CHAPTER 4

SIMULATION OF THE TUBE HYDROFORMING PROCESS

4.1 Introduction

The finite element solution of a tube hydroforming of metal process is complex but time saving in determining the best optimal ways to get the desired shapes. Finite Element Method (FEM) is one of the versatile numerical techniques used for solving physical problems posed in terms of differential or integral equations. In FEM modeling, the work-piece is represented by a collection of sub-domains called finite elements. The elements are bounded by sets of nodes. They define localized mass and stiffness properties of the model. Equations of equilibrium, in conjunction with applicable physical considerations such as compatibility and constitutive relations, are applied to each element to construct a system of equations. The system is then solved to find unknown values using advanced numerical techniques [89].

There are two modes of description of the deformations in continuous media, the Lagrangian and the Eulerian. The Lagrangian description employs the coordinates of a material point in the undeformed state as independent variables. The Eulerian description employs the coordinates of a material point in the deformed state. Because of this, the finite element mesh in the Eulerian system is spatially fixed, whereas in the Lagrangian system it moves with the material. The solution procedure is either explicit or implicit [89].

In the last few decades, the use of finite element for modeling the sheet metal hydro-forming process has increased tremendously. These simulations can predict the deformation process and the failure modes, such as wrinkling and local thinning, of the workpieces during sheet metal forming processes. In literature, both quasi-static implicit and dynamic explicit approach has been used for hydroforming simulation.

In the static implicit methods, which were the very first methods used in simulation of metal forming processes, static equilibrium is satisfied in the unknown final configuration of a time increment. Also, inertia forces are neglected in implicit method. This method enables a full static solution of the deformation problem with convergence control. Theoretically the increment sizes can be very large; practically, however, it is limited by the contact conditions. Computational times increase almost quadratically with increasing element number. Because of the matrix inversion step and accurate integration schemes, memory requirements are also high. Another disadvantage of the implicit methods is said to be the problem of divergence of the solution. This is basically initiated due to the large number of contact nodes which can overload many implicit contact algorithms. Another disadvantage of the implicit methods is the singularity of the stiffness matrix at bifurcation points, such as instabilities at wrinkling initiation [90].

In comparison to implicit method, inertia forces are included in dynamic explicit method. Because of the quasi-static nature of most hydro-forming processes, the dynamic explicit approach has the advantage of less memory requirement and higher computational efficiency. The approach has several significant advantages over the conventional static implicit method for hydro-forming problems. In the explicit method, there is no direct, banded, linear equation solver. Consequently, the computational cost of a solution does not grow quadratically with the problem size. In general, the computational cost is linearly proportional to the problem size in the dynamic explicit procedure. Large deformation, and sliding and contact constraints, are relatively easy to implement in the explicit procedure. The kinematic contact constraints can be enforced explicitly by the direct trial-and-error method and, since there is no equation solver, change of contact conditions does not require the consideration of bandwidth optimization [91]. A by-product of the explicit scheme is the determination of the wrinkles. The wrinkles are initiated through numerical inaccuracies. Usually, the region of the wrinkles is quite accurately determined; but the number of wrinkles and the amplitude of the wrinkles may be inaccurate due to the numerically driven initiation process.

The explicit character of the numerical scheme is fulfilled if and only if the mass matrix is lumped, i.e. is diagonal. Furthermore, the speed advantages can be hold only if the element computations are as few as possible. This is maintained by using single-quadrature elements, which deliver rather poor stress and strain accuracy. It is claimed that the error introduced by a lumped mass matrix is compensated by the reduced integration schemes of the elements. However, this leads to the disadvantages that local stresses and spring-back is not reliably computed [91]. Another drawback of dynamic explicit approach is that it is conditionally stable. The stability limit for the explicit integration operator is that the maximum time increment must be less than a critical value of the smallest transition times for a dilatational wave to cross any element in the mesh. Also, the nature of the explicit method limits it to the analysis of short transient problems. Thus, if this approach is used for quasi-static problems, the inertia effects or dynamic effects must be small enough to be neglected. However, that requires very small-time increments and it leads to a large amount of computational time. Therefore, to solve quasi-static problems, it is conventional to convert the real problem to a virtual problem with a different mass density and processing time by means of mass and time scaling [90].

In the present work, dynamic explicit approach has been used for simulating the sheet hydro-forming process in ABAQUS/explicit.

4.2 Concepts of time

Three different time concepts are used in connection with explicit finite element simulation of sheet metal forming operations – the process time, the simulation time, and the computational time. The process time is the real time used in a manufacturing machine to produce a part. The simulation time is the fictitious time used for application of forces and boundary conditions in the computer simulations. The computational time is the real time used for the computer simulations. The process time is usually 0.5-10 seconds, the simulation time is in the range 10 milliseconds (ms) to 300 ms and the computational time is from 1 hour to a week, depending on computer and model size.[92]

For typical sheet forming operations, the time step is in the order of one microsecond. Since the process time is in the order of one second, millions of time steps are required for a simulation run. A standard approach of reducing the computational time is to model the forming process as if it occurs in a shorter time, i.e., using the simulation time. Another way achieving this speed-up of the process time is to change the mass of the moving parts in the forming process.

There is a limit for the speed-up. The kinetic energy of deformation must be kept below a certain level so that the dynamic effect in the simulation is negligible. With appropriate care, accurate solutions should be possible for metal forming processes. Ahmad Assempour et al (2009) suggest a pressure estimate for how close a metal forming simulation is to a static process using upper bound method [93]. It could be a useful input for the choice of simulation time. The effects of speeding up the deep drawing process and the consequences of different shapes of the punch load curve also investigated. The two-dimensional simulations are done in NIKE2D and DYNA2D [94].

4.3 Contact analysis

In ABAQUS, the contact interface consists of a master surface and a slave surface. The contact algorithm performs a so-called slave search which means that the algorithm checks for penetration of slave nodes into the master surface see Figure 4.1.

When the slave nodes and the corresponding master segments are identified, the contact

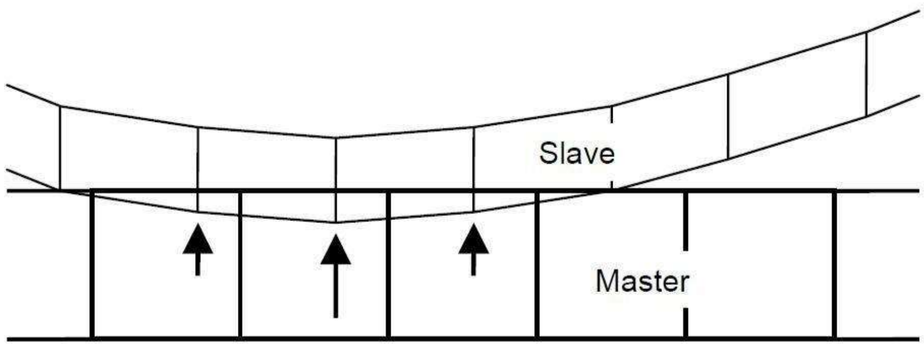

Figure 4.1: Penetration of three slave nodes and application of interface forces [95]

algorithm may proceed according to several different schemes. A widely used method is the penalty procedure. It gives a smooth initial impact and minimal disturbance to the mesh. The penetration of nodes is prevented by the application of artificial springs between the penetrating slave nodes and their corresponding master segments. The interface force to a slave node is calculated from the supposed

Penetration length and an interface stiffness based on the bulk modulus and the dimension of the element. The slave nodes which have penetrated are relocated to the master surface. Other typical contact analyses in current use are direct methods and Lagrange multiplier methods.

4.4 Loading path determination

The selection of loading paths for axial feed and inner pressure are usually based on past experience and/or trial-and-error FEM simulations. The former relies heavily on a coworker with years of experience and the latter can easily be a time-consuming process of obtaining satisfying results. Therefore, it is desirable to construct an algorithm, so that the loading paths also are the result of the FEM simulation. The objective is to obtain a good solution from only one simulation run, thus eliminating the trial-and-error procedure. Koc (2002) [96] discuss the importance of designing correct loading paths and presents a few tube hydroforming processes, done both experimentally and numerically. Bulge forming of tubular components to complex shapes is often difficult to analyze by classical analytical procedures. By adding other forming phenomena, such as preforming by bending or crushing, make the analytical computation impossible.

4.5 Simulation study and component modeling

4.5.1 CAE Simulation:

To conduct CAE simulation, it is first represented in detail i.e., in geometry content, and therefore, a CAD model in a suitable format is made for the metal forming simulation systems. The CAE engineers need to create the simulation test models for a given deformation system. The physical model idealizes the real engineering problems and abstracts to comply with certain physical theory with assumptions.

The mathematical model specifies the mathematical equations such as the differential equations on FEM analysis along with detailed boundary and initial conditions and their constraints. The numerical model describes the element type, mesh density and solution parameters. Usually, most CAE packages have part of built-in content of these models, but users still need to prepare and input most of the model information into a CAE system.

The CAE simulation process consists of four main steps (1) pre-processing (2) simulation analysis, (3) post processing and (4) result analysis and evaluation. After the simulation, the calculated results need to be analyzed and evaluated. If the results and solutions are satisfactory then changes and modifications may be suggested for the metal forming system (part design, tooling design, process configuration and material selection) which can be made for next round of simulation. The process is therefore, iterative in nature [Ming Wang Fu (2006)][97].

The integrated numerical simulation of tube hydroforming system, including metal forming process and tooling deformation provides a systematic approach for simultaneously modeling of mechanical deformation behaviors, metallurgical phenomena and thermal phenomenon of the metal forming processes. Besides, the interaction and interplay of these physical behaviors and phenomena, their interaction to each other are also taken into consideration. On the other hand, the simulation and analysis of tooling deformation and stress strain distribution during the process can be conducted simultaneously along with the simulation.

4.5.2 Simulation Steps

The FEM package ABAQUS 19 is used to simulate of tube hydroforming process. The simulation process consists of following five steps:

1. Initial geometry- In this step CAD model of work piece, die file is imported into the pre-processor of the simulation package, (ABAQUS).
2. Material data and meshing- In this step, already existing material and their material properties are assigned to the previously imported products. Also, there is a provision for defining new materials. The material properties available from literature can be imported easily in this step. Meshing can be done with the use of tetrahedral elements.
3. Defining boundary conditions- Different types of boundary conditions such as internal pressure, coefficient of friction is applied. Different types of combinations of these conditions are tested to get the best possible results. Finally a output data base file (.odb) is generated. This file is run in the simulation engine.

4. Simulation engine- Performs the numerical analysis and writes the resulting information in the data base file.

5. Post-processing- This is used to extract the information from output data base file. In this step different typed of defects, load requirement and thickness distribution are studied. Effective stress and strain also strain rate and velocity also are found.

4.5.3 Simulation of Tube Hydroforming machine using FEA

Finite element simulation of tube hydro-forming process, for elliptical shape made up of aluminium 6061 T4 alloy samples, has been carried out in finite element code ABAQUS-explicit. The FE modeling steps is same as discussed in section 4.5.2.

4.5.4 Plasticity Model and Boundary Conditions

Aluminium alloy 6061 T4 with thickness 0.5 mm is used for this study. In general power law is adopted for modelling the plasticity of stress strain curve of the material. In this study nonlinear properties are considered for modelling of true stress-strain curve from the tensile test. The advantage of using this plasticity model is the tested stress strain curve is considered into the simulation. Following details of parameters used for simulation on Abaqus

Table 4.1: Details of Parameters used for simulation in Abaqus

S.No.	Properties Used in Simulation	Corresponding values
1	0.2% offset yield strength (MPa)	312
2	UTS (MPa)	327
3	Total Elongation	15.6%
4.	Strength Coefficient	315 MPa
5.	Strain hardening exponent	0.2
6.	Coefficient of friction between tool and blank	0.2

7.	Hardening model used	Isotropic
8.	Mesh size	2.23 mm
9.	Mesh Type	Triangular
10.	Total Number of Mesh Element	312 x 312
11.	Simulation Run Time	17 Hours

Symmetric boundary conditions have been applied on the nodes lying at the symmetric planes. The die, die supporting fixture, and plates have assumed to be rigid and have been completely constrained. Except axial direction, ram have been constrained in other directions. The ram is locked in die after sealing it in axial direction. Contact (Penalty-based approach) is modeled assuming Coulomb friction between the die and the tube samples, between the ram and the die end contact, and between the tube and ram. Friction between the tube and the die surface was very low (friction coefficient 0.1 or lower) due to the fluid pressure. Higher friction (friction coefficient 0.2) is assumed between the ram (about 10 mm in radial direction) and the tube end surface, where the contact condition is severe due to the thickening and lower fluid pressure. The friction coefficient between die and ram is assumed to be 0.2; between tube and liquid inside tube, the friction was assumed to be much lower than that between the tube and the outer surface contact with die because of the liquid itself behaves as lubricant used between tube and tube inner surface with liquid fill during forming. The liquid was not modeled; instead, a uniform distribution pressure within the die opening was used to apply.

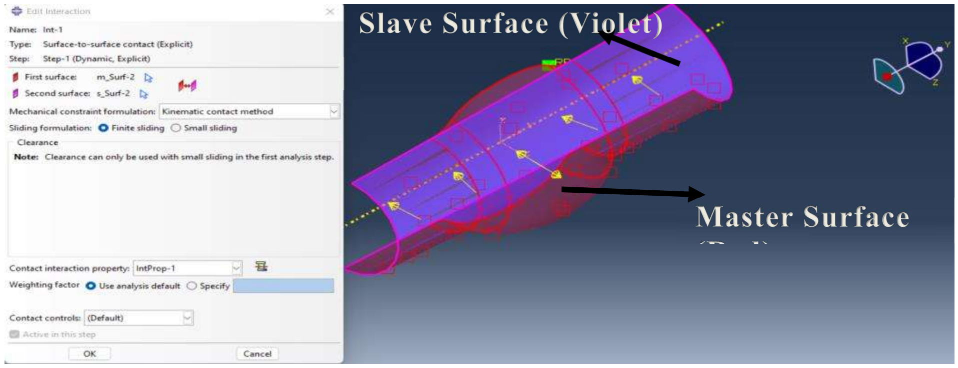

Figure 4.2: Assembly of tube and die set-up for tube hydroforming simulation

the fluid pressure directly applies inside tube inner surface, while the fluid pressure was assumed to be linearly increases in the radial direction under the closed die holding, and end sealing until it reaches to maximum. The pressure distribution was assumed to vary with the ram travel along the elliptical bulge area. The FEM model is shown in figure 4.2.

4.5.5 Validation of Numerical Model

The finite element model has been validated for the tube hydroforming process experimentally. The thickness variation and deformed shape as obtained from the finite element analysis has been compared with the obtained from experiment.

In order to study the introduction of fluid pressure in tube hydroforming process. Finite element simulation has been carried out for elliptical bulge with fluid pressure. The thickness variation along the axial length by an applying fluid pressure is shown in figure 5.4 to 5.8. The von-Mises stresses, the maximum principal stresses and axial deformation decreases in tube hydroforming due to fluid pressure.

4.5.6 Measurement of thickness along the axial length of the elliptical bulge tube

To perform challenging research with a directional valuable solution by giving clear insight into complex systems of hydroforming and to study its parameter which affects the sample wall thickness during the forming process, finite element software having nonlinear property was used. Finite element analysis of elliptical tube hydroforming was developed in Abaqus software with some experimental conditions. The analysis includes a die which is rigid, deformable tube with the same condition of internal pressure and constrained axial force. Half die-tube wall interface profile was drawn and meshed instead of a whole die drawing to reduce the computational time. Only the half-elemental surface of the required rigid die was defined. The finite element model and specification of the tube sample are shown in fig. 4.3 and 4.4.

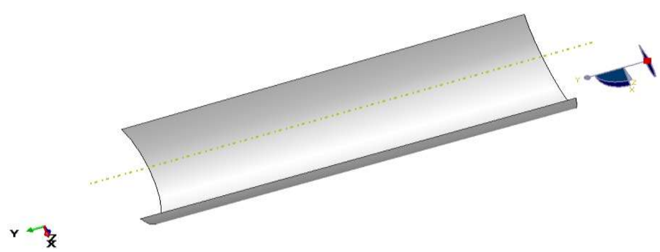

Figure 4.3: Tube model in Abaqus software

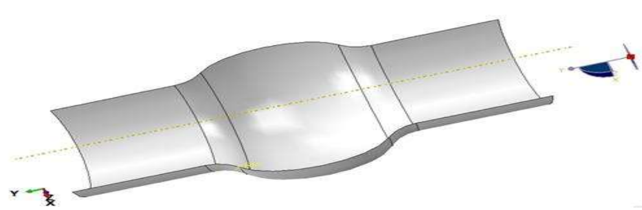

Figure 4.4: Die model in Abaqus software

The tube wall experiences both stretching as well as bending motions while hydroforming. Due to their efficiency in resolving bending problems, linear quadrilateral elements are chosen for use in such simulations. 350 nodes and 312 elements make up the modeled tube section [17]. Die, which is made to be a stiff surface, serves as the master surface while the tube surface serves as the slave surface. The assumed friction coefficient is 0.1. The finite sliding approach is used to formulate the interaction of the tube surface with the die and allows for the separation of two surfaces during sliding. As depicted in Fig. 4.5 the pressure rises linearly at a rate of 2.5 MPa/s.

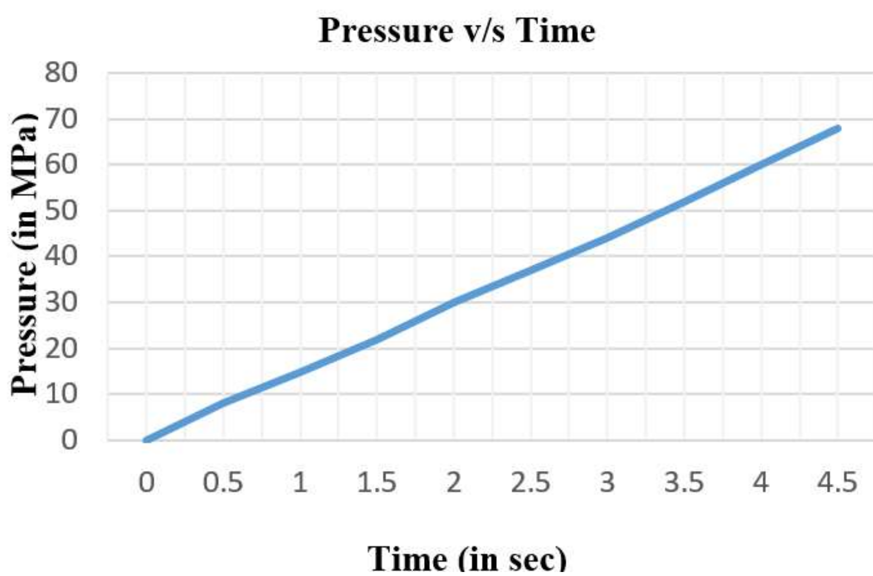

Figure 4.5: The loading path of the sample was taken in the simulation