Design and Modelling With Solidworks Application and Program Generation of A Triclover

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Abstract- *This paper aims to the developed software application for product design and its CAD model updating by automating repetitive tasks using SolidWorks application. A case study of Tri claver is a mechanical product for which a program using Numerical control coding. Such developed application is integrated with SolidWorks CAD package through application programming interface. Developed application is having front end and back end, the front end having a graphical user interface through which, input design data/parameter is taken from the user. Backend having internal program according to this design of CNC machine is obtained in output. Output design result is passed to SolidWorks CAD package, which updating CAD models of machine and its manufacturing drawing. CAD modelling hence overall cost of the design is also reduced. The user need not require CAD modelling skills and design knowledge of product highly.*

Keywords- SolidWorks, CAD, Programme, CNC, etc.,

I. INTRODUCTION

Drawing and SolidWorks are written to educate and assist students, designers, engineers, and professionals in the drawing and detailing tools of Solid Works. Investigate the learning procedure through a sequence of design situation, industry scenarios, projects, and objectives target towards the beginning to intermediate Solid Works user. Work through various activities to generate multiple-view, multiple-sheet, complete drawings, and assembly drawings. Develop Drawing templates, Sheet formats, and Custom Properties. Construct drawings that incorporate part configurations, assembly configurations, and design tables with equations. Manipulate remarks in parts, drawings, assemblies, Revision tables, Bills of Materials and more**.** SolidWorks is a mechanical design automation software package used to build parts, assemblies and drawings that take advantage of the familiar Microsoft Windows graphical user interface. SolidWorks is an easy to learn design and analysis tool, which makes it possible for designers to sketch 2D and 3D concepts quickly, create 3D parts and assemblies and detail 2D drawings. In SolidWorks, you create 2D and 3D sketches, 3D parts, 3D assemblies and 2D drawings. The part, assembly and drawing documents are related. Additional information on SolidWorks and its family of products can be obtained at their URL. Drawing refers to the SolidWorks module used to insert, add, and modify views in an engineering drawing. Detailing refers to the robust work model.

II. PROBLEM DEFINITION

1. Tri-clover fitting has commonly used the connection between fluid lines that need to be connected and disconnected frequently. We are designed with the help of SolidWorks .it is designed is comfortable and safe.
2. Also, this design process is currently done on experience; experienced personnel are needed every time for designing. In this involves selection of standard parts like gearbox, motor, bearing, hardware like nut, bolt, screw etc. There is no standard design procedure adopted after the design calculation is completed.
3. It is time-consuming to make CAD models of a machine part and it's manufacturing drawing which also requires skill
4. To develop methodology and to integrate above application with SolidWorks CAD package to set automatically updated versions of CAD model and manufacturing drawing

III. OBJECTIVES OF THE WORK

To develop standard method or procedure for designing a component.

To develop application, this can automate the design process. It means to store or formulate past design experience, knowledge, a procedure in a computer database, which can be used repetitively again and again. To develop methodology and to integrate above application with SolidWorks CAD package to set automatically updated versions of CAD model and manufacturing drawing

IV. MODELLING DEPARTMENT OVERVIEW

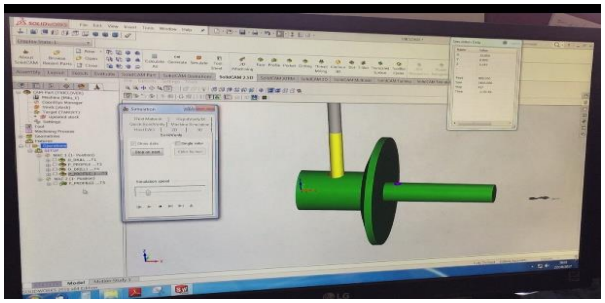
Production in Unnathi CNC technologies is directly connected with its modelling team which consists of three members who drafts and simulate the components. They will always be in contact with the production line in charges and operators for the smooth flow of production. Drafting and the simulation are done by the use of different software packages like ZWCAD (for drafting) and SOLIDCAM (for simulation).ZWCAD is similar to AutoCAD which is used for drafting (converting the 3D into views). These drawings are given to operators for their convenience during production. Solid CAM is used to simulate the 3D model while generating the codes.

4.1 Procedure for program generation

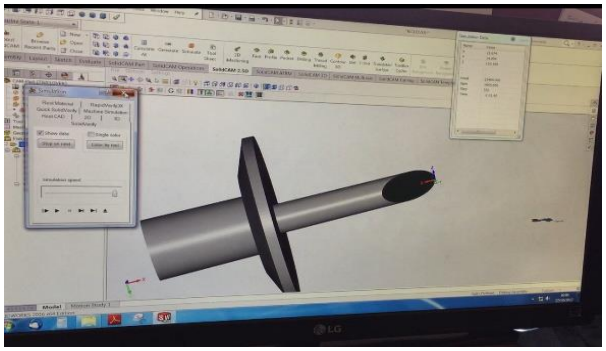
In most of the cases modelling file (modelled component) will be sent to the modelling team by the company which ordered the component, or at least they will send a relevant drawing. The modelling team will check the possibilities of producing the component with production in charges (s) like whether the components can be produced with the available machines and resources. Once the modelling team gets the approval of producing the component from the production in charge they will start planning. First of all start with checking tools, if any individual tools are used. If they are not available, then they will order it. Later they will plan the procedure for production by noting the expected cycle times (turning 1st, turning 2ndmilling). Once all this procedure is gone through they will start simulating the component with the help of Solid CAM software, and the code is generated. If there are any corrections to be done in the code, the no operators will alter it for themselves.

V. SOLIDCAM OPERATING PROCEDURE

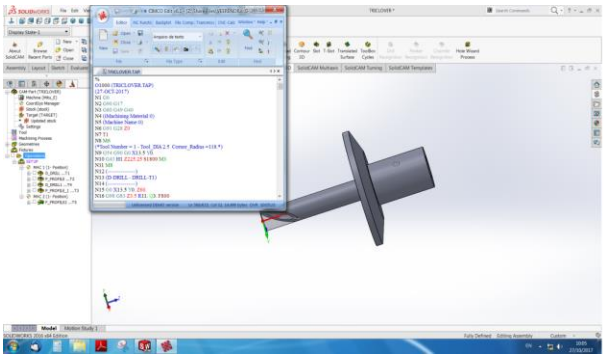
Step1: the Solid model is to be imported.
Step 2**:** We have to do turning operations, in that we have done the first turning-1 operation as follows:
Step 3: After completion of turning one we have to perform the turning-2 operation, and we can see as follows:
Step 4: Now we have to generate G-codes for turning and same as to be saved in a file for reference to CNC machining. As of now the programming for turning is completed. Now programming for milling at the required positions is to be performed according to the given drawing.
Step 5: Now we have to perform milling for TRI CLOVER piece milling 1 & milling two as follows:
Milling 1:-We have to perform on the head and execute the code;



Milling 2: we have to perform bottom side of tri clover and execute the program.



Step:-6 Now we have to generate G-Codes for both milling 1 and milling 2.



About Solid CAM: Solid CAM is the leading Integrated CAM software - seamlessly integrated into Solid works and Autodesk Inventor. Solid CAM is the only CAM with the unmatched, patented iMachining solution. Solid CAM & machining provide unbelievable savings and increased efficiency in CNC milling and turning operations.

- Solid CAM provides the high value of the revolutionary and amazing iMachining technology.
- iMachining saves 70% and more in CNC machining time
- iMachining extends cutting tool life dramatically
- The unique iMachining Technology wizard provides optimal feeds and speeds, taking into account the tool

path, stock and tool material and machine specifications.

- Solid CAM runs directly inside SOLID WORKS or Autodesk inventor CAD system, with seamless integration and full toolpath associatively - design changes initiate automatic tool path updates to the CAD model.
- With the single-window integration, all machining operations can be defined and verified without leaving the parametric CAD assembly environment you are already used to.
- Provides a complete CAM solution for all CNC applications
- Seamless integration in both SOLIDWORKS and Autodesk Inventor.
- Save time and frustration: Eliminates import/export challenges.

5.1 Machine setting:

The first step involved in machining is machine setting. Machine setting is making the machine suitable for producing the particular component, be it the tools or the jaws. Further explanation is based according to HSK A-63.All HSK holders have the same taper hence the first turning is same for all HSK shanks, So in first turning the setting is the same for all HSK components and hence there is no requirement for grinding or boring of jaws. Just by adjusting the diameter we can do the turning 1. The tools required for first turning are listed below table 1.

Table: 1 Tools required for the first turning of HSK-A63 shank.

S.No	Parameter	Dimensions
1	Flange OD	63.3 mm (-0.05)
2	Length	31.75 mm (-0.05)
3	VG Position	18.15 mm (+0.05)
4	VG Ball Dia	73.3 mm (-0.05)
5	VG Width	3.75 mm (+0.15)
6	VG Dia	55.0 mm (-0.1)
7	Taper Major Dia	48.35 mm (-0.05)
8	Thread length	12 mm (-0.15)
9	Taper(OD) stock	0.3 mm (-0.05)
10	Depth	12 mm (-0.15)
11	Length from face to ID	10 mm (-0.1)
12	ID Angle stock	0.3 mm (+/- 0.02)
13	ID Dia	33.7 mm (-0.05)
14	Groove profile ID	40.05 mm (+0.05)
15	U/C ID	18.5 mm (+0.1)
16	U/C ID width	3 mm
17	U/C Dia	34.15 mm (+0.05)
18	ID Profile Dia	37 mm
19	U/C Face depth	0.25 mm (+0.05)
20	U/C Width	3.1 mm (+/- 0.1)
21	U/C Depth	0.45 mm (+0.1)
22	U/C Angle	15°
23	OD Taper angle	2°52'5"

VI. RESULT AND DISCUSSION

Using above developed application, it reduces 90 present times required for the overall design process hence significant amount of saving in cost. Following table shows a comparison of the timeline between conventional and automated method. The manual method 5-6 days 2 hours Using this application 40 minutes are required for design calculation process and generating updated versions of CAD models hence overall cost reduction is 20 to 30%. As there is no 100 percent design and drawing automation is achieved. In this 1.30 hrs required for drawing sheets checking because the design is based on customer requirement sometimes drawing views are scale down or scale up this have to do manually.

VII. CONCLUSION

In the present investigation to study the design procedure of Tri claver is development with the help of SolidWorks graphical interface. The comparison of manual method and SolidWorks model has developed it taken very less time, and cost factors are decidedly less in the graphical user interface. The HSK A-63 also studied the physical dimension with tool holders. SolidWorks is best suitable for the tri-clover design.

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