Development of IT Framework for the Dimensional Inspection of Milling Parts using STEP-NC Compliant Standard ISO 14649

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Abstract

In modern manufacturing environment, manufacturing facilities are spread around the world and have equipment's from different manufacturers. A large amount of product information flow across different facilities of a manufacturing enterprise. A lot of work is being done on integration of product data. For mechanical parts, product information has been standardized by STEP (ISO 10303) in the form of exchangeable files, application programming interfaces and database implementation. STEP has enabled the use of standard data across the entire manufacturing environment. Machining process is one of the most extensively used process across any manufacturing enterprise. There is a need of integration of information at machining level. Most modern CNC machines used ISO 6983 (G&M Codes) as programming language. This program of machine can either be generated manually or by CAM system that uses the information from a CAD system. Programs based on ISO 6983 contains very less information and have limitations.

STEP NC (ISO 14649) is a new standard for transfer of data from CAD/CAM systems to CNC machines. In this new standard, product data from STEP (10303) is used for communication between CAD/CAM systems and CNC controllers. It is a two-way communication. The STEP-NC system is based on features. It not only contains information about the tool path but also contains the information about the feature.

Measurement and inspection of manufactured parts is one of the most important step in manufacturing enterprises. In this research work, STEP NC compliant framework is used for the dimensional inspection of the parts made on CNC milling machine. This framework uses the information from STEP to generate a feature based inspection plan of prismatic parts. A prototype to check the functionality of this framework is made using php language.

Key Words: STEP (ISO 10303), STEP NC (ISO 14649) Inspection Processes, CNC

Abbreviations

ANSI	American National Standards Institute
AP	Application Protocol
CAD	Computer Aided Design
CAM	Computer Aided Manufacturing
CAPP	Computer Aided Process Planning
CLM	Closed Loop Manufacturing
CMM	Coordinate Measuring Machine
CNC	Computer Numerical Controlled
DIS	Draft International Standard
DMIS	Dimensional Measuring Interface Standard
DXF	Drawing Exchange format
GD&T	Geometric Dimensioning and Tolerancing
IGES	Initial Graphics Exchange Specification
ISO	International Organisation for Standardisation
IT	Information Technology
NC	Numerically Controlled
PDES	Product Description Exchange for Standard
STEP	Standard for the Exchange of Product Model Data

Table of Contents

List of Figures

List of Tables

CHAPTER 1: INTRODUCTION

1.1 Background

Manufacturing is defined as converting of raw material into some useful products after passing it through several processes. All things around us are manufactured by some means and technology. In today's world, there is a tough competition for finding new, inexpensive and quality methods of manufacturing. Manufacturing methods and procedures have a great impact on the economy of any country. Modern technique of manufacturing can help to reduce the cost and improve the quality of product, because modern techniques reduce the rejection and increase the durability of any product. So, it is necessary to adopt modern techniques for manufacturing to decrease the cost and improve the quality of products.

Machining is one of the major manufacturing process used that is costly and time consuming as compared to other manufacturing processes. The cost and the time involved in machining can be reduced by adopting new techniques and machine tools. A part can be machined in several ways and the time and cost involved depends upon the way which is adopted. The selection of efficient way will help to reduce the cost and time of the machining part.

CNC (Computer Numerical Control) machines are one of the modern machine tools that is used for the machining of the parts. In the beginning, NC (Numerical Control) machines were made but with the advancements in computer technology, CNC machines came into existence. First NC machines was made in early 1940s. The slightly modern machine was made in 1950s. These machines were basic with motorized path control [1]. The CNC machines now available are the most advanced CNC machines with multi axis control and are equipped with multipurpose tools. The modern CNC machines are also equipped with inspection tools that help to inspect the part after it is machined. If we look back over the last 50 years we see a lot of advancement in CNC machines technology. The CNC machines have proved to be economical in mass production, batch production and some single-item production cases [2]. The most important factors of CNC are high productivity, low rejection rate, improved quality, less operator involvement, low tooling cost and better control over complex geometries [3].

CNC machines programming is based on ISO 6983 also known as G & M codes. These G & M codes represent the tool path and ON/OFF commands of machines. The programs of CNC are either manually generated or by computer (CAM systems) by using the CAD files. The generated program consists of G, M, S, F etc. commands [4]. ISO 6983 programming language has many short comings. Firstly, is only focuses on cutter path rather than machining tasks with respect to part. Secondly, it only defines the syntax of the program and leave the semantics unclear. Thirdly, it is sometimes vendor specific and varies from machine to machine. Fourthly, the flow of information is unidirectional, any change that is made at shop floor level cannot be fed to the designer. Fifthly, CAD data cannot be used on the machine directly. It should be passed through a machine specific post processor so that program for a specific machine can be generated [5].

Keeping in view the above mentioned short comings many data exchange formats have been developed over the past three decades. The primary purpose of all data exchange formats was to exchange geometric data. The formats mostly accepted were DXF (Drawing Exchange Format), IGES (Initial Graphic Exchange Specification) and PDES (Product Description Exchange for Standard). STEP (Standard for exchange of Product Model)/ISO 10303 was a new standard for data exchange that was published in 1994. STEP provided a neutral mechanism and can describe the entire product data throughout the lifecycle of product, irrespective of the system. Part AP11 of STEP contains a data description language "EXPRESS". EXPRESS language is an object-oriented programming language. Its graphical form is known as EXPRESS-G. STEP contains a large amount of information regarding any product [5].

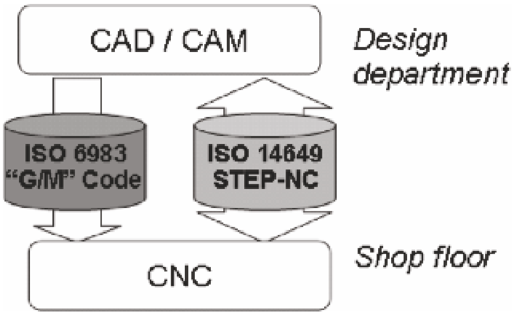


Figure 1.1 Comparison of ISO 6983 & STEP-NC [6]

For CNC machines a new data exchange standard was published. This standard was published using the basics of STEP and name given to it was STEP-NC (ISO 14649). This standard can generate CNC program by using CAD system only. STEP NC was introduced to an ISO group as DIS (Draft International Standard) in 1997. The general title for STEP-NC was Data Model for Computer Numerical Control and AP-238 of STEP [7]. STEP-NC gives an object-oriented data model for CNC machines that contains information about the feature to be machine, tool to be used, operation to be performed and workplan. STEP-NC has many parts (given in chapter 2) that provides information such as general process data, manufacturing features, machining processes (milling, turning, Wire-EDM etc.), set up and tooling, and inspection.

1.2 Aims and Objectives

The main objective of this research is to investigate the area of component inspection in a manufacturing environment and to find the application of standards using a STEP-NC compliant inspection process. STEP-NC will be used for the feature based inspection of the prismatic part made on milling machine and the result of the inspection will be fed back to the physical file based on STEP-NC. The feature selected for inspection purpose are:

1. Through holes drilled in rectangular pattern
2. Step (Shoulder milled part)

A STEP-NC compliant inspection framework is proposed for above mentioned parts and this framework is made using STEP-NC (ISO 14649-16 WD). Part 16 of ISO 14649 is for inspection of parts that are made on CNC machines. This part was then withdrawn and merged in ISO 10303-219. The STEP-NC compliant framework will generate an interoperable physical file for the generalized inspection plan of the rectangular pattern and step (shoulder milled parts) and will store the inspection results. The framework will also generate inspection files based on IS0 6983 and this file will be used for the inspection of parts on CNC milling machine.

The aims and objectives of the research are summarized as follows:

i. Selection of features to be inspected
ii. Selection of inspection operation
iii. Selection of probing strategy
iv. Review the automated inspection programming and planning in a manufacturing environment, feature based inspection process and different inspection standards like STEP, STEP-NC and DMIS (ISO 22093).
v. Developing STEP-NC compliant inspection information model for selected features. Model will provide information about shape of part, geometry of part, features, dimensions, geometric tolerances and probing tool.
vi. Developing framework based on STEP-NC complaint information model. This framework will generate physical file based on STEP-NC. The framework will also generate a file based on ISO 6983. The framework will

generate files after getting input from the user. The input will include information about the holes to be drilled, arrangement of pattern, depth of step, length of step etc.

vii. The file based on ISO 6983 will be checked on CNC machine for validation purpose.

1.3 Structure of Thesis

This thesis is divided into five chapters. The description of the five chapters is as follows:

1.3.1 Chapter One

Chapter one of this research work is about background for this research is discussed. Aims and objectives of this research are also discussed in this chapter.

1.3.2 Chapter Two

Chapter two discusses STEP and STEP-NC in detail. Parts of STEP and STEP-NC relevant to research work are also discussed in detail in this chapter. Feature selected for research are also discussed in chapter two

1.3.3 Chapter Three

Chapter three of this research work contains the information about the standards that are used for inspection around the world. ISO 10303 and ISO 14649 are discussed in detail in this chapter. The schemas that are used for generation of physical file are given in this chapter.

1.3.4 Chapter Four

Chapter four is heart of this research work. In chapter four development of framework is discussed in detail. Structure of physical file the inspection of the selected feature (Rectangular Pattern of through Holes & Step feature (shoulder mill)) is also discussed in this chapter alongwith the examples.

1.3.5 Chapter Five

In chapter five, working of framework is shown alongwith example. The generated file is mapped with G&M code file and is physically checked on the machine. The results of inspection are shown in chapter five of the research. The conclusions and future research on this research is also discussed in chapter five.

Chapter 2: STEP-NC

2.1 Introduction

In last two decades, a lot of research has been done on STEP (ISO 10303) standard. STEP is involved in manufacturing at all level but in this research only part related to dimensional inspection will be considered. The part of STEP-NC ISO 10303-219, ISO 10303-238 and ISO 14649-16 will be used for this research work. STEP and STEP-NC will be discussed in detail in this chapter.

2.2 STEP

STEP (Standard for the Exchange of Product Model Data) is large and powerful set of ISO under ISO 10303. The main purpose of STEP is to provide the detailed information about the product throughout the lifecycle of a product. STEP provides useful data modelling methods and data model focused on industrial uses. The STEP contains several documents for different type of product data [8].

2.2.1 History of STEP

In earlier 1980s, work started on making a new data standard for CAD. The data standard that was released was IGES. The people and organizations working on IGES were aware of its weaknesses. Those people and organization helped in making of new data standard, PDES. IGES and PDES standards were made by United States. The same type of work was in progress in United Kingdom, Germany and France. Keeping in view the efforts of United States, Germany, France and United Kingdom, in 1984 a technical committee was formed by ISO. This committee ISO TC184/SC4 tried to develop a new standard for computerized product data model. The expertise used in making IGES and PDES were utilized in making this new product data exchange model. These efforts resulted in STEP, a new standard for product data exchange model [8].

STEP evolved continuously in its early years and by 1989 STEP focussed on the concept of AP (Application Protocols). First version of STEP AP was made standard in 1994. STEP standard into small individual documents. Each document is known as part of STEP. There are about 120 parts of STEP. The parts of similar types fall in same groups. The groups of STEP is as follows [9]:

1. **Overview and fundamental principles (1)**

   It gives an introduction of STEP Standard. The information of basics of STEP is also discussed in this document.

2. **Description methods (11–19)**

   It provides the information about the modelling language of STEP. There are two languages i.e. EXPRESS and EXPRESS-G.

3. **Implementation methods (21–29)**

   The ways of representing the modelled data are covered in this part of STEP Standard.

4. **Conformance testing methodology and framework (31–39)**

   Generalised concepts of conformance testing are given in this part alongwith actual testing methods and requirements on testing labs and clients.

5. **Integrated generic resources (41–59)**

   Information model for EXPRESS that are of widely used are discussed in this part.

6. **Integrated application resources (101–199)**

   Information model for EXPRESS that are not widely used are discussed in this part.

7. **Application protocols (201–299)**

   The parts of standard that are for industrial use are discussed in this part. Application protocol gives detail about different industrial processes.

8. **Abstract test suites (301–399)**

   Implementation of application protocols can be verified by using the test suites that are discussed in this part. For every application protocol that is numbered 2__ there is test suite numbered 3__.

9. **Application interpreted constructs (501–599)**

   These modules are written in EXPRESS language and it is used to make that part of application protocol that is implementable.

10. **Application modules (1001–1999)**

    This part is like mini application protocols. Application protocol are usually made up of several application modules. Part 203 in second edition of STEP has been made in this way. Using application modules is a more recent architectural approach than using application interpreted constructs and may replace application interpreted constructs [9].

2.3 Background

STEP-NC is a new model for data exchange between CAD/CAM system and CNC machines. A lot of work is being done on STEP-NC and it will replace G&M codes one day. Here is an example which help to understand that how STEP-NC will replace G & M Codes. A designer “X” makes a 3D design in Japan by any CAD software. This file is sent to CAM expert “Y” in France where “Y” generates a process plan by any CAM software. This process plan is sent to operator “Z” in Brazil where the process plan is checked on CNC machine simulator and after validation it is executed on CNC machine. After machining and inspection of part the report is sent to “Z”. This example is shown in figure 2.1 [10]. This is how STEP-NC will replace conventional G&M codes (ISO 6983).

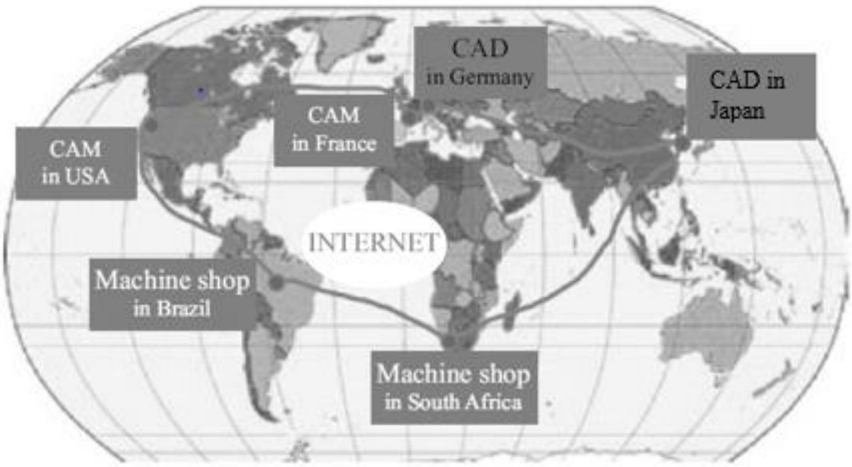


Figure 2.1 STEP-NC way of manufacturing [10]

STEP-NC is being adopted by many companies and institutions around the world. There are some terminologies related to STEP-NC which are described as follows:

2.3.1 STEP-NC

There are two meanings of STEP-NC which are in use around the world. One meaning is in a narrow sense and one meaning is in a broad sense. In narrow sense, STEP-NC is an interface between CAD/CAM system and CNC system. In a broad sense, STEP-NC is not only an interface but also provides methods and technologies to implement this interface across different machining setups. In this research, only narrow sense of STEP-NC will be considered [10].

2.3.1.1 ISO 14649

ISO 14649 is an international standard that defines the data model for STEP-NC. It provides data about the information and ICS (Integrated Computer Solutions) for different manufacturing processes and machine tools [10].

2.3.1.2 STEP-NC Data Model

STEP-NC data model is same as the narrow sense of STEP-NC. It is an interface between CAD/CAM system and CNC machines [10].

2.3.1.3 STEP-NC Compliant CNC

STEP-NC compliant CNC means a new CNC system that accepts the STEP-NC file. The new controller will take STEP-NC file as input and finished machined part will be output [10].

2.3.2 History of STEP-NC

There is a rapid change in manufacturing technology in last two decades. High speed machining, multi-axis control and high-precision manufacturing has increased the production and quality of manufacturing systems. A new concept of e-manufacturing has been introduced by Internet. The part can be designed anywhere in the world and produced in some other part of the world. The conventional G&M codes are the main obstacle in this type of e-manufacturing. Because some of the G&M codes are vendor specific and creates obstacles in transfer of data. These codes were developed when computer technology was limited and machines were controlled offline. Some of the problems of G&M codes are as follows:

2.3.2.1 Information Loss

Part programs based on G&M codes consists of alphabets and numbers. This part program only contains information about the tools path and machine operation. There is no information about the geometry and shape of the path, geometric and dimensional tolerances, tools to be used etc. Usually this part program is generated from some CAD system using

CAM software. And when we look at the part program, it contains a very little information of the CAD file from which it is being generated. So, a lot of information of CAD file is lost in generation of G&M codes [10].

2.3.2.2 Lack of Traceability:

G&M codes contains coded set of information in the form of tools path. It is difficult for operator to decide which tool is to be used and which shape will result after the program is executed by only looking at the program. So, there is lack of traceability in G&M codes [10].

2.3.2.3 Lack of interoperability

G&M codes are normally vendor and the type of controller specific. There are many types of controller for CNC machines like FANUC, SIEMENS, HAAS, HEIDENHAIN etc. These codes vary from controller to controller. For example, in FANUC 21i TB G90 is canned cycle for turning while in SIEMENS 840 DSL G90 is command for absolute system. So, part program of one controller cannot be run on other's controller directly. It should be passed through a machine specific post processor [10].

2.3.2.4 No Feedback

There is no feedback system in G&M codes. If the operator made any change to the program on the machine. The change cannot be traced back in the design. The conventional G&M codes cannot be used with modern manufacturing system like CIM, CAPP directly [10].

These are some of the short comings of G&M codes that helped in the development of new standard for data transfer. The initial work on this standard was started in 1994 in University of Aachen in European Project OPTIMAL (ESPRIT III 8643). In this project, data model for milling was investigated based on STEP. After that other projects were also started in the other parts of world and led to the development of STEP-NC [10].

STEP-NC is a advanced model for the transfer of data between CAD/CAM system and CNC. G&M codes (ISO 6983) only contains information about the toolpath, tool change, coolant ON/OFF, spindle speed and feed. STEP-NC contains large amount of information. The difference between G&M codes and STEP-NC is shown in figure 2.1. STEP-NC contains working steps, feature to be machined, machining operation, machining strategy, tools and complete detail of workpiece [10].

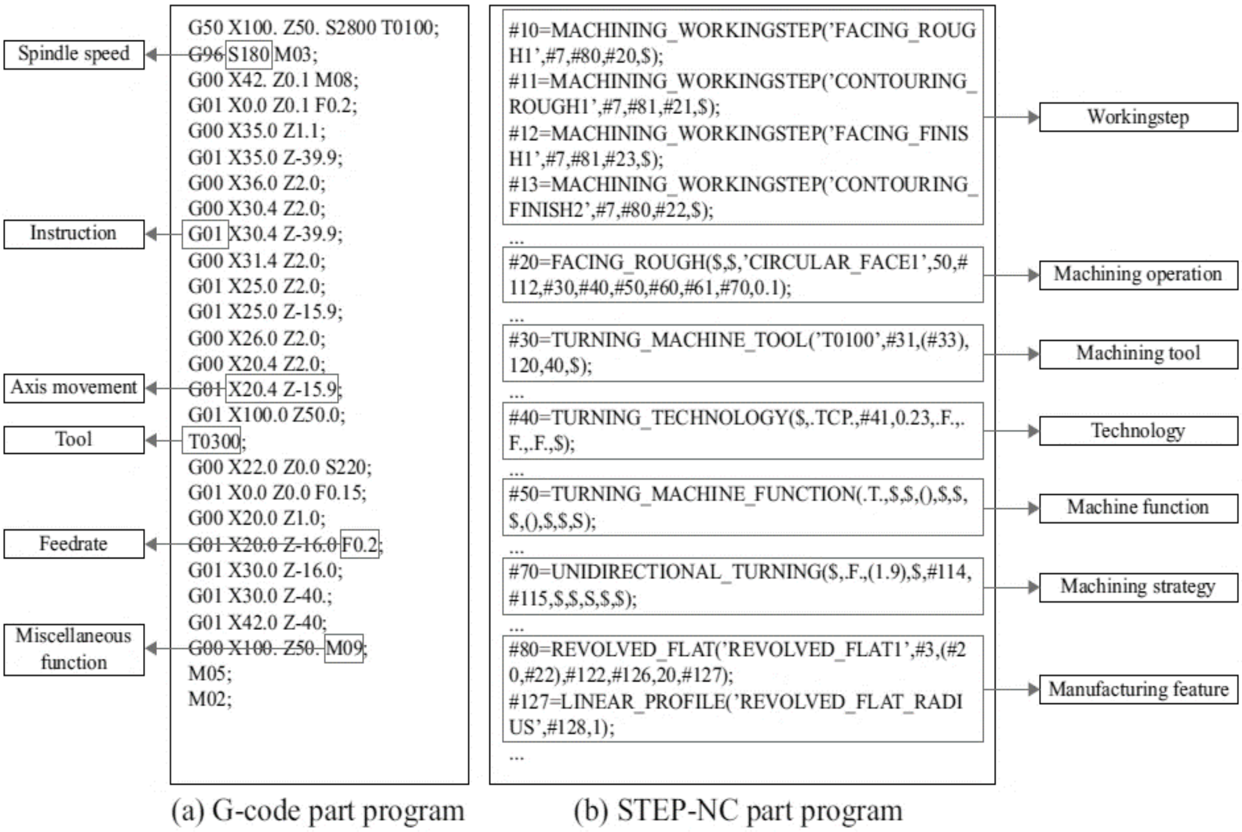


Figure 2.2 Difference between G&M Codes and STEP-NC [10]

2.4 Relationship Between STEP and STEP-NC

STEP-NC utilizes the definition of STEP and modifies it for the use in CNC machines. The relationship between STEP and STEP-NC is shown in figure 2.3

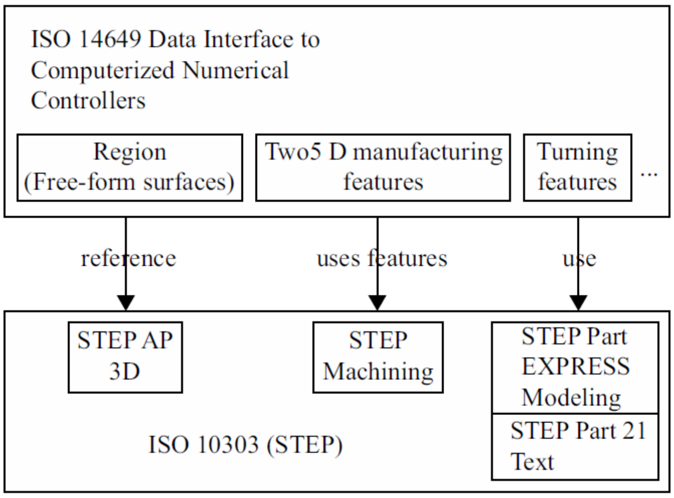


Figure 2.3 Relationship between STEP & STEP-NC [10]

2.5D machining feature in ISO 14649 uses data from STEP ISO 10303 AP 224, free form surface in ISO 14649 uses data from ISO 10303 AP 203 and machining data in ISO 14649 uses data from ISO 10303 part 21.

2.5 ISO 14649

ISO 14649 provides the machining model of data transfer between CAD/CAM system and CNC machines. This standard was formed by ISO Technical Committee ISO/TC 184/SC 1/WG 7. The parts of ISO 14649 are [11]:

1. ISO 14649 Part 1: Overview and Fundamental Principles
2. ISO 14649 Part 10: General Process Data
3. ISO 14649 Part 11: Process Data for Milling
4. ISO 14649 Part 12: Process Data for Turning
5. ISO 14649 Part 13: Process Data for wire-EDM
6. ISO 14649 Part 14: Process Data for sink-EDM
7. ISO 14649 Part 15: Contour Cutting
8. ISO 14649 Part 16: Process Data for inspection
9. ISO 14649 Part 17: Process Data for Rapid Prototyping
10. ISO 14649 Part 110: Machine Tools for General Process
11. ISO 14649 Part 111: Tools for Milling Machines
12. ISO 14649 Part 121: Tools for Turning Machines

2.6 Research Scope

In this research STEP-NC standard will be explored and its capabilities and limitations will be studied for the inspection of parts. A STEP-NC compliant framework is proposed in this research that generates a generalised inspection plan for dimensional inspection of a component. The features that are considered for inspection are:

1. Through Holes Drilled in Rectangular Pattern
2. Step (Shoulder Mill)

The part STEP ISO 10303 AP 219, ISO 14649-16 and DMIS (Dimensional Measuring Interface Standard) will be used in this research for the development of STEP-NC compliant framework. This framework will enable the online inspection of prismatic part and the inspection results will be saved in physical file generated by framework. This framework is validated on the machine FIRST MCV 2000 CNC milling machine with FANUC controller

O*i* MF. The limited concept of CLM (Closed Loop Manufacturing) will be used in this research. CLM is manufacturing of a part on a machine and then the inspection of part on machine. In most CLM systems, CMM (Coordinate Measuring Machines) are used for inspection purpose but there are some problems in using this type of CLM system. These problems are:

1. Inspection through CMM is an offline process. It increases the lead time in machining process
2. Inspection on CMM is carried out in three steps: programming, execution and evaluation of results. These activities sometimes cause interfacing problems as they are carried out on different systems.
3. The inspection results cannot be incorporated in NC programs because of two different systems i.e. CNC and CMM [12].

Figure 2.4 shows the on-machine inspection plan for selected features.

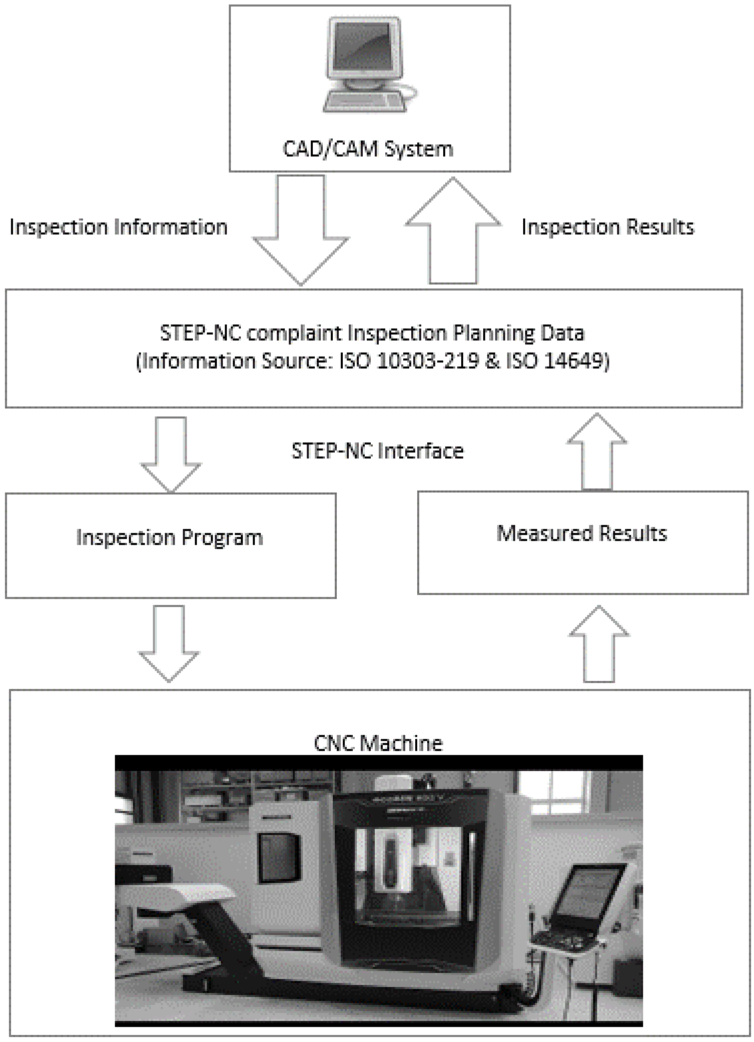


Figure 2.4 On Machine Inspection of Selected Feature

The purpose of using ISO 14649 and ISO 10303-219 is an inspection software platform.

2.7 STEP-NC Compliant Inspection Framework for Prismatic Parts

Inspection planning of a prismatic part has different levels. This includes inspection planning of whole production, inspection planning for a batch and inspection planning for individual part. In this research inspection planning of individual part is discussed. The process of inspection can be divide into several steps:

1. Identification of geometry and shape of the part
2. Identification of tolerances associated with part
3. Identification of inspection resource i.e. CNC machining with probing capability
4. Inspection planning and programming according to the geometry, tolerance and resource
5. Execution of inspection program and analysis of results

The STEP-NC compliant inspection framework is based on product information model and inspection information model. The main aspects of STEP-NC compliant inspection framework are [13]:

1. Product information that include geometry, shape and tolerances
2. Inspection planning that include inspection strategy, inspection procedures, execution of program
3. Inspection results and feedback of results for process control

The description of above mentioned aspects is as follows:

2.7.1 STEP-NC Compliant Product Information Model

The product model gives information about form, shape and feature of the part to be inspected. It also provides the information about the geometric tolerances of the feature to be inspected. This product model is an overlap between design and manufacturing phase. The detail about the shape, geometry and tolerances is provided by STEP and its Applications Protocol.

2.7.2 STEP-NC Inspection Planning

The STEP-NC compliant framework consists of inspection planning information, inspection code generation and inspection results analysis. The inspection plan develops a strategy based on knowledge of product data model and workplan that is provided by ISO 14649. This plan is then converted into an inspection program. A STEP-NC compliant file will be generated by the framework. This file will contain items to be inspected, inspection strategies and working steps. This file will be converted into machine specific file so that the plan can be executed on the machine.

2.7.3 Feedback of Inspection Results

Execution of the program on machine yields inspection results. These results can be interpreted back to the STEP-NC compliant physical file. The results will include dimension of features that are inspected.

2.8 Features Selected to Validate the Framework

The features that are selected for this research are:

1. Rectangular Pattern of through Holes
2. Step Feature (Shoulder Mill)

Both features are shown in figure 2.5.

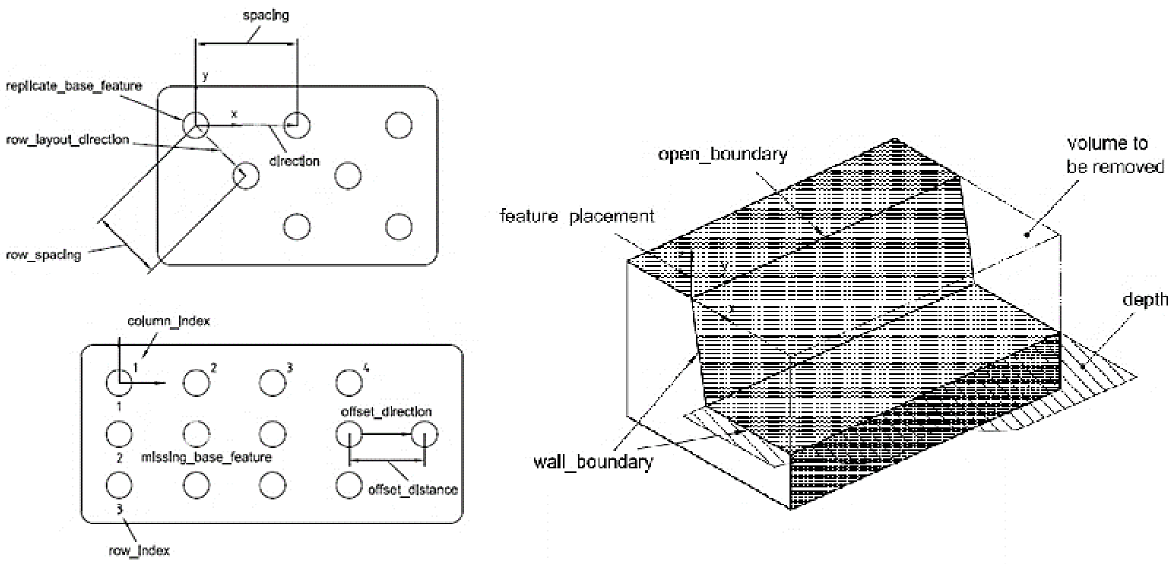


Figure 0.5 Feature Selected for Research [14]

Chapter 3 Inspection Planning and Programming

3.1 Introduction

This chapter highlights the capabilities and limitation of STEP-NC standard and its comparison with other standard used for dimensional inspection of parts. This chapter discusses the limitation of DMIS (now ISO 22093) and reviews the STEP-NC compliant inspection system.

3.2 Standards used for Inspection of Parts

Currently, several standards are in use for dimensional inspection of prismatic parts in manufacturing industry. The most commonly used standards are DMIS, ISO 10303-219, STEP-NC, I++ etc. The detail of the standard used for inspection of parts is as follows:

3.2.1 Dimensional Measurement Inspection Interface

The mostly used standard for measurement on CMM is DMIS (Dimensional Measurement Interface Standard). It is standard for communication of dimensional measurement sequence and is most widely used in CMMs. It is for two-way communication between computer and the machine used for inspection. DMIS is basically an intermediate file. It links CAD system and CMM machine. It is also used as a programming language for the machine which is being used for inspection of the part. The DMIS standard specifies the parameters for the inspection of a part. It also specifies tolerances, datum reference frames, part coordinate system and sequence for machine motion [15].

Dimensional Measuring Interface Specification Project was started in 1985. This project helped in development of DMIS Standard. Manufacturers and users of dimensional measurement equipment participated in the development of new standard. They wanted to develop a system that allow the communication of inspection data between automated system. The first version was published in March 1986, DMIS 1.0. Second version of DMIS was published in September 1987, DMIS 2.0. The American National Standard Institute ANSI accepted DMIS 3.0 in 1995. DMIS 4.0 was published in 2001 and was accepted by ANSI. Development of DMIS was then carried out by ISO and the latest standard of DMIS was published by ISO in 2011 as ISO 22093.

DMIS provide a format for inspection program and inspection results. The inspection program is processed according to DMIS format and then this is executed on the machine. Results are post processed into the CAD system.

The tolerances supported by DMIS are based on ANSI standards. The standard ANSI Y14.5, 1994 is used in DMIS. DMIS 3.0 is an offline version for communication of inspection program and result. Initially DMIS was only used to link between CAD system and CMM but now it has developed into a programming language for inspection [15]. DMIS provides an inspection program for the machine that is used for the inspection purpose. It also yields inspection results and feed these results back to the collection system as shown in fig 4.1.

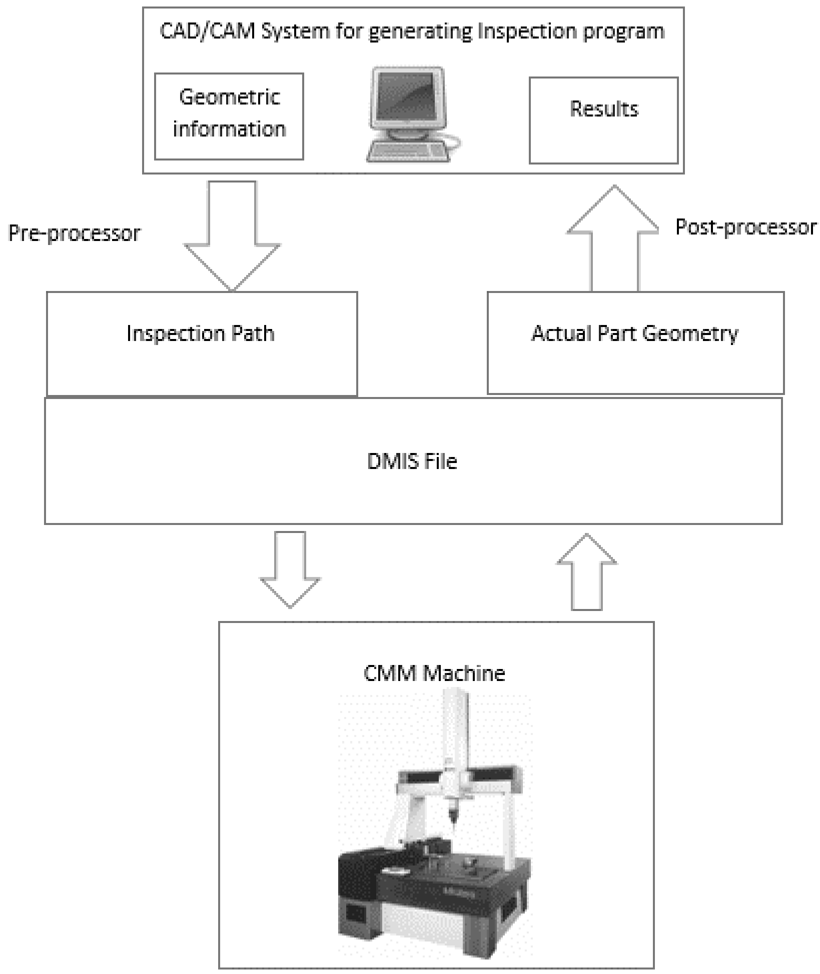


Figure 3.1 Inspection planning and programming using DMIS

The CAD data is passed through a pre-processor to convert the file into DMIS style. The file is sent to the CMM and after the inspection the results are passed through the post-processor and are incorporated in the CAD file.

DMIS provides a readable and writable format for the inspection of the parts. This format can directly be used as programming for the machine which supports DMIS language. An example of DMIS file is given below:

```
F(CIRCLE_1)=FEAT/CIRCLE,INNER,CART,10,10,5,0,0,1,4
MEAS/CIRCLE, F(CIRCLE_1),3
GOTO/10,10,5
PTMEAS/CART,12,10,5,-1,0,0
PTMEAS/CART,8,10,5,1,0,0
PTMEAS/CART,10,12,5,0,-1,0
```

The explanation of the above example is as follows:

F(CIRCLE_1) is the name used for feature

CIRCLE, INNER means that selected feature is a Hole

CART,10,10,5 show x, y & z coordinate

0,0,1 is axis of hole

4 is diameter of hole

The following lines show the points that are to be measured.

3.2.2 STEP & STEP-NC

STEP is name which is used for ISO 10303. It has many individual parts (discussed in chapter 2) and application protocols. STEP-NC is name which is used for ISO 14649. It also many individual parts. It is basically used as product model data for CNC machines. Programming of CNC machines is done with ISO 6983 (G&M codes). ISO 6983 is unable to deal with demands of modern CNC technology (discussed in chapter 2). ISO 6983 program only defines the path of tool and machine ON/OFF instructions while STEP-NC contain much higher-level information. STEP-NC is based on the features that are to be produced as a result of manufacturing. It also contains information of the material that is to be used. It provides information from start to end on the shop floor level. It eliminates the need of post processor between CAD/CAM and CNC. The information can be sent directly to the CNC machine and vice versa. It also standardizes the manufacturing process in a manufacturing enterprise. The same file will be sent to the all manufacturing sites and hence same process plan will be executed at all the site of a manufacturing enterprise. As a result, the parts made will have same tolerances and will be same quality.

Figure 4.2 shows a manufacturing environment according to STEP-NC.

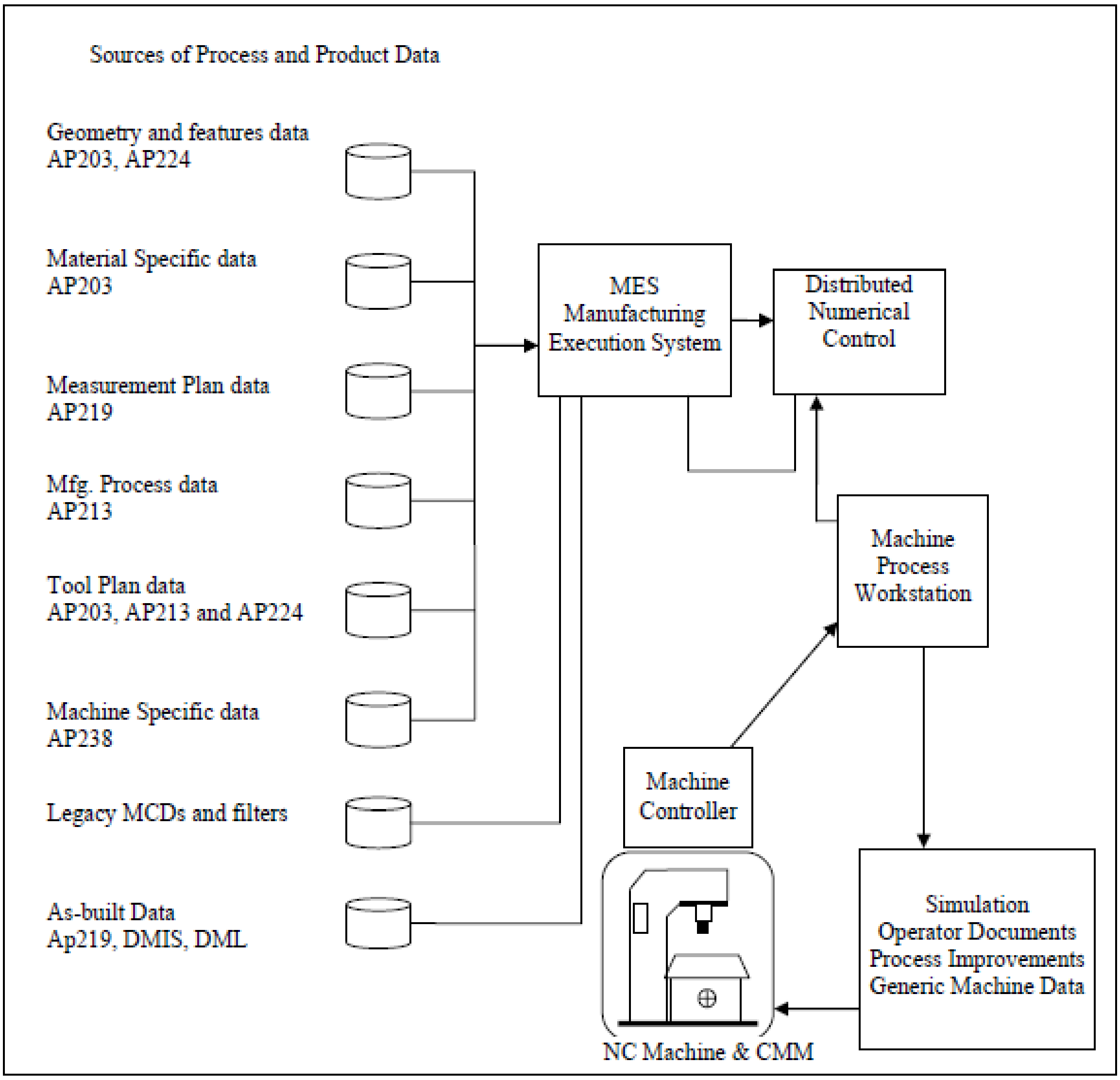


Figure 3.2 STEP-NC manufacturing environment [13]

The STEP-NC product data model is a program in terms of features to be manufactured. In STEP-NC concept of working-steps is used. Working-Steps contain detail of feature and technical information. Detail of feature include feature shape, tolerances etc. technical information includes tool data, machine functions, strategies etc [11]. Working-steps act as building block for program based on STEP-NC. One working-step defines one operation with one tool.

In future, STEP-NC will replace ISO 6983 (G&M codes), where each file must be post-processed and converted to machine specific G & M codes (Figure 4.4). STEP-NC depends on attributes of the features (hole, pocket, pattern, step etc.) and processes (roughing, profiling, finishing etc.) involved in manufacturing. STEP-NC will replace the need of post-processor and creates a new manufacturing environment for design, process planning and the CNC controller.

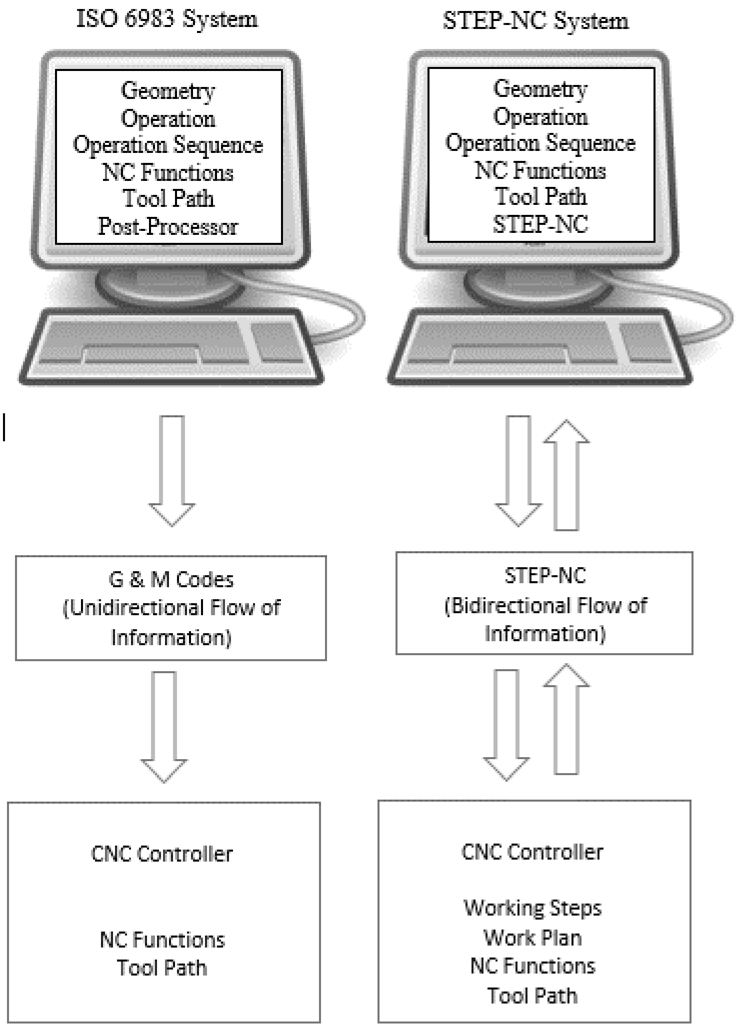


Figure 3.3 Comparison of ISO 6983 and STEP- NC [13]

The research work on STEP-NC around the globe focusses on the integration of design, process planning and manufacturing along with inspection and measurement of the parts. In this research, focus is on the application STEP-NC standard for the inspection and measurement of the prismatic parts. As discussed in chapter 2, the part of ISO 14649 that is responsible for the inspection of prismatic part is 14649-16. ISO 10303-219 also describe inspection features and process but it does not describe the probing working-steps. These steps are defined in ISO 14649.

3.3 Structure of STEP-NC file

STEP-NC based program contains geometric information and technological information. STEP-NC program has many sections. The first section of program is called "HEADER". In header section general information about the part is available. This information includes filename, date, author name, organization etc. The second section of program is known as "DATA". This section contains all the information related to manufacturing like geometry, features, machining operations etc. The data section is further divided into three parts (figure 4.5) [11].

1. Workplan and executables
2. Geometry
3. Manufacturing Features

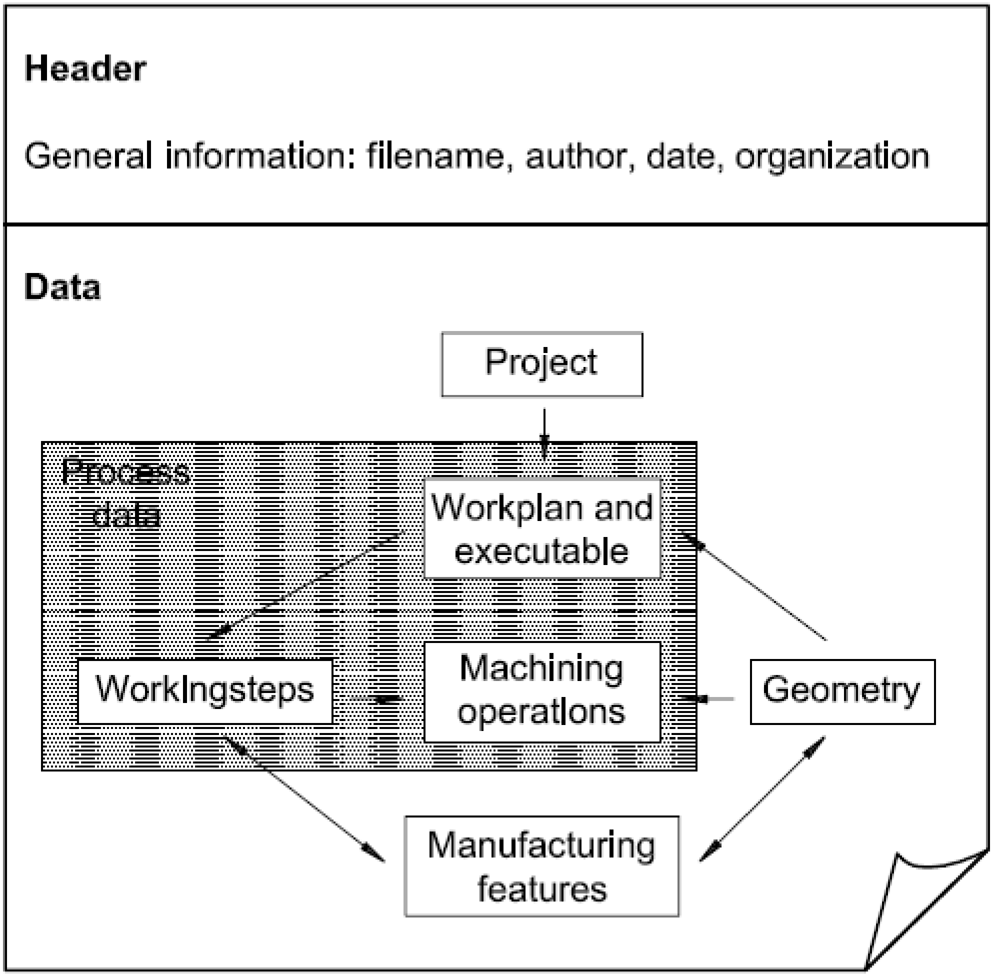


Figure 3.4 Program structure of STEP-NC [11]

The workplan is combination of three steps i.e. working steps, NC function and program structure. The change in sequence of operation can be achieved by changing this part of the program only. The geometry and manufacturing features of the program will remain unchanged [11].

Geometry section of the program utilizes the data of ISO 10303. Since CAD/CAM systems have standardized there data with ISO 10303 so this data is directly used in the program [11].

Manufacturing in the program are defined by mapping of ISO 10303-224 and feature of ISO 14649. This helps in the development of process plan for machining.

3.3.1 Limitations of STEP-NC for Inspection of Parts

ISO 14649-16 contains information about the inspection activities, inspection items, tolerances, reference datum for inspection, inspection results storage and probing strategy. ISO 14649-10 contains information about general process data, project, workplan, workpiece etc. This information of part 10 will be used in making of program using ISO 14649-16. A lot of information is provided in ISO 14649-16 but there are still some limitations. These limitations are:

1. Inspections features are not defined
2. Relation between inspection feature and items does not exist
3. No link between probing working steps and manufacturing features is defined
4. Inspection result statement does not specify where to store inspection results

3.4 STEP-NC Data Model for Inspection of Parts

ISO 14649-16 provides an interface between CNC controller and CAD/CAM system. In figure 4.6, EXPRESS-G diagram is used to describe the functionality of STEP-NC inspection planning for this research work. EXPRESS and EXPRESS-G are the programming language that is defined in STEP (ISO 10303). EXPRESS schemas for inspection are:

3.4.1 Touch Probing

Touch probing defines the probing task and is inserted into the sequence of probing-_operation.

```
ENTITY touch_probing
  ABSTRACT SUPERTYPE OF (ONE OF (workpiece_probing,
  workpiece_complete_probing, tool_probing))
  SUBTYPE OF (Workingstep, OPERATION);
  measured_offset: nc_variable;
END_ENTITY;
```

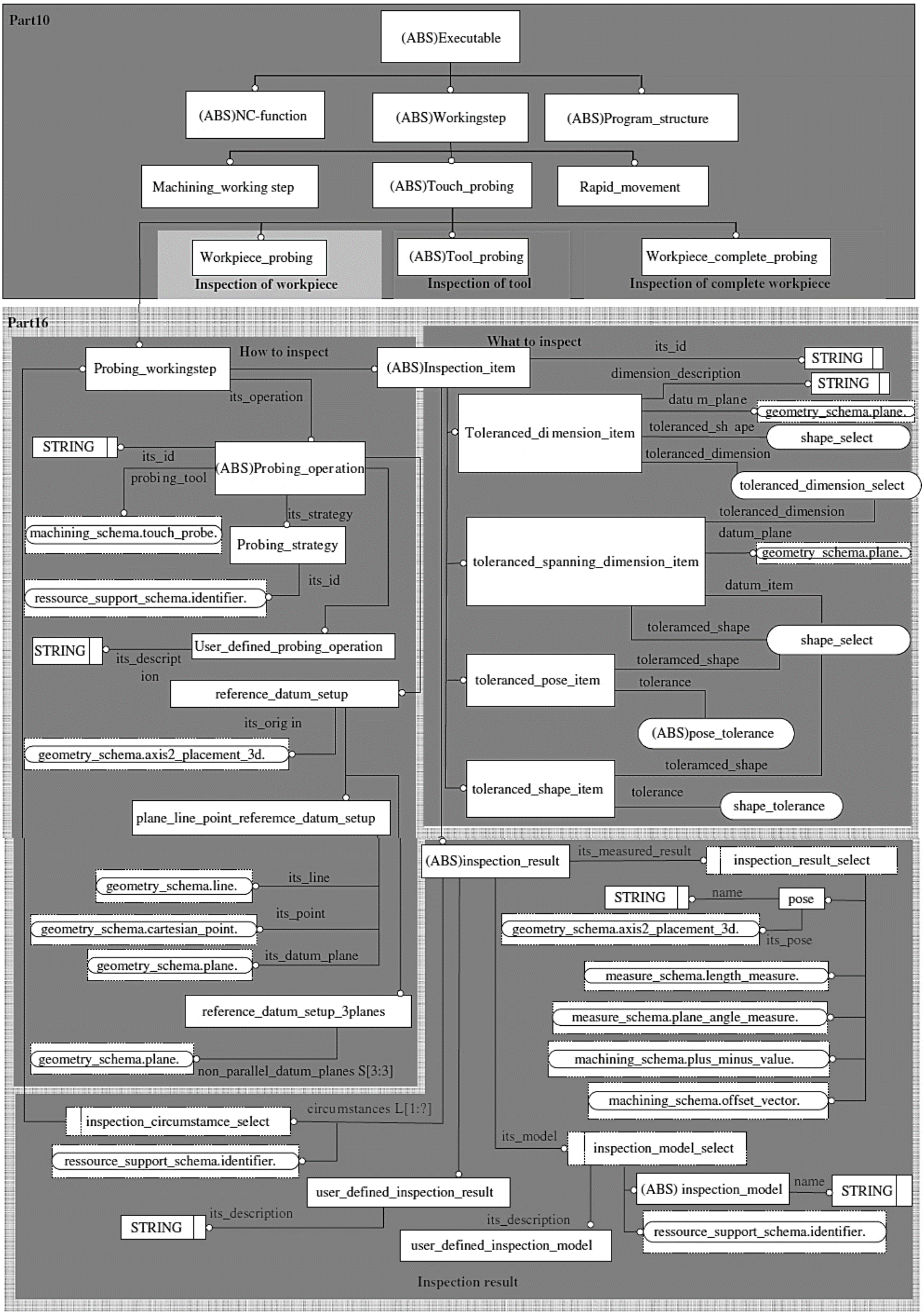


Figure 3.5 EXPRESS-G for inspection of prismatic part[12]

3.4.2 Probing Workingstep

Probing_workingstep probing activity that is to be performed. It contains probing_operation and inspection_item.

```
ENTITY probing_workingstep
  SUBTYPE OF (touch_probing);
  its_operation: probing_operation;
  its_items: SET[1:?]OF inspection_item;
END_ENTITY;
```

3.4.3 Probing Operation

Probing_operation contains circumstances for probing operation and instructions about how to carryout operation.

```
ENTITY probing_operation
  ABSTRACT SUPERTYPE OF (user_defined_probing_operation);
  its_id:STRING;
  reference_datum:reference_datum_setup;
  its_strategy:OPTIONAL probing_strategy;
  probing_tool:touch_probe;
END_ENTITY;
```

3.4.4 Inspection Item

Inspection_item is the entity used for item to be inspected. It has two entities tolerance item and inspection result.

```
ENTITY inspection_item
  ABSTRACT SUPERTYPE OF (ONEOF(toleranced_dimension_item, toleranced_spanning_dimension_
  item, toleranced_pose_item, toleranced_shape_item));
  its_id: STRING;
  toleranced_result: OPTIONAL inspection_result;
END_ENTITY;
```

3.4.5 Inspection Result

Inspection_result store the results of the inspection program.

```
ENTITY inspection_result
  ABSTRACT SUPERTYPE;
  its_measured_result: inspection_result_select;
  its_model: inspection_model_select;
  circumstances: LIST [1:?]OF inspection_circumstance_select;
END_ENTITY;
```

The above-mentioned schemas are related to inspection part of the program. The EXPRESS schemas related to features are:

3.4.6 Workpiece

Entity workpiece is used for the complete description of the workpiece. In this entity, material of workpiece, tolerance of workpiece, raw piece used, geometry of workpiece and clamping positions are defined.

```
ENTITY workpiece;
  its_id:                  identifier;
  its_material:            OPTIONAL material;
  global_tolerance:        OPTIONAL shape_tolerance;
  its_rawpiece:            OPTIONAL workpiece;
  its_geometry:            OPTIONAL advanced_brep_shape_representation;
  its_bounding_geometry:   OPTIONAL bounding_geometry_select;
  clamping_positions:      SET [0:?] OF cartesian_point;
END_ENTITY;
```

3.4.7 Round Hole Feature

This entity is used for the description of both holes and threaded holes. In this entity, diameter of hole, bottom condition of hole and change in diameter is defined.

```
ENTITY round_hole
  SUBTYPE OF (machining_feature);
  diameter:                 toleranced_length_measure;
  change_in_diameter:       OPTIONAL taper_select;
  bottom_condition:         hole_bottom_condition;
END_ENTITY;
```

3.4.8 Rectangular Pattern

This entity defines the rectangular pattern in detail. No. of rows & columns, spacing between rows and columns etc. are defined in this entity.

```
ENTITY rectangular_pattern
  SUBTYPE OF(replicate_feature);
  spacing:                  toleranced_length_measure;
  its_direction:            direction;
  number_of_rows:           INTEGER;
  number_of_columns:        INTEGER;
  row_spacing:              OPTIONAL toleranced_length_measure;
  row_layout_direction:     OPTIONAL direction;
  relocated_base_feature:   SET[0:?] OF rectangular_offset;
  missing_base_feature:     SET[0:?]OF rectangular_omit;
  WHERE
   WR1: ( (SELF.number_of_rows > 1 )
        AND EXISTS(SELF.row_spacing)
        AND EXISTS(SELF.row_layout_direction)
        );
END_ENTITY;
```

3.4.9 Step (Feature)

This entity is used to define the step feature. In this entity, boundary condition for the step, depth, length etc. are defined.

```
ENTITY step
   SUBTYPE OF (machining_feature);
   open_boundary:          linear_path;
   wall_boundary:          OPTIONAL vee_profile;
   its_boss:               SET[0:?] OF boss;
  (*
Informal propositions:
- The entire linear_path lies in the same plane.
*)
END_ENTITY;
```

Chapter 4: IT Framework for STEP-NC Compliant Inspection Plan

4.1 Introduction

In this chapter, in-process inspection of part on CNC machine will be discussed in detail. The program for the inspection of part will be generated by the framework. Input and output of framework will also be discussed in this chapter. In process inspection has many advantages because the part is still on the machine and any change can be done to the part efficiently. The output of the framework will be physical file based on the feature to be inspected. In this research, physical file will be mapped with file based on ISO 6983 and this mapped file will be used for CNC controller. The input parameters for both the files will be same.

Inspection plan for the inspection of part on CNC machines depends upon the controller of CNC machine. A larger number of CNC controllers are present around the world. For this research, DMG CNC milling machine is used. The physical file generated by framework will be mapped with the files based on ISO 6983. This file is used for the execution of inspection program on the machine. The framework will generate two files upon getting input from the user. One file will be STEP-NC compliant and the second file will be according to the controller of the CNC machine. The work is in progress for the development of an intelligent controller which can directly use STEP-NC file for inspection of parts.

In this research rectangular pattern and step (shoulder mill) feature are only considered due to limitation of time. The other features available can be used for future research.

4.2 Aims and Objectives of STEP-NC Compliant Framework

The aims and objectives of the framework is to generate a file that is not machine specific. The framework generates a generalized file that is interoperable. Figure 4.1 depicts the functionality of the framework. The program generated is independent of the CNC controller. It contains following information:

1. Defining Feature Using STEP-NC
2. Dimensional Inspection Requirement of the Feature
3. Generating an Inspection Plan
4. Conversion of STEP-NC into Inspection Code

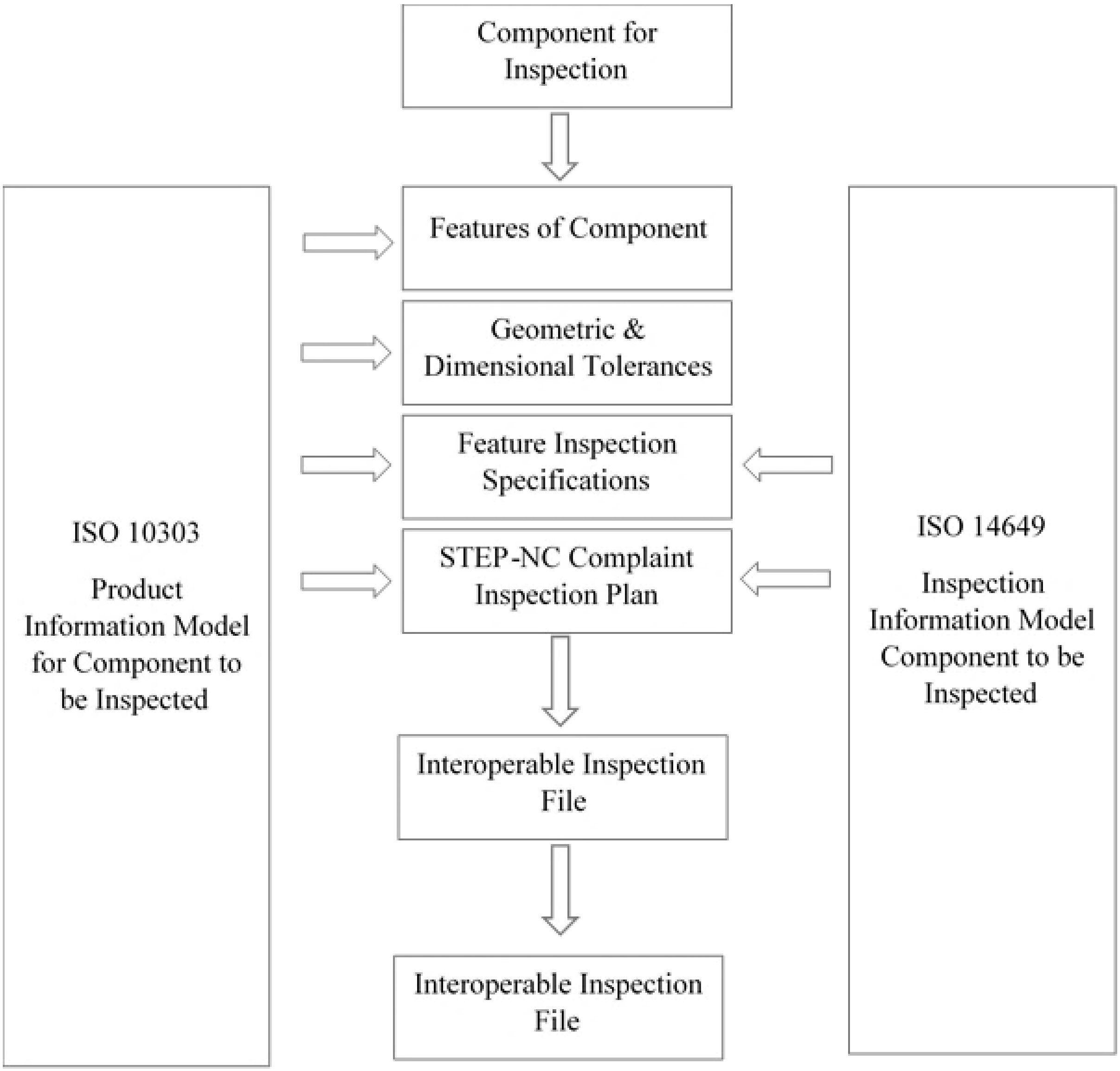


Figure 4.1 Structure of STEP-NC Compliant Inspection Framework

4.2.1 Defining Feature Using STEP-NC

The first step is to define the geometry and shape of the prismatic part to be inspected. Its geometry and shape is defined by the dimensions of the workpiece. Workpiece is defined by using the information from ISO 14649-10. The workpiece information has seven attributes. Five of them are optional. These attributes are:

1. Workpiece identity i.e. unique name for the workpiece
2. Workpiece's material (optional)
3. Global tolerance of the workpiece. It is valid when no other tolerance is defined (optional)

4. Shape of raw material for the workpiece (optional)
5. Geometry of workpiece i.e. dimensions of the workpiece (optional)
6. Bounding geometry of the workpiece i.e. a box or cylinder according to the shape of workpiece (optional)
7. Position of clamping devices

For inspection of the part only three attributes are used. Workpiece identity, workpiece material and clamping positions. These attributes are shown in the STEP-NC compliant file, later in this chapter. In this research, two features are selected. One is rectangular pattern and the other is step. For rectangular pattern, base feature is round hole. These features are already defined in ISO 14649-10. For inspection purpose, ISO 10303-219 is also used.

4.2.2 Dimensional Inspection Requirements of the Feature

The requirements for the inspection of features include defining inspection items associated with the feature, defining reference datum of feature and defining touch probe for the inspection.

4.2.2.1 Defining Inspection Items

The dimensions of the feature to be inspected are specified by dimension item in the STEP-NC (ISO 14649-16) standard. Geometric and dimensional tolerances are also defined in ISO 14649-16. The entity "INSPECTION_ITEM" is used to identify the feature to be inspected. This entity is of three types: 1. "TOLERANCED_DIMENSION_ITEM", 2. "TOLERANCED_SPANNING_ITEM" and 3. "TOLERANCED_SHAPE_ITEM".

Toleranced dimension item entity is used specify general dimension of the feature. It measures the item by applying upper and lower limit to the feature. Toleranced spanning item entity type is used to specify flatness of surface. Shape item entity type is used to specify parallelism etc. of the feature. In this research, TOLERENCED_DIMENSION_ENTITY will be only used.

4.2.2.2 Defining Reference Datum of Feature

The reference datum for the feature to be inspected is the datum from which all the dimensions of the feature are measured. The datum axis and datum plane are established for the part using ISO 14649-10. It can either be three orthogonal planes of the part or a plane, a line or a point.

4.2.2.3 Defining Touch Probe

The touch probe is defined by using entity "TOUCH_PROBE" defined in ISO 14649-10. It defines the length of the stylus of touch probe and the diameter of the stylus.

4.2.3 Generating an Inspection Plan

An inspection plan is generated using ISO 14649-10 and ISO 14649-16. There is some information that is missing in part 10 and 16. This missing information is:

1. Inspection features are not defined separately
2. Inspection items for feature to be inspected
3. Probing working-step is not defined for a feature and inspection item
4. No link between the probing working-step and inspection item

In framework, there is one probing working-step for the base feature present in the part. The manufacturing feature are defined in part 10 of ISO 14649 but attributes for inspection are not defined. The inspection attributes are taken from ISO 10303-219 and DMIS.

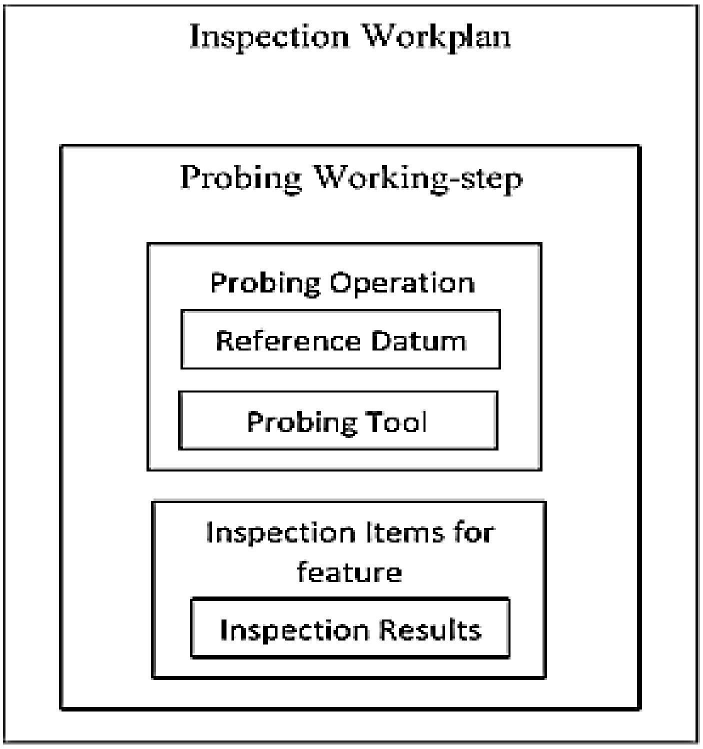


Figure 4.2 Inspection work plan for the feature

An inspection plan is generated after collecting information from part 10 of ISO 14649, AP 219 of ISO 10303 and DMIS. The inspection plan contains executables. These executables have probing working-step for the inspection of feature in the part. The probing working-step contain probing operation and inspection items. The probing operation is user defined and touch probe is used as inspection tool. General for of workplan for inspection of a feature is shown in figure 4.2.

4.2.4 Conversion of STEP-NC into Inspection Code

The STEP-NC file can be converted into a machine specific inspection file for in-process measurement of part. The mapping of STEP-NC file with G&M code is manually achieved. The mapping of file is shown in figure 4.3.

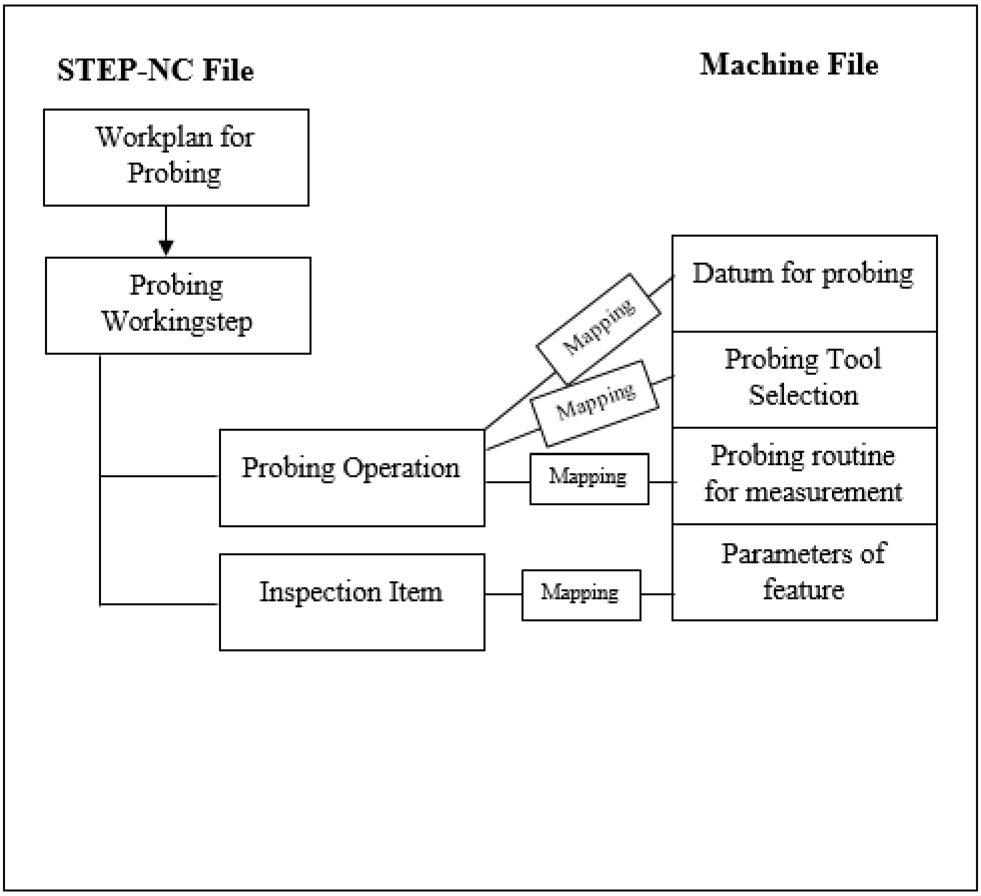


Figure 4.3 Mapping of STEP-NC and Machine Codes

The workplan in a STEP-NC file can have several probing working-steps depending upon feature to be measured. The probing working-step contain inspection items along with probing operation. Probing operation also contain probing tool and datum surface for probing. The tool selection in machine specific file can be mapped with probing operation.

Datum surface is also mapped with probing operation because information about datum surface is present in the probing operation schema. Machine can have a routine for the inspection of the selected feature. This routine is mapped with the probing operation as information about the feature is present in the probing operation. Parameters to be measured are mapped with inspection item. The inspection item entity contains information about the parameters of the feature.

4.3 Generation of STEP-NC file for Rectangular Pattern of Hole Feature

The entity rectangular pattern along with its attributes is discussed in chapter 3. Consider figure 4.4 for the generation of STEP-NC file. Let the workpiece is defined as "workpiece1" and the name of project is "project1". The inspection item selected for this research work is diameter of the hole. The upper and lower limits of the diameter is specified as shown in figure 4.4. The inspection datum is also set for the inspection of the feature as shown is figure 4.4.

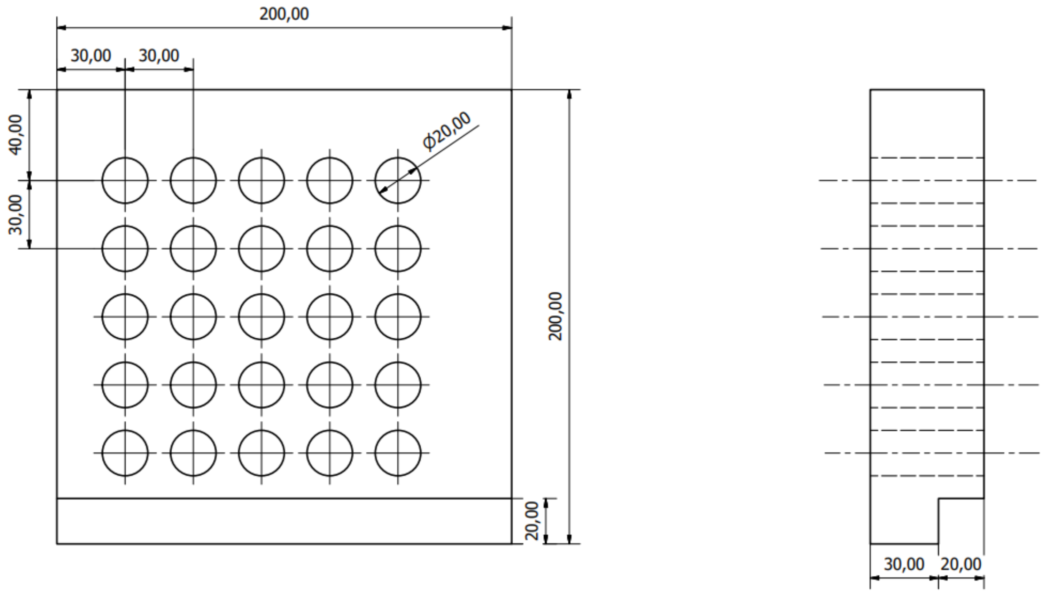


Figure 4.4 Sample drawing to check framework

Probing working-steps for the rectangular pattern of hole are defined. Since only diameter is the selected as feature to be inspected so there is only one probing operation and one-dimension item in the workplan.

All the information mentioned above will help in the generation of STEP-NC file as shown below. Let the diameter of the hole is 20 mm, tolerance is 0.01 and upper and lower limit is 20.01 and 19.99, then STEP-NC file will be:

#1=PROJECT ('project1', #5,(#2),$,$,$);

```
#2=WORKPIECE('workpiece1',#3,.01,$,$,$,(#42,#43,#44,#45));
#3=MATERIAL('A-1024,'Steel',(#4));
#4=PROPERTY_PARAMETER('200000N/mm2')
#5=WORKPLAN('inspection',(#14),$,$,$);
#6=ROUND_HOLE('Round hole 1',#2,(),#8,#7,#9,$,#10);
#7=PLANE('bottom surface',$);
#8=AXIS2_PLACEMENT_3D('A1',#11,#12,#13);
#9=TOLERANCED_LENGTH_MEASURE(20.000,#29);
#10= THROUGH_BOTTOM_CONDITION();
#11=CARTESIAN_POINT('c1',($,$,$));
#12=DIRECTION('axis direction',($,$,$));
#13=DIRECTION('reference direction',($,$,$));
#14=PROBING_WORKINGSTEP('inspectpattern',#15,#16,#17,(#27));
#15=PLANE('secplane',$);
#16=NC_VARIABLE('measured offset',$);
#17=PROBING_OPERATION('measurement operation 1',#32,#30,#31);
#18= RECTANGULAR_PATTERN('PTRNHOLE1',#2,(),#23,#6,#22,#26,5,5,#21,#261,(),());
#19= PLUS_MINUS_VALUE(.02,.02,4);
#20= PLUS_MINUS_VALUE(.02,.02,4);
#21= TOLERANCED_LENGTH_MEASURE(30,#20);
#22= TOLERANCED_LENGTH_MEASURE(30,#19);
#23= AXIS2_PLACEMENT_3D('HOLE1',#24,#25,$);
#24= CARTESIAN_POINT('HOLE1: LOCATION ',(30.00,40.00,0.000));
#25= DIRECTION(' AXIS ',(0.000,0.000,1.000));
#26= DIRECTION(' ROW_DIRECTION',(1.000,0.000,0.000));
#261=DIRECTION(' COLUMN_DIRECTION',(0.000,1.000,0.000));
#27=TOLERANCED_DIMENSION_ITEM(diameter item,#39,'diameter',#9,#18,#28);
#28=PLANE('datum plane',$);
#29=PLUS_MINUS_VALUE(20.01,19.01,4);
#30=PROBING_STRATEGY('probing strategy 1 ');
#31=TOUCH_PROBE('probe1');
#32=PLANE_LINE_POINT_REFERENCE_DATUM_SETUP(#33,#34,#35,#36);
#33=AXIS2_PLACEMENT_3D('A2',#37);
#34=LINE('datum line',#38,);
#35=CARTESIAN_POINT('c2',(0.0,0.0,1.0));
#36=PLANE('Ref datum plane',$);
#37=CARTESIAN_POINT('c3',($,$,$));
#38=CARTESIAN_POINT('c4',($,$,$));
#39=INSPECTION_RESULT(#41,#40,(#14));
#40=INSPECTION_MODEL(Rectangular Pattern Model);
#41=TOLERANCED_LENGTH_MEASURE($,$);
#42=CARTESIAN_POINT('CLAMPING_POSITION1',($,$,$);
#43=CARTESIAN_POINT('CLAMPING_POSITION2', (300.000,150.000,0.000);
#44=CARTESIAN_POINT('CLAMPING_POSITION3', (150.000,300.000,0.000);
#45=CARTESIAN_POINT('CLAMPING_POSITION4', (0.000,150.000,0.000);
```

4.3.1 Description of STEP-NC File

The STEP-NC file contains a lot of statements beginning with “#”. “$” is used to show the optional attributes. The “#1” shows project statement, “#2” describe workpiece and “#5” describes workplan for inspection.

The lines “#6”, “#7”, “#8”, “#9”, “#10”, “#11”, “#12”, “#13” & “#18” describe the feature for inspection. #6 describe the base feature for rectangular pattern. #7 describes the plane where base feature exists. #8 describes the axis of base feature. #9 describes the dimension of base feature. #10 describes the bottom condition of the feature. #11 describe the origin of workpiece. #12 & #13 describe the direction and reference for pattern feature. #18 describe the rectangular feature.

The remaining lines show the inspection requirement and probing working-step for the inspection of the item. The mapping of the STEP-NC file and G&M code file is shown in table 4.1.

Sr. No.	STEP-NC	G&M Codes
1		% O0001 N1 G21 N2 G17 G40 G80 N3 G90 G54 G00 X30. Y-40. Z50
2	#14=PROBING_WORKINGSTEP('inspectpattern',#15,#16,#17,(#27)); #15=PLANE('secplane',$); #16=NC_VARIABLE('measured offset',$); #17=PROBING_OPERATION('measurement operation 1',#32,#30,#31); #18= RECTANGULAR_PATTERN('PTRNHOLE1',#2,(),#23,#6,#22,#26,5,5,#21,#261,(),()); #19= PLUS_MINUS_VALUE(.02,.02,4); #20= PLUS_MINUS_VALUE(.02,.02,4); #21= TOLERANCED_LENGTH_MEASURE(30,#20); #22= TOLERANCED_LENGTH_MEASURE(30,#19); #23= AXIS2_PLACEMENT_3D('HOLE1',#24,#25,$); #24= CARTESIAN_POINT('HOLE1: LOCATION ',(30.00,40.00,0.000)); #25= DIRECTION(' AXIS ',(0.000,0.000,1.000)); #26= DIRECTION(' ROW_DIRECTION',(1.000,0.000,0.000)); #261=DIRECTION(' COLUMN_DIRECTION',(0.000,1.000,0.000)); #27=TOLERANCED_DIMENSION_ITEM(diameter item,#39,'diameter',#9,#18,#28); #28=PLANE('datum plane',$); #29=PLUS_MINUS_VALUE(20.01,19.01,4); #30=PROBING_STRATEGY('probing strategy 1 '); #31=TOUCH_PROBE('probe1');	N4 G43 Z-5. H01 N5 G99 G65 P9810 X30. Y-40. D20. F5000. N6 G91 G65 P9814 D20. Y-40. K4 N7 G65 P9814 D20. X30. N8 G65 P9814 D20. Y40. K4 N9 G65 P9814 D20. X30. N10 G65 P9814 D20. Y-40. K4 N11 G65 P9814 D20. X30. N12 Y-50 K5
3		N13 G80 G90 N14 G28 Z0 N15 G28 X0 Y0 N16 M30 %

4.4 Generation of STEP-NC file for Step Feature

The entity step along with its attributes is discussed in chapter 3. Consider figure 4.4 for the generation of STEP-NC file. Let the workpiece is defined as “workpiece2” and the name of project is “project2”. The inspection item selected for this research work is width of the step feature and depth of step feature. The upper and lower limits of the diameter is specified as shown in figure 4.4. The inspection datum is also set for the inspection of the feature as shown is figure 4.4.

Probing working-steps for the step feature are defined. There are two items that are selected for inspection. Both the inspection items are defined in same working-step.

All the information mentioned above will help in the generation of STEP-NC file as shown below. Let the depth of the step is 20 mm, tolerance is 0.01 and upper and lower limit is 20.01 and 19.99, and width of step is 20mm and upper and lower limit of step width is 20.01 and 19.99 then STEP-NC file will be:

```
#1=PROJECT ('Project2, #5,(#2),$,$,$);
#2=WORKPIECE('Workpiece2',#3,0.01,$,$,$,(#42,#43,#44,#45));
#3=MATERIAL('A-1024','Steel',(#4));
#4=PROPERTY_PARAMETER('200000N/mm2')
#5=WORKPLAN('inspection',(#14),$,$,$);
#6= STEP('Step',#2,(),#8,#7,#$,$,());
#7= ELEMENTARY_SURFACE('STEP1-DEPTH',#18);
#8=AXIS2_PLACEMENT_3D('A1',#11,#12,#13);
#9=TOLERANCED_LENGTH_MEASURE(20,#25);
#10=TOLERANCED_LENGTH_MEASURE(20,#26);
#11= CARTESIAN_POINT('STEP1 PROFILE:LOCATION',(0,0,20));
#12=DIRECTION('axis direction',(0.000,0.000,1.000));
#13=DIRECTION('reference direction',(1.000,0.000,0.000));
#14=PROBING_WORKINGSTEP('inspectstep',#15,#16,#17,(#22,#23));
#15=PLANE('secplane',$);
#16=NC_VARIABLE('measured offset',$);
#17=PROBING_OPERATION('measurement operation 1',#29,#27,#28);
#18= AXIS2_PLACEMENT_3D('Step',#19,#20,$);
#19= CARTESIAN_POINT('Step depth1',(0.000,0.000,20));
#20= DIRECTION(' AXIS ',(0.000,0.000,1.000));
#21= DIRECTION(' DIRECTION ',(1.000,0.000,0.000));
#22=TOLERANCED_DIMENSION_ITEM(depth,#36,'depth',#9,#6,#24);
#23=TOLERANCED_DIMENSION_ITEM(width,#37,'width',#10,#6,#24);
#24=PLANE('datum plane',$);
#25=PLUS_MINUS_VALUE(0.01,0.01,4);
#26=PLUS_MINUS_VALUE(0.01,0.01,4);
#27=PROBING_STRATEGY('probing strategy 1 ');
#28=TOUCH_PROBE('Probe2');
#29=PLANE_LINE_POINT_REFERENCE_DATUM_SETUP(#30,#31,#32,#33);
```

```
#30=AXIS2_PLACEMENT_3D('A2',#34,#35,$);
#31=LINE('datum line',#35,);
#32=CARTESIAN_POINT('c2',(0.0,0.0,1.0));
#33=PLANE('Ref datum plane',$);
#34=CARTESIAN_POINT('c3',($,$,$));
#35=CARTESIAN_POINT('c4',($,$,$));
#36=INSPECTION_RESULT(#39,#38,(#14));
#37=INSPECTION_RESULT(#41,#40,(#14));
#38=INSPECTION_MODEL(depth of step);
#39=TOLERANCED_LENGTH_MEASURE($,$);
#40=INSPECTION_MODEL(width of step);
#41=TOLERANCED_LENGTH_MEASURE($,$);
#42=CARTESIAN_POINT('CLAMPING_POSITION1',($,$,$);
#43=CARTESIAN_POINT('CLAMPING_POSITION2', (0,100,0);
#44=CARTESIAN_POINT('CLAMPING_POSITION3', (50,200,0);
#45=CARTESIAN_POINT('CLAMPING_POSITION4', (200,100,0);
```

4.4.1 Description of STEP-NC File

The “#1” shows project statement, “#2” describe workpiece and “#5” describes workplan for inspection.

The lines “#6”, “#7”, “#8”, “#9”, “#10”, “#11”, “#12” & “#13” describe the feature for inspection. #6 describe the step feature. #7 describes the plane where step feature exists. #8 describes the axis of step feature. #9 describes the width of step feature. #10 describes depth of step feature. #11 describe the origin of step feature with respect to workpiece origin. #12 & #13 describe the direction and reference for step feature.

The remaining lines show the inspection requirement and probing working-step for the inspection of the item. The mapping of the STEP-NC file and G&M code file is shown in table 4.1.

Sr. No.	STEP-NC	G&M Codes
1		% O0002 N1 G21 N2 G17 G40 G80 N3 G90 G54 G00 X0 Y0 Z50
2	#6= STEP('Step',#2,(),#8,#7,#$,$,()); #7= ELEMENTARY_SURFACE('STEP1-DEPTH',#18); #8=AXIS2_PLACEMENT_3D('A1',#11,#12,#13); #9=TOLERANCED_LENGTH_MEASURE(20,#25); #10=TOLERANCED_LENGTH_MEASURE(20,#26); #11= CARTESIAN_POINT('STEP1 PROFILE:LOCATION',(80,0,-5)); #12=DIRECTION('axis direction',(0.000,0.000,1.000)); #13=DIRECTION('reference direction',(1.000,0.000,0.000)); #14=PROBING_WORKINGSTEP('inspectstep',#15,#16,#17,(#22,#23)); #15=PLANE('secplane',$);	N4 G43 Z 2.0 H01 N5 G65 P9770 X10 Y0 Z-10 F5000. N6 G65 P9023 Y20 N7 G0 Z2.0 N8 G65 P9770 X90 Y0 Z-10 F5000. N9 G65 P9023 Y20 N10 G0 Z2.0 N11 G65 P9770 X10 Y10 Z0.

	#16=NC_VARIABLE('measured offset',$); #17=PROBING_OPERATION('measurement operation 1',#29,#27,#28); #18= AXIS2_PLACEMENT_3D('Step',#19,#20,$); #19= CARTESIAN_POINT('Step depth1',(0.000,0.000,20)); #20= DIRECTION(' AXIS ',(0.000,0.000,1.000)); #21= DIRECTION(' DIRECTION ',(1.000,0.000,0.000)); #22=TOLERANCED_DIMENSION_ITEM(depth,#36,'depth',#9,#6,#24); #23=TOLERANCED_DIMENSION_ITEM(width,#37,'width',#10,#6,#24); #24=PLANE('datum plane',$); #25=PLUS_MINUS_VALUE(0.01,0.01,4); #26=PLUS_MINUS_VALUE(0.01,0.01,4); #27=PROBING_STRATEGY('probing strategy 1 '); #28=TOUCH_PROBE('Probe2');	F5000. N12 G65 P9023 Z-20 N13 G65 P9770 X90 Y10 Z0. F5000. N14 G65 P9023 Z-20
3		N15 G80 G90 N16 G28 Z0 N17 G28 X0 Y0 N18 M30 %

Chapter 5: Validating of STEP-NC Compliant Framework for Selected Component

5.1 Introduction

In this chapter, the files generated by framework will be validated on machine. The STEP-NC file is generated by the framework for the selected feature i.e. rectangular pattern or step feature. The G&M code file is also generated by the framework. This G&M code file is then fed to the machine and is checked. The block is machined first and then inspection program is run on the machine.

5.2 IT Framework

The information technology used for this framework is based on php programming (Hypertext Preprocessor) language. It uses web browser for its working. The input side is web browser. The inputs are given and as a result STEP-NC text file and G&M code text files are generated.

5.3 Validation of Inspection File for Rectangular Pattern of Hole

For validation, the rectangular pattern of 3 x 3 is selected. The inputs of framework are shown in figure 5.1 & figure 5.2. The project ID is “Project1”, workpiece material is “Mild Steel”, designation of material is “AISI 1020”, Tensile Strength of material is “420000000 MPa”, workpiece ID is “Sample1”, size of workpiece “X=164”, “Y=164” & “Z=25.4”, global tolerance is “0.02”, detail of base feature is “X=20” & “Y=20”, Diameter of hole is “11”, no. of rows is “3”, no. of columns is “3”, row spacing is “30”, column spacing is “30”, touch probe ID is “Probe1”. These were the main inputs of the framework. Other inputs are along with these inputs are shown in figure 5.1 & 5.2.

When all the inputs are entered in the framework and the “Submit” is pressed two files are generated. One is STEP-NC as shown in figure 5.3 and the second file is shown in figure 5.4 which is G&M Code file.

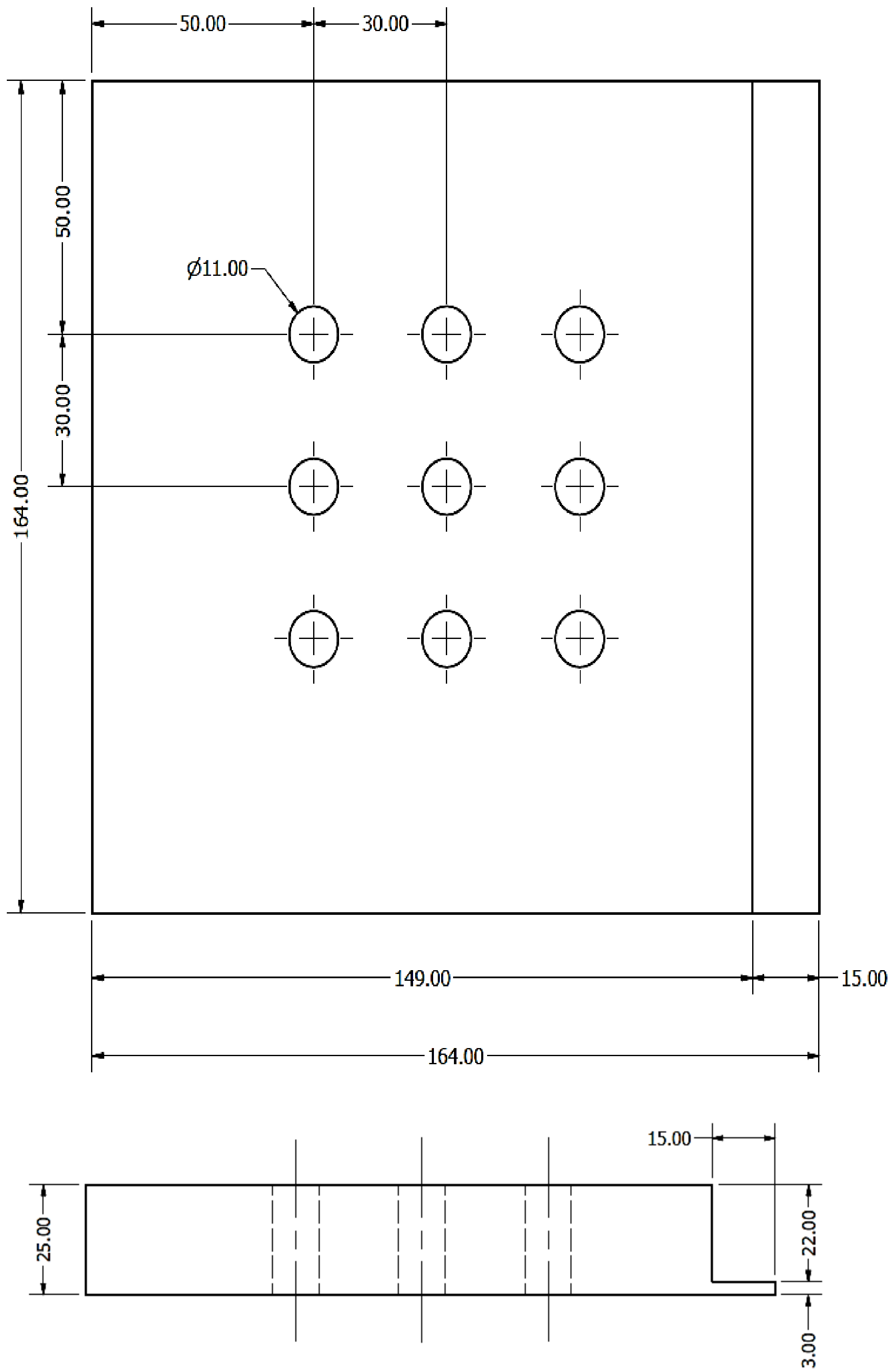
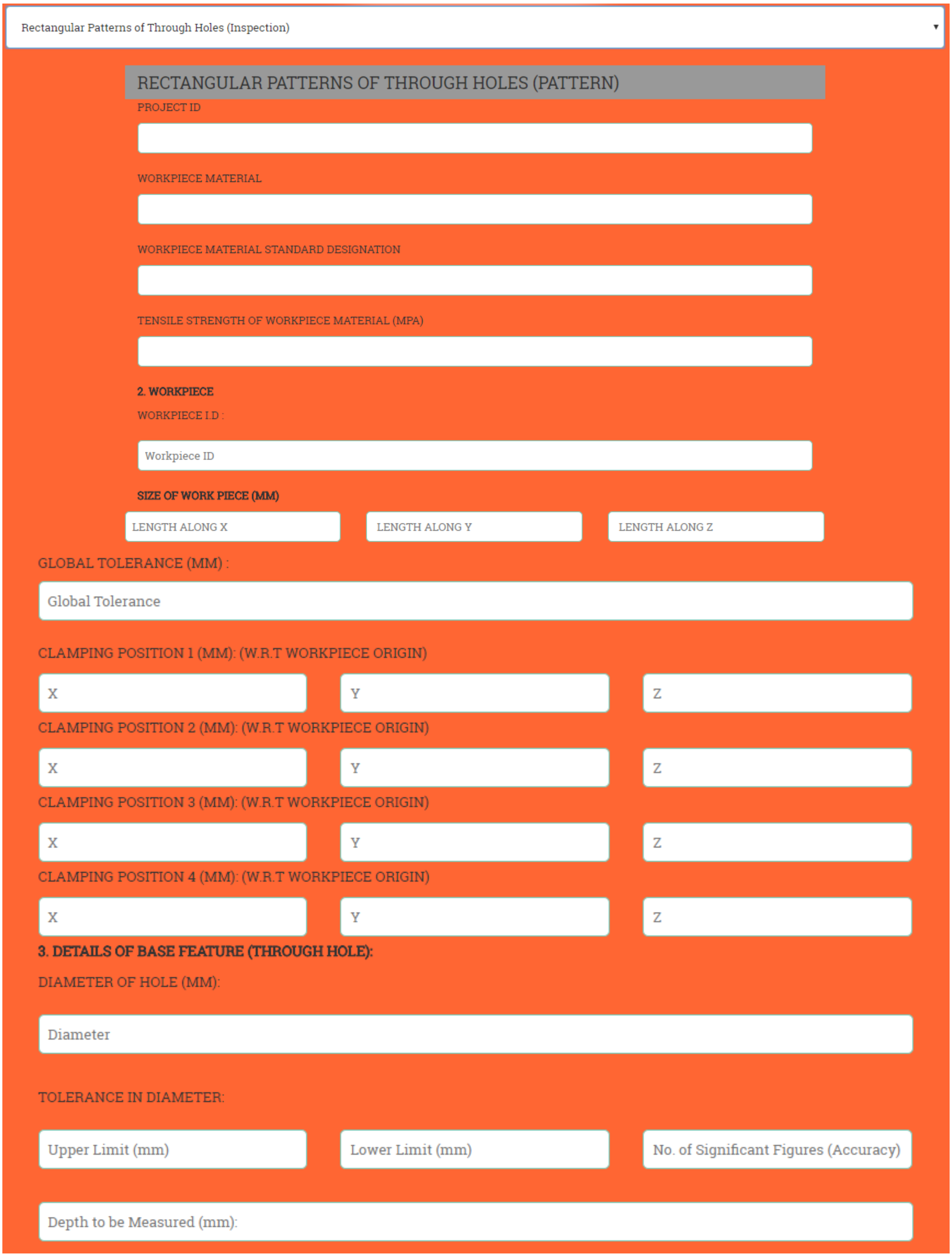


Figure 5.1 Testpiece for validation


Rectangular Patterns of Through Holes (Inspection)

RECTANGULAR PATTERNS OF THROUGH HOLES (PATTERN)

PROJECT ID

WORKPIECE MATERIAL

WORKPIECE MATERIAL STANDARD DESIGNATION

TENSILE STRENGTH OF WORKPIECE MATERIAL (MPA)

2. WORKPIECE

WORKPIECE I.D :

Workpiece ID

SIZE OF WORK PIECE (MM)

LENGTH ALONG X | LENGTH ALONG Y | LENGTH ALONG Z

GLOBAL TOLERANCE (MM) :

Global Tolerance

CLAMPING POSITION 1 (MM): (W.R.T WORKPIECE ORIGIN)

X | Y | Z

CLAMPING POSITION 2 (MM): (W.R.T WORKPIECE ORIGIN)

X | Y | Z

CLAMPING POSITION 3 (MM): (W.R.T WORKPIECE ORIGIN)

X | Y | Z

CLAMPING POSITION 4 (MM): (W.R.T WORKPIECE ORIGIN)

X | Y | Z

3. DETAILS OF BASE FEATURE (THROUGH HOLE):

DIAMETER OF HOLE (MM):

Diameter

TOLERANCE IN DIAMETER:

Upper Limit (mm) | Lower Limit (mm) | No. of Significant Figures (Accuracy)

Depth to be Measured (mm):

Figure 5.2 Input for Rectangular Pattern of Hole 1

Figure 5.3 Input for Rectangular Pattern of Hole 2

```
#1=PROJECT('Testpiece',#5,(#2),$,$,$);
#2=WORKPIECE('workpiece1',#3,.02,$,$,$,(#42,#43,#44,#45));
#3=MATERIAL('AISI 1020','Mild Steel',(#4));
#4=PROPERTY_PARAMETER('420000000');
#5=WORKPLAN('inspection',(#14),$,$,$);
#6=ROUND_HOLE('Round hole 1',#2,(),#8,#7,#9,$,#10);
#7=PLANE('bottom surface',$);
#8=AXIS2_PLACEMENT_3D('A1,#11,#12,#13);
#9=TOLERANCED_LENGTH_MEASURE(11,#29);
#10=THROUGH_BOTTOM_CONDITION();
#11=CARTESIAN_POINT('c1',($,$,$));
#12=DIRECTION('axis direction',($,$,$));
#13=DIRECTION('reference direction',($,$,$));
#14=PROBING_WORKINGSTEP('inspectpattern',#15,#16,#17,(#27));
#15=PLANE('secplane',$ );
#16=NC_VARIABLE('measured offset',$);
#17=PROBING_OPERATION('measurement operation 1',#32,#30,#31);
#18=RECTANGULAR_PATTERN('PTRNHOLE1',#2,(),#23,#6,#22,#26,3,3,#21,#261,(),());
#19=PLUS_MINUS_VALUE(.02,.02,4);
#20=PLUS_MINUS_VALUE(.02,.02,4);
#21=TOLERANCED_LENGTH_MEASURE(30,#20);
#22=TOLERANCED_LENGTH_MEASURE(30,#19);
#23=AXIS2_PLACEMENT_3D('HOLE1',#24,#25,$);
#24=CARTESIAN_POINT('HOLE1: LOCATION ',(50.000,50,0.000));
#25=DIRECTION(' AXIS ',#37);
#26=DIRECTION(' ROW_DIRECTION',(1.000,0.000,0.000) );
#261=DIRECTION(' COLUMN_DIRECTION',(0.000,-1.000,0.000));
#27=TOLERANCED_DIMENSION_ITEM(diameter_item,#39,'diameter',#9,#18,#28);
#28=PLANE('datum plane',$);
#29=PLUS_MINUS_VALUE((11.02,10.98,4));
#30=PROBING_STRATEGY('probing strategy 1');
#31=TOUCH_PROBE('probe1');
#32=PLANE_LINE_POINT_REFERENCE_DATUM_SETUP(#33,#34,#35,#36);
#33=AXIS2_PLACEMENT_3D('A2',#37,$);
#34=LINE('datum line',#38);
#35=CARTESIAN_POINT('c2',(0.0,0.0,1.0));
#36=PLANE('Ref datum plane',$);
#37=CARTESIAN_POINT('c3',($,$,$));
#38=CARTESIAN_POINT('c4',($,$,$));
#39=INSPECTION_RESULT(#41,#40,(#14));
#40=INSPECTION_MODEL(Rectangular Pattern Model);
#41=TOLERANCED_LENGTH_MEASURE($,$);
#42=CARTESIAN_POINT('CLAMPING_POSITION1',(82,0,0));
#43=CARTESIAN_POINT('CLAMPING_POSITION2',(82,164,0));
#44=CARTESIAN_POINT('CLAMPING_POSITION3',(0,82,0));
#45=CARTESIAN_POINT('CLAMPING_POSITION4',(,,));
```

Figure 5.4 STEP-NC File generated for Rectangular Pattern of Hole

5.3.1 Result of Inspection of Rectangular Pattern of Hole

The G&M code file generated by framework is checked by running the program on the machine. The results of inspection are shown in table 5.1

```
N1G21
N2    G17   G40    G80
N3    G90   G54    G00    X50    Y50    Z50
N4    G43   Z10    H01
N5    G99   G65    P9810  X50    Y50    D11    F5000
N6    G91   G65    P9814  D11    X30    K2
N7    G65   P9814  D11    Y-30
N8    G65   P9814  D11    X30    K2
N9    G65   P9814  D11    Y-30
N10   G65   P9814  D11    X30    K2
N11   G80   G90
N12   G28   Z0
N13   G28   X0     Y0
N14   M30
```

Figure 5.5 G&M code file generated by Framework

Sr. No.	Column 1	Column 2	Column 3
Row 1	11.011	11.009	11.010
Row 2	11.005	11.008	11.010
Row 3	11.009	11.008	11.007

Table 5.1 Inspection results of Rectangular Pattern of Hole

5.4 Validation of Inspection File for STEP Feature

For validation of step feature, the drawing is shown in figure. The inputs of framework are shown in figure 5.5 & figure 5.6. The project ID is “Project2”, workpiece material is “Mild Steel”, designation of material is “AISI 1020”, Tensile Strength of material is “420000000 MPa”, workpiece ID is “workpiece2”, global tolerance is “0.02”, detail of step feature is “X=0”, “Y=20” & Z=-20, touch probe ID is “Probe2”. These were the main inputs of the framework. Other inputs are along with these inputs are shown in figure 5.5 & 5.6.

When all the inputs are entered in the framework and the “Submit” is pressed two files are generated. One is STEP-NC as shown in figure 5.7 and the second file is shown in figure 5.8 which is G&M Code file.

Figure 5.6 Input for Step Feature 1

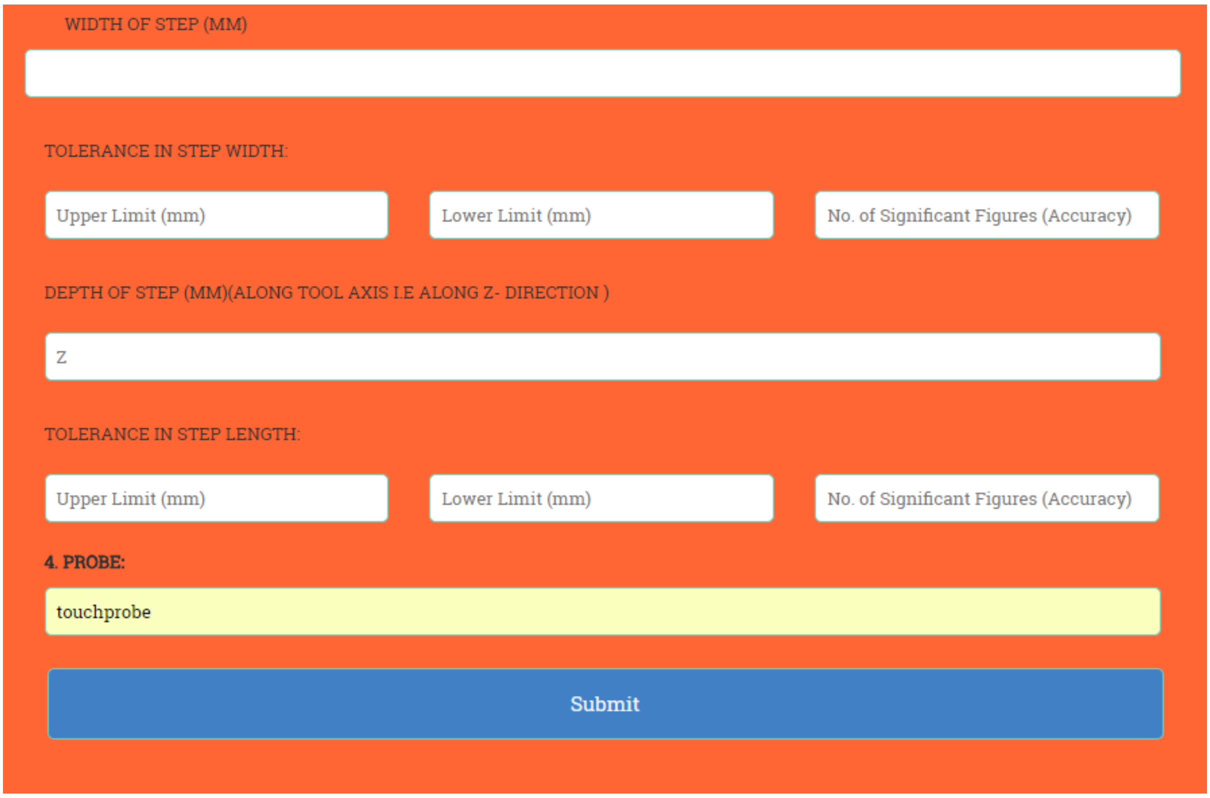


Figure 5.7 Input of Step Feature 2

5.4.1 Results of Inspection for Step Feature

The items selected for inspection of step feature is width of step and depth of step. The width is inspected at two selected pints as shown in framework. Similarly, depth is also measured at two selected points as shown in framework. The results of inspection are shown in table 5.2.

Both the files are generated using framework are tested on3-axes CNC Milling FIRST MCV 2000 with controller FANUC O*i* MF. Different file where generated with different inputs. All the files validated the STEP-NC compliant framework. The generated G&M codes files are only valid for the machines with FANUC O*i* MF controller. G&M codes are controller and machine specific. But STEP-NC file generated are same irrespective of the controller.

```
#1=PROJECT('Testpiece',#5,(#2) ,$,$,$);
#2=WORKPIECE ('workpiece1',#3,0.02,$,$,$,(#42,#43,#44,#45));
#3=MATERIAL('AISI 1020','Mild Steel',(#4));
#4=PROPERTY_PARAMETER('420000000');
#5=WORKPLAN('inspection',(#14),$,$,$);
#6=STEP('Inspect Step',#2,(),#8,#7,$,$,());
#7=ELEMENTARY_SURFACE('STEP1-DEPTH',#18);
#8=AXIS2_PLACEMENT_3D('A1',#11,#12,#13);
#9=TOLERANCED_LENGTH_MEASURE(20,#25);
#10=TOLERANCED_LENGTH_MEASURE(15,#26);
#11=CARTESIAN_POINT ('STEP1 PROFILE:LOCATION',(149,-164,0));
#12=DIRECTION('axis direction'  ,(0.000,0.000,1.000));
#13=DIRECTION('reference direction',(1.000,0.000,0.000));
#14=PROBING_WORKINGSTEP('inspectstep',#15,#16,#17,(#22,#23));
#15=PLANE('secplane',$ );
#16=NC_VARIABLE('measured offset',$);
#17=PROBING_OPERATION('measurement operation1',#29,#27,#28);
#18=AXIS2_PLACEMENT_3D('Step',#19,#20,$);
#19=CARTESIAN_POINT ('Step depth 1',(0.000,0.000,20));
#20=DIRECTION('AXIS',(0.000,0.000,1.000));
#21=DIRECTION('DIRECTION',(1.000,0.000,0.000));
#22=TOLERANCED_DIMENSION_ITEM(depth,#36,'depth',#9,#6,24);
#23=TOLERANCED_DIMENSION_ITEM(width,#37,'width',#10,#6,#24);
#24=PLANE(('datum plane',$));
#25=PLUS_MINUS_VALUE(20.02,19.98,4);
#26=PLUS_MINUS_VALUE(15.02,14.98,4);
#27=PROBING_STRATEGY('probing strategy 1');
#28=TOUCH_PROBE('probe1');
#29=PLANE_LINE_POINT_REFERENCE_DATUM_SETUP(#30,#31,#32,#33);
#30=AXIS2_PLACEMENT_3D('A2',#34,#35,$);
#31=LINE(''datum line'',#35,);
#32=CARTESION_POINT('c2',(0.0,0.0,1.0));
#33=PLANE('ref datum plane',$);
#34=CARTESION_POINT('c3',($,$,$));
#35=CARTESION_POINT('c4',($,$,$));
#36=INSPECTION_RESULT(#39,#38,(#14));
#37=INSPECTION_RESULT(#41,#40,(#14));
#38=INSPECTION_MODEL(depth of step );
#39=TOLERANCED_LENGTH_MEASURE($,$);
#40=INSPECTION_MODEL(width of step);
#41=TOLERANCED_LENGTH_MEASURE  ($,$);
#42=CARTESION_POINT('CLAMPING_POSITION1',(82,0,0));
#43=CARTESIAN_POINT ('CLAMPING_POSITION2',(82,164,0));
#44=CARTESIAN_POINT ('CLAMPING_POSITION3',(0,82,0));
#45=CARTESIAN_POINT ('CLAMPING_POSITION4',(,,));
```

Figure 5.8 STEP-NC file for Step Feature

```
N1    G21
N2    G17   G40    G80
N3    G90   G54    G00    X149   Y-164
N4    G43   Z2.0   H01
N5    G65   P9770  X159   Y-154  Z-7.5  F5000.
N6    G65   P9023  X149
N7    G0    Z2.0
N8    G65   P9770  X159   Y-10   Z-7.5  F5000.
N9    G65   P9023  X149
N10   G0    Z2.0
N11   G65   P9770  X159   Y-154  Z0     F5000.
N12   G65   P9023  Z-15
N13   G65   P9770  X159   Y-10   Z0.    F5000.
N14   G65   P9023  Z-15
N16   G80   G90
N17   G28   Z0
N18   G28   X0     Y0
N19   M30
%
```

Figure 5.9 G&M Code File for Step Feature

Sr. No.	Width	Depth
Point 1	15.007	20.003
Point 2	15.011	20.005

Table 5.2 Inspection Results of Step Feature

Chapter 6: Conclusions and Future Research

In this chapter the conclusions of the research work and future directions for research in this field are discussed.

6.1 Conclusions

Following conclusions are drawn from the research work;

1. A web based I.T Framework is developed for feature based inspection of milling parts on CNC Milling Machine. The milling features selected for this research are;
   a. Rectangular Pattern of Through Holes
   b. Step Feature
2. In this research work, the application of STEP-NC (ISO 14649) compliant physical file generated by STEP-NC compliant I.T Framework by mapping it to G & M codes for FANUC Series O*i*-MF Controller.
3. STEP-NC standard is a powerful product data model that can integrate CAD/CAM with CNC without the need of post processors and manufacturing data loss.
4. The STEP-NC compliant I.T Framework developed in this research for inspection of milling parts on CNC Milling Machine, has accomplished the milestone of providing higher level generic product information which is independent of machine used for manufacturing.
5. This research has shown the integration of manufacturing features with machining process on CNC Milling Machine through STEP-NC compliant I.T Framework.
6. The development of STEP-NC compliant I.T Framework has enabled less skilled workers who cannot use CAD/CAM applications to generate part programs for CNC Milling Machine just by entering description of machining feature and workpiece in I.T Framework.

6.2 Future Directions

In view of author, future research in following direction can be carried out in this field;

1. The I.T Framework developed in this research is an open system and does not contain feedback. In future, I.T framework with complete feedback loop for automated correction of errors in STEP-NC physical files can be developed.
2. The I.T Framework developed in this research is only for rectangular pattern of through holes and step feature. I.T framework for other features like slot, boss, chamfer, circular pocket etc. can also be developed.
3. Integration of this STEP-NC Complaint I.T Framework with CNC milling machining centers having existing controllers which cannot interpret STEP-NC physical files, so that feature based machining can be performed on these machines as well.

REFERENCES

[1] Anonymous, "The History of Computer Numerical Control (CNC) – CNC.com." [Online]. Available: http://www.cnc.com/the-history-of-computer-numerical-control-cnc/. [Accessed: 12-Apr-2018].

[2] Y. Yusof and K. Latif, "A novel ISO 6983 interpreter for open architecture CNC systems," *Int. J. Adv. Manuf. Technol.*, vol. 80, no. 9–12, pp. 1777–1786, 2015.

[3] N. Kassim, Y. Yusof, and M. Z. Awang, "The Development of New STEP-NC Code Generator (Milling STEP Coder)," *Appl. Mech. Mater.*, vol. 465–466, pp. 667–671, 2014.

[4] X. W. Xu and S. T. Newman, "Making CNC machine tools more open, interoperable and intelligent—a review of the technologies," *Comput. Ind.*, vol. 57, no. 2, pp. 141–152, 2006.

[5] X. W. Xu and Q. He, "Striving for a total integration of CAD, CAPP, CAM and CNC," *Robot. Comput. Integr. Manuf.*, vol. 20, no. 2, pp. 101–109, 2004.

[6] X. W. Xu, L. Wang, and Y. Rong, "STEP-NC and function blocks for interoperable manufacturing," *IEEE Trans. Autom. Sci. Eng.*, vol. 3, no. 3, pp. 297–307, 2006.

[7] M. Hardwick, "Digital manufacturing using STEP-NC," *Tech. Pap. - Soc. Manuf. Eng. MS*, no. MS02-252, pp. 1–20, 2002.

[8] X. Xu and A. Y. C. Nee, *Advanced Design and Manufacturing Based on STEP*. 2009.

[9] Standard ISO, "ISO 10303-1:1994," *Ind. Autom. Syst. Integr. -- Prod. data Represent. Exch. -- Part 1 Overv. Fundam. Princ.*, 1994.

[10] S. H. Suh, S. K. Kang, D. H. Chung, and I. Stroud, *Theory and Design of CNC System*. 2008.

[11] Standard ISO, "ISO 14649-1:2003," *Ind. Autom. Syst. Integr. -- Phys. device Control -- Data Model Comput. Numer. Control. -- Part 1 Overv. Fundam. Princ.*, 2003.

[12] F. Zhao, X. Xu, and S. Xie, “STEP-NC enabled on-line inspection in support of closed-loop machining,” *Robot. Comput. Integr. Manuf.*, vol. 24, no. 2, pp. 200–216, 2008.

[13] L. Ali, S. T. Newman, and J. Petzing, “Development of a STEP-compliant inspection framework for discrete components,” *Proc. Inst. Mech. Eng. Part B J. Eng. Manuf.*, vol. 219, no. 7, pp. 557–563, 2005.

[14] Standard ISO, “ISO 10303-219:2007,” *Ind. Autom. Syst. Integr. -- Prod. data Represent. Exch. -- Part 219 Appl. Protoc. Dimens. Insp. Inf. Exch.*

[15] Standard ISO, “ISO 22093:2011,” *Ind. Autom. Syst. Integr. -- Phys. device Control -- Dimens. Meas. Interface Stand.*