the experiment. These inputs are dependent upon the Finite Element Modeling (FEM) solver which has to be used in the simulation experiment. Each FEM solver has its own input and output data formats. The experimenter specifies the inputs, such as boundary conditions (material properties), mesh type (box model, coil model), through the SIFEM Visualiser interface and is then transformed according to the solver specific input data format. The input transformation is done by representing the input data using the SIFEM conceptual model, i.e. RDFizing the input data. Figure 1 shows a set of finite element input parameters, values and their data types required in a simulation experiment. Solver specific inputs are then generated from the RDFized input data. This approach leads us to overcome the data input heterogeneity of FEM solvers and makes SIFEM Visualiser a solver independent platform.

Once the input parameters and their values are collected and transformed into solver specific format, the next step is to select and invoke the numerical Finite Element Methods (FEM) solver service, such as PAK, OpenFOAM etc. In our use-case, we use PAK FEM Solver [2]. The output generated by the FEM solver is in solver's native format, which leads to data heterogeneity - in terms of data interpretation using different tools (like matlab, RStudio etc.). To achieve interoperability and resolve heterogeneity at the output level, the output is transformed into a standard and open format, i.e. RDF. Again, the RDFization of output data is conducted using the SIFEM conceptual model.

The next step is to show the simulation results in a graphical format and to analyze the simulation results. To do so, SIFEM Visualiser extracts the numerical results from the RDF output data to Visualise it in either two-dimensional or three-dimensional form. The 2D graphs are static (non-movable) while the 3D graphs are static as well as dynamic (movable across axis). The dynamicity or ability to move 3D graphs help the experimenter to analyze the results in a more detailed and convenient way. The plotted graphs are then analyzed and important features (Examples of such features are graph behaviors, such as Periodic, Linear etc) are extracted from the graph. This is done by analyzing the numerical data plotted in the graph.

ACKNOWLEDGMENTS: *This publication has emanated from research supported in part by the research grant from Science Foundation Ireland (SFI) under Grant Number SFI/12/RC/2289 and EU project SIFEM (contract Number 600933).*

REFERENCES

[1] M. Mehdi, Y. Khan, A. Freitas, J. Jares, S. Decker, and R. Sahay, "A linked data platform for finite element biosimulations," in *Proceedings of the 11th International Conference on Semantic Systems, SEMANTICS 2015, Vienna, Austria, September 15-17, 2015*, 2015, pp. 89–96. [Online]. Available: http://doi.acm.org/10.1145/2814864.2814884

[2] V. Isailovic, M. Obradovic, D. Nikolic, I. Saveljic, and N. D. Filipovic, "SIFEM project: Finite element modeling of the cochlea," in *13th IEEE International Conference on BioInformatics and BioEngineering, BIBE 2013, Chania, Greece, November 10-13, 2013*, 2013, pp. 1–4. [Online]. Available: http://dx.doi.org/10.1109/BIBE.2013.6701611