13th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2009, Hammamet, Tunisia, 21 – 26 October 2009

FINITE ELEMENT BASED DYNAMIC ANALYSIS OF A DEVICE IN COILED TUBING DRILLING

Hirpa L. Gelgele, Dr.Ing, Assoc. Professor
University of Stavanger
Dept. of Mechanical and Structural Engineering and Material Technology
N- 4036 Stavanger, Norway

ABSTRACT

The need for flexibility in operation and minimized drill time has necessitated the extended use of coiled tubing drilling technology. Among others, the technology eliminates the time to connect drill pipes and thus enables a quick retraction and insertion of different tools in the well. On the other hand, the complexity of the operating conditions makes the task of testing functionality of components in the system, and getting full understanding of the acting loads is difficult. For example, recent studies show that the working mechanism of a jar device (Down Tor Hammer (DTH®)) used in coiled tubing drilling is not fully understood. There exists no well established mathematical description of the mechanism that can be used to ensure a proper functionality of the device. Furthermore, the device works in an extremely complex environment involving several fields such as dynamics of fluids, solid and structural mechanics that are in iterative interaction. This paper presents a Finite Element Method (FEM) based study of DTH® used in coiled tubing drilling. After giving a short description of the operation principle of the device, the paper illustrates the dynamic modeling technique used to get better understanding of the stress distribution in the device.
Keywords: Finite element analysis, dynamic modeling, coiled tubing, fluid-structure interaction

1. INTRODUCTION

The use of coiled tubing drilling has expanded recently mainly due to the need for improved flexibility in operations and minimized drill time. Among others, the technology eliminates the time used to connect drill pipes that directly influences the retraction and insertion time of different tools in the well such as drill bits, valve sleeve insertions, etc. We witness in several cases that new technological developments in drilling operations are mostly driven by economic interests. Before putting new tools in operation however it is very crucial that the tools are studied for their capability to function in the designated manner. Particularly, the product development phase of the design process requires thorough understanding of both how the tool functions and its failure modes so as to prevent early product failure. Good design should aim to identify the weakest or most vulnerable parts of a system, and manufacture the product to withstand all foreseeable stresses as well as active environmental agents within the projected lifetime of the product. This demands a great amount of calculations (analysis) and testing with respect to a clearly defined loading and boundary conditions, materials, dimensions and connectivity with other interfaces. A starting point for all such activities is a working knowledge of the different operating states and the potential failure modes of the product or the process.

Closer study of the literature has confirmed that the mechanism that drives this tool (DTH) has not previously described mathematically. This is one indication that though numerical simulations have been used for quite some time in several fields, their application in the interface between mechanical components and fluid flow, referred to as "Fluid-structure interaction", in short FSI, is recent and very

limited [1]. Research in this area is nowadays a rapidly growing discipline that represents the natural next step in simulating mechanical systems. The problem can involve either a simple flow within complex geometry or highly complex flow phenomena in a simpler geometry or both are complex systems. The structure is generally undergoing large deformations and the fluid is governed by the incompressible or compressible Navier-Stokes equations [2]. Since Finite Element Methods (FEM) are now widely used in the analysis of solids and structures, implementing these tools in the fluid-structure interface can provide great benefits in product design.
The objective of this article is to present some of the results of the study done to establish the fundamental understanding of the stress distribution in a jar device used in coiled tube drilling technology. The study focuses on the dynamic modeling and analysis of the mandrel assembly using Finite Element Methods.

2. BACKGROUNDS AND MANNER OF OPERATION

The DTH®, on which this study is based, is a tube shaped jar activated device for use in an oil- or gas well, and has an external diameter of 43 mm, a length of about 1400 mm and weighs nearly 6 kg. It is operated by fluid flow that creates the pressure needed to charge two springs. While one of the springs (spring 1) functions as the main spring used to transform fluid energy into impact energy, the spring 2 functions as part of the valve mechanism crucial for the functionality. In other words, spring 1 controls the movement of the mandrel assembly and spring 2 controls the movement of the piston assembly.
Compared with other jar devices that have to be charged in either direction after a single impact, this device vibrates upon charging. Furthermore, the device is taking over the role of other commercial jar devices like long coil tube device in many areas because it can perform better and can be operated at far more controllable forces. In case of long coil tubes, it is not easy to control mechanical forces, whereas the fluid flow in DTH® can easily be monitored.

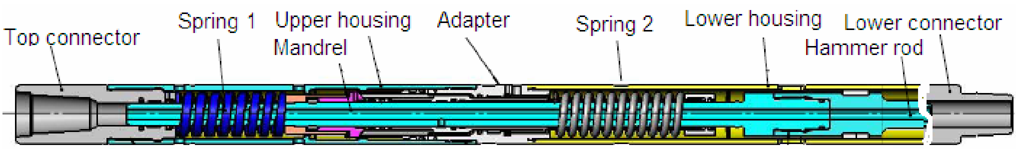


Figure 1. Pictorial section (partial) of the DTH® showing the main components, adapted from [3]

The DTH® has a wide range of applications including maintenance or cleaning up of existing production wells and special operations such as setting plugs and other finishing operations. Another configuration of the tool that strokes upward also exists and it is called Up Tor Hammer (UTH®). This configuration differs only in the direction of action.
The tool is activated by a pressure difference generated between the inner pipe of the mandrel and outer annulus of the hammer rod when the fluid is flushed through a nozzle at the lower connector. The pressure difference comes from an irreversible energy loss due to sudden contraction at the entrance of the fluid. This pressure difference can be expressed using Benoulli equation as follows

$$\Delta P = p_1 - p_2 = \frac{\rho}{2}\left(v_2^2 - v_1^2\right) \qquad \text{.......... (1)}$$

Where p_1 and p_2 are the pressures at entrance and exit and; v_1 and v_2 are fluid velocities at entrance and exit respectively.

3. BUILDING FINITE ELEMENT MODEL FOR DYNAMIC ANALYSIS

The commercial FEA tool ABAQUS® was used to build the dynamic model and analyze it and the major part of the analytical calculation was done in MATCAD. The impact energy on the mandrel assembly consists of two parts: mandrel and hammer rod. The foundation for the stiffness matrix in the FEA was created by assuming the general elastic properties of steel (i.e. E = 206 GPa and poisons ration ν = 0.3). Since no plastic deformation of the tools has been so far documented, only elastic properties are assumed in this analysis. The fluid in this study is water with density $\rho = 10^3$ kg/m^3 and dynamic viscosity $\mu = 1.002 \times 10^{-3}$ Pa s (at T = 20 ^{0}C).

3.1. Choosing Element Type

The preprocessing step of a numerical analysis represents a crucial phase. One important step to guarantee a near exact solution in any FEA is choosing a suitable finite element model (beam, plate or shell) that can define the problem without too much deviation from the physical geometry. ABAQUS® element library provides a set of geometric elements and allows using any possible combination of them [4]. For this type of problem, one can choose either the tetrahedral or hexahedral element types. Hexahedral elements were employed for this analysis to ensure better visualization of the results. While all triangular and tetrahedral elements use full integration, quadrilateral and hexahedral elements can use reduced integration that give sufficient integration points to integrate exactly the contributions of the strain field that are one order less than the order of interpolation. This element type, also referred to as C3D8R, has 8 nodes and three degrees of freedom per node. The letter R in the element designation indicates that this element uses reduced integration. As elements with reduced integration compute average strain values using uniform strain formulation, they provide greater accuracy than fully integrated elements. Furthermore, using reduced number of integration points decreases CPU time and storage requirements particularly in cases where analysis at many nodes and for many elements is executed. To allow complete generality in material behavior, all elements are integrated numerically by replacing the virtual work integral into summation as below:

$$\int_v \boldsymbol{\sigma} \cdot \tilde{\boldsymbol{\varepsilon}}\, dV = \sum_i^n \sigma_i \cdot \tilde{\varepsilon} V_i \qquad \text{.......... (2)}$$

Where σ is the stress vector, $\tilde{\varepsilon}$ is the virtual strain vector, n is the number of integration points in the element and V_i is the volume associated with integration point *i*.

3.2. Meshing the Model

Tetrahedral elements have been used to mesh model of the mandrel assembly because these types of elements contribute to the simplification of the model. Ideally, the mandrel assembly is expected to have a homogenous strain circumferentially. Therefore, it is preferable to have a homogenous mesh as well. As shown in Figure 2, the mesh-generating algorithm in ABAQUS generates meshes based on edge seeds. Since the comparative size of radial holes with respect to the size of mandrel assembly does not allow a uniform mesh size, the assembly was first meshed and analyzed without the hole.

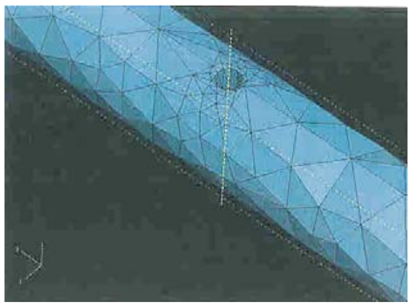


Figure 2.

3.3. Loads and Boundary Conditions

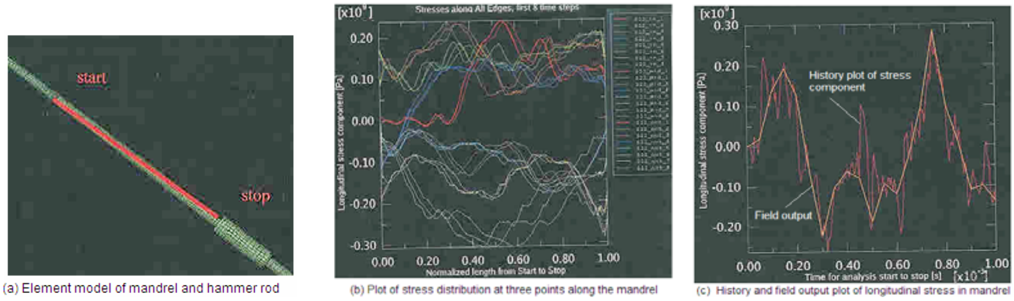
The initial velocity of the mandrel assembly was calculated (calculation details omitted) and set to 6.63 m/s at a distance 1 mm from the impact point, i.e., the lower connector. This gives an impact after just 0.15 ms. Neglecting the small amount of spring energy compared with the impact force, the model is exposed only to the impact force. The displacement boundary conditions of the mandrel assembly was defined in such a way that it is radially fixed at the inside wall of either the upper or lower housing.

4. DISCUSSION OF RESULTS

The main interest in this study is to get better understanding of the stress (axial) distribution in the mandrel. ABAQUS® lets two ways to get output results: the field output request and the history output request. The former writes data to the output file at low frequencies while the latter method generates data at a rather higher frequency. Furthermore, results can be post-processed and visualized by means of tabular list and graphical plots such as deformed shape plots, contour plots, vector plots and X-Y plots.

For the sake of simplicity, the mandrel assembly was analyzed without stress concentration (radial holes removed) and the result was multiplied by a factor derived from a simplified model of a cutout of the mandrel with the hole. As

Figure 3(b) shows the stress wave propagation that follows an irregular pattern after impact. This was expected because of the characteristics of the mass matrix in the dynamic analysis model. The chaotic behavior of the stress wave propagation is attributed to the fact that the mass matrix is not lumped; it is rather consistent.



(a) Element model of mandrel and hammer rod (b) Plot of stress distribution at three points along the mandrel (c) History and field output plot of longitudinal stress in mandrel

Figure 3. Element model of analysed area and output plots

A lumped mass matrix that is concentrated at the nodes could have been obtained if the geometry had been homogenous and the boundary conditions were defined at the nodes. From the first 8 plots in

Figure 3(b), we can observe that the results lack a systematic stress pattern that varies between -300 and +200 MPa without considering the stress concentration factor. Material of the mandrel has the following specifications: AISI 4340 36-40 HRC, proof stress 896 MPa and ultimate tensile strength 1034 MPa. Accounting for the stress concentration (about 3) this shows a safety factor of the mandrel assembly ranging between 1 and 1.5.

To investigate the stress level at the position where the hole is located (92 mm below the start point in Figure 3(a)), a randomly chosen element in the vicinity was extracted and studied. The history plot of stress and the field outputs for this element over 1 ms has been given in

Figure 3(c). This stress history also shows an irregular stress pattern with the axial stress varying between ± 300 MPa.

5. CONCLUSION

Closer study of the literature shows that the dynamic behavior of DTH® has not previously been described mathematically, and the stresses within the mechanical components have not been mapped. One possible reason for this is the complex fluid-structure interaction in the mechanism. As this field is currently a very exciting research area involving a domain of a very general nature with complex geometries, the study partially reported in this article has attempted to establish better understanding of the dynamic processes of the mechanism using FEMs. The analysis assumed only elastic deformation of the mandrel assembly and performed the stress analysis. In order to ensure a near exact solution of the analysis, a convergence study along randomly selected path has been done by using different mesh refinement techniques until a sufficient convergence is registered. The current state of the analysis is far from sufficient and the continuous further advances will for sure lead to many exciting challenges and solutions for practical applications.

6. REFERENCES

[1] Bathe, K. J., Zhang, H. and Li, S., Finite element analysis of fluid flows fully coupled with structural interactions. *Computers and Structures,* **72,** pp 1–16, 1999.

[2] Chakrabarti, S.K., *Numerical models in Fluid Structure Interaction*, WIT press, 2005.

[3] Svartvatn. T., *Dynamical analysis of a jar device for use in coiled tubing drilling using Finite Element Methods*, MSc thesis, University of Stavanger, Norway, 2007.

[4] ABAQUS® users manual (PDF documentation), version 6-7.