Integrated Parametric Shaft Generator

Mohamed Khalil[1], Zeno Korondi[1], Mohamed Elhaddad[2], Tino Bog[2]

mohamed.khalil@tum.de, korondi.zeno@gmail.com
mohamed.elhaddad@tum.de, tino.bog@tum.de

[1]Master's Computational Mechanics students, Technical University in Munich

[2]Chair of Computation in Engineering, Technical University in Munich

Abstract. The Integrated Parametric Shaft Generator (IPSG) is an integrated framework for designing and analysis of shafts, a vital component in a plethora of mechanical applications. This framework allows the user to flexibly build the shaft's geometry and mount common shaft features such as fillets, chamfers and keyways, as well as apply the necessary loads at given sections and conduct a structural finite elements analysis, all under one graphical user interface (GUI). The analysis is being conducting using the finite cell method (FCM) for higher order FE analysis, developed at the Chair of Computation in Engineering at Technical University in Munich. This work has also involved implementing a polar cell grid to conform to the cylindrical base geometry of the shafts, aiming to reduce the complexity of the FCM analysis using cartesian grid. The efficiency of both methods have been compared and presented in this work.

1 Introduction

In the introduction we should talk about:

- Shafts
- Shaft analysis methods (analytical, FEM, etc.)

2 Motivation

In the motivation, we should write a line or two about why we want to do this. Examples:

- An integrated environment of analysis and design compared to separate CAD and FEM packages
- Using HOFEM to obtain a conforming elements to the shaft's geometry and possibly for the features as well since they all have a closed-form geometric description

3 Scope

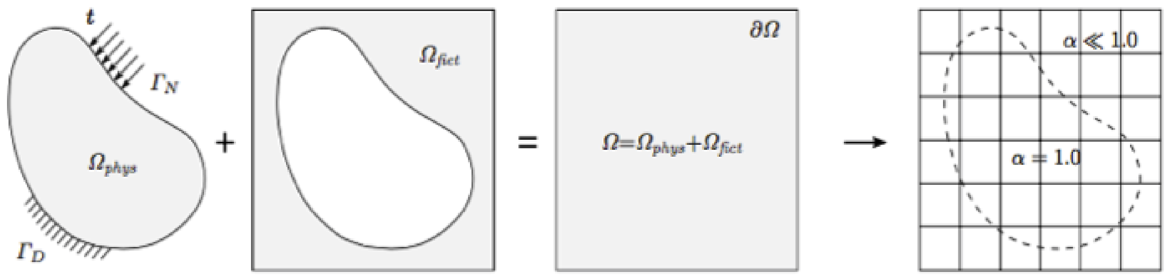

Figure 1: Schematic representing the principle of the finite cell method

3.1 The Finite Cell Method

Finite cell method is an embedded domain method in which the subject domain under investigation is *embedded* in a grid of cartesian finite cells as shown in figure 1. Cells that cross the boundary of the domain are partitioned by a bisecting their sides (forming bi-tree in 1D, quad-tree in 2D and octree in 3D) for a given number of times, known as the partioning depth. For every cell, the equation system is evaluated at a number of integration points n using equation 1

$$\delta W = \int_\Omega \delta v \mathbf{B}^T \mathbf{C} \mathbf{B} d\mathbf{v} - \int_\Omega \delta v \mathbf{P} \alpha d\mathbf{v} - \int_\Gamma \delta v \mathbf{t} \alpha d\mathbf{a} \tag{1}$$

where Ω is the volume of the cells' domain, Γ is the boundary of the cells' domain, δv and δW are the virtual displacement and work, respectively, **B** is the B matrix, **C** is the constitutive law matrix, P and t are the body and surface loads, respectively. The distinction in the evaluation of the physical domain Ω_{phys} and the fictious domain $\Omega_{fict} = \Omega - \Omega_{phys}$ is determined by the value of α given by:

$$\alpha = \begin{cases} 1.0 & \forall x \in \Omega_{phys} \\ \approx 0 & \forall x \in \Omega_{fict} \end{cases} \tag{2}$$

3.2 Higher Order Finite Elements Method

The choice of FCM is usually associated with using HOFEM due to its robustness in representing state-variables' field with higher order functions and the ability to achieve logarithmic rates of convergence (at worst linear for non-smooth problems) by increasing the polynomial degree of the shape functions (p-refinement) compared to decreasing the edge size (h-refinement) of linear elements. In addition, the usage of the hierarchical approach of order elevation offers the versatility of formulating elements with exact conformity to the meshed topology (or highly conforming for non-definite topologies). The latter feature allows the shaft's base geometry, which is of cylinderical natrue, to be represented exactly by *blending* two edges of a hexagonal element[1]. Hierarchical order elevation is based on overlaying the linear shape functions N_1 and N_2 in equation 3, referred to as *node modes* by a hierarchy of higher order Legendre polynomials ϕ_i across the edges in one dimensions, referred to as *edge modes*. To extend this concept in 2D and 3D, the edge modes are blended across the element by performing a tensor product with orthogonal nodal modes or edge modes, forming internal modes and face modes, respectively. Details on this topic are more thoroughly presented in (Duester, 2011)

$$N_0 = \frac{1}{2}(1-\xi) \tag{3a}$$

$$N_1 = \frac{1}{2}(1+\xi) \tag{3b}$$

$$N_i = \phi_i(\xi,\eta) \tag{3c}$$

In the case of a shaft, each section was represented by n blended hexes extruded over its lenth. Figure 2 shows a quad with two edge modes, representing equations of an arc, being applied to its edges E_0 and E_2. The formulation of this blending is given in equations 4.

$$\begin{aligned} x = &\sum_{i=0}^{3} N_i(\xi,\eta) X_i \\ &+ \big(E_{x0}(\xi) - (\frac{1-\xi}{2}X_0 + \frac{1+\xi}{2}X_1)\big)\frac{1-\eta}{2} \\ &+ \big(E_{x2}(\xi) - (\frac{1-\xi}{2}X_3 + \frac{1+\xi}{2}X_2)\big)\frac{1+\eta}{2} \end{aligned} \tag{4a}$$

$$\begin{aligned} y = &\sum_{i=0}^{3} N_i(\xi,\eta) Y_i \\ &+ \big(E_{y0}(\xi) - (\frac{1-\xi}{2}Y_0 + \frac{1+\xi}{2}Y_1)\big)\frac{1-\eta}{2} \\ &+ \big(E_{y2}(\xi) - (\frac{1-\xi}{2}Y_3 + \frac{1+\xi}{2}Y_2)\big)\frac{1+\eta}{2} \end{aligned} \tag{4b}$$

[1]Different features of the shaft such as fillets, keyways, etc. could also be represented by blended hexes due to their closed-form geometry description but this wasn't implemented within the scope of this work

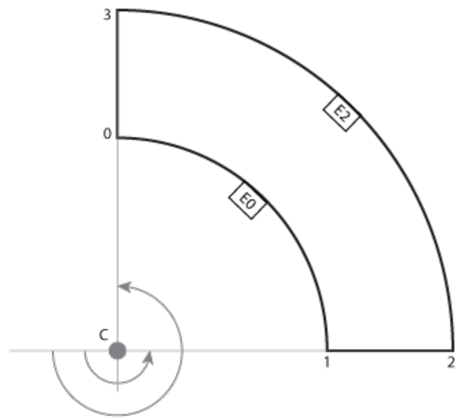

Figure 2: Blended quad element

where X_i and Y_i are the nodal coordinates, N_i are the linear nodal modes of a quad, and E_{xi} and E_{yi} are the edge modes of the ith edge in x and y described by an arc with radius R_i, center coordinates (X_c, Y_c), start angles ϕ and end angle θ as shown in the following equations:

$$E_{xi}(\xi) = X_c + R_i cos(\frac{1-\xi}{2}\phi + \frac{1+\xi}{2}\theta) \tag{5a}$$

$$E_{yi}(\xi) = Y_c + R_i cos(\frac{1-\xi}{2}\phi + \frac{1+\xi}{2}\theta) \tag{5b}$$

The HOFEM concept was in principle used to reduce the intensity of partitioning needed by FCM. As explained in 3.1, the cut cells need to be partitioned up to a given depth at the integration level in order to reduce the intergration error. however, using blended hexes, as formulated above, for the FCM grid cells forming a polar grid rather than a cartesian grid results in cells conforming to the base domain of the shaft and isolating the need for partitioning to the zones where the features are located, as shown in figure 3. The reduction in partitioning depth and locations is reflected in turn in a reduction in the number of integration points to be evaluated (which are typically high for HOFEM, usually one point more than the polynomial order)

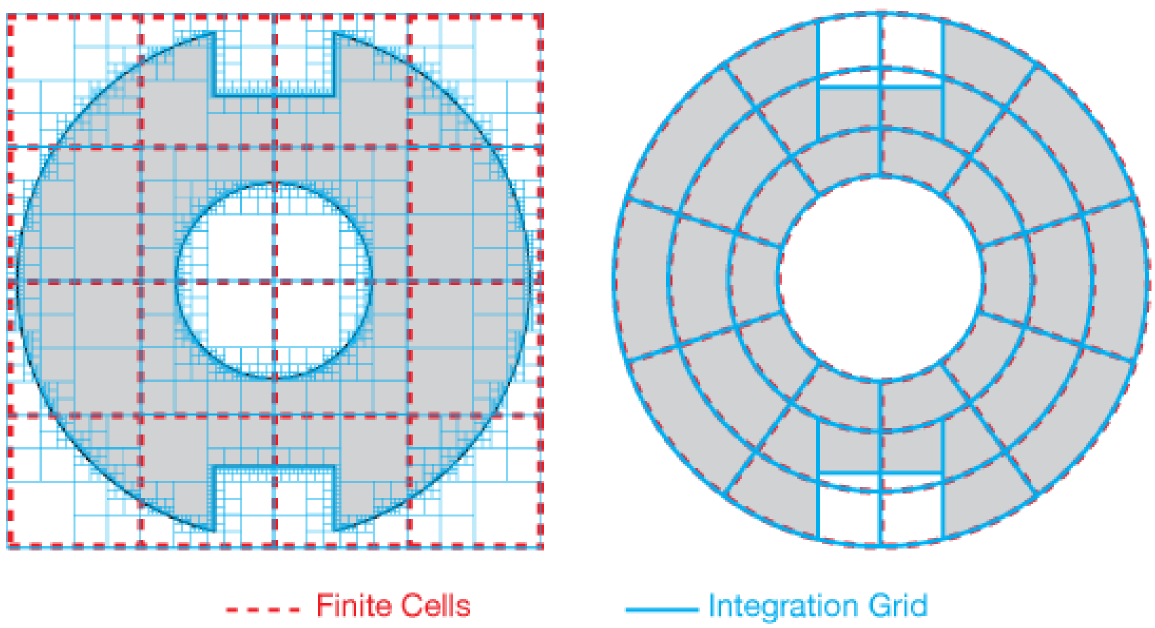

Figure 3: Comparison between the integration grid for cartesian FCM grid (right) and polar FCM grid (left)

3.3 Data Structure

The IPSG is composed of three major blocks:

- Shaft Data: This block is scripted in Python language and is concerned with the data storage of the shaft details as well as its components (loads, features, etc.)
- Analysis Kernel: This block is scripted in C++ as part of Adhoc++ and is concerned with conducting FCM analysis on the shaft, obtaining its geometry from STL files generated by OpenCascade libraries utilized by the GUI.
- GUI:

  Mohamed: *responsible for GUI: Zeno*

The link between the first two blocks is indicated in the UML class diagram in figure **??**. The interface *AnalysisKernelFcmWrapper* was responsible for wrapping the functionalities of the c++ class *AnalysisKernelFcm* to make them accessible through the corresponding Python class *ShaftAnalysisKernel* using *boost::python* library.

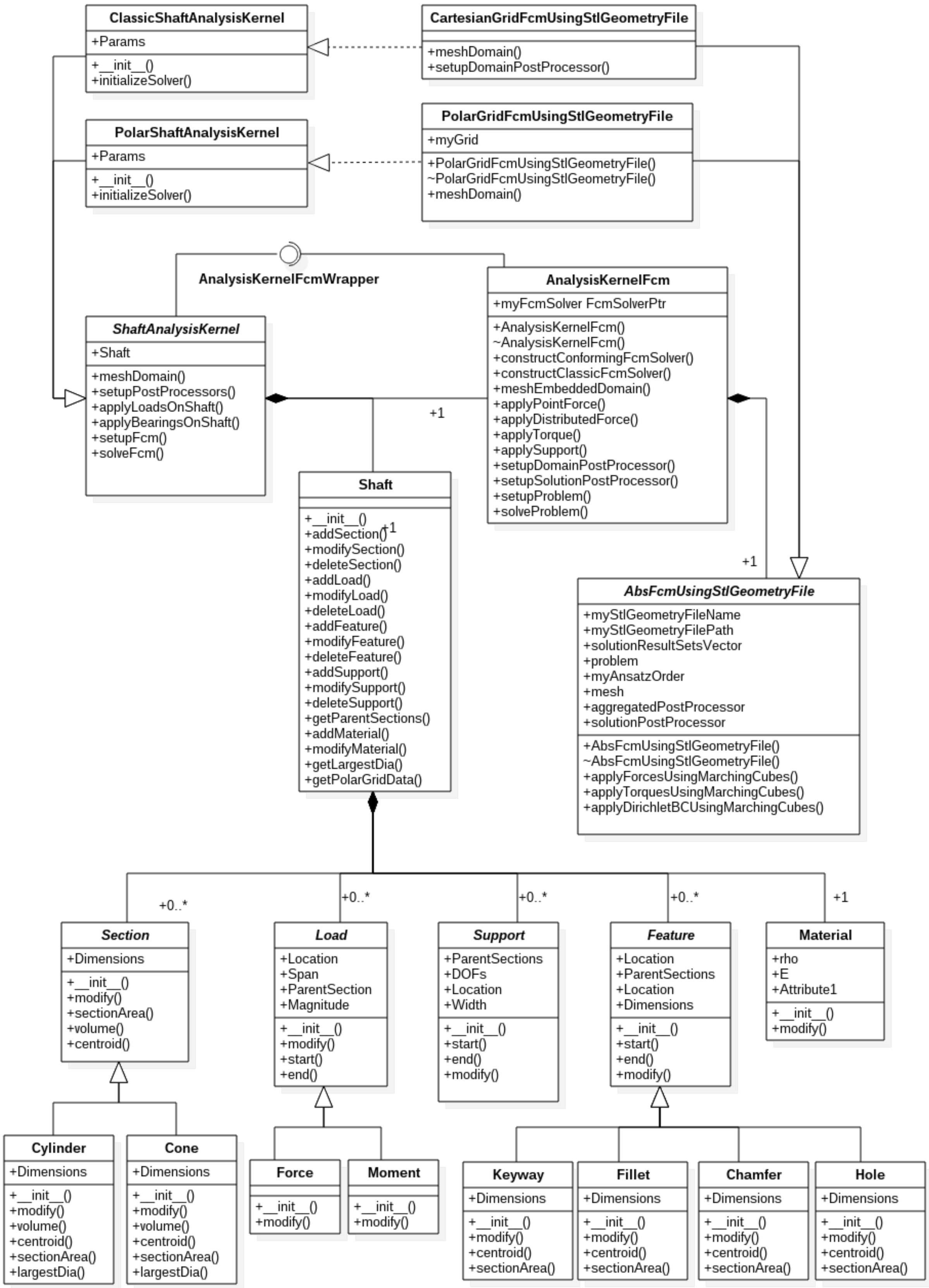

Figure 4: UML Diagram for Data Structure of the Shaft and Analysis Kernel

4 Tools Used

Different tools we have used for the software lab. Last slide of the first presentation. python and c++ for programming, VTK and OC for GUI and results visualisation, boost libraries for wrapping, etc.

5 Results

When we get results, I will write something here

6 Conclusion

summary of the work and comments on the results. Future work of this project etc.

References

Duester, A. (2011), *Higher Order FEM*.