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Virtual Product Development
Adjustable Wrench

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1 Introduction

The Main aim of this assignment is creating a product virtually to prove the following

- Understanding on the topics such as parametric and feature modeling.
- Based on the given constraints selecting the manufacturing process.
- Prove the understanding on file exchange standards.

The product selected for the modeling is **Adjustable spanner** also known as crescent wrench.
The report consists of three main section first section explains about the modeling of the component in Siemens NX software, second one talks about the selection of manufacturing process, and the third section talks about the file exchange standards.

2 Modeling of the product

This section talks about how the product is modeled in the Siemens NX software.
During modeling of product following things are held in mind

1. Shape element should not only acts like a geometric feature it should have technical information. For example a hole should not only be like a circle it should have features of hole.
2. The same model should be made compatible to be used in different departments for eg it should be used for design optimization and it should be used for manufacturing of the part. So while drawings are issued for different departments the changes that needed be done on the model should be minimum.

The above said requirements are met with the following techniques

- **Parametric Modeling** instead of defining the part length a constant value it is defined as a parameter so one model can be used for creating different variants of the same product. This is very much needed in designing for example if we want to optimize the thickness of a product by testing with finite element model for varying thickness, then different models with varying thickness is required instead if the thickness of the product is made as a parameter then it can be varied by the software and analysis can be done. Thus a single model can serve the purpose.
- **Feature modeling** features are shape elements with technical information embedded in it For example if a part has a hole in it instead of drawing a circle and extruding *drill* option is used this option is useful in CAPP (computer Aided Process Planning) since it is a hole the program can easily identify it need to be drilled (depending on the tolerance values) and it will create a plan with drilling as one of the operations.

The product adjustable wrench has 4 parts

- main body
- moving jaw
- screw
- pin

all these four parts need to be modeled in parametric sense.

2.1 Part Modeling

This section talks about how each part is modeled and how while modeling the above said requirements are met.
Here **bottom up** modeling methodology is followed that is each part is drawn separately and then assembled in an assembly environment.

2.1.1 Part 1 Main Body

From a manufactures data sheet some of the dimensions are taken [1] . The figure(1) shows the location of the measurement point in the component. And the figure(2) shows the dimension values of 8 different models they provide for the customers. In order to make one model for all their products, parametric modeling of the parts is required this is done by the following way.

- All the possible values of each and every parameter is made into a list and stored in expression of NX.
- A variable named *model_no* is introduced (this is the only variable that need to be changed to get a different model of the part).
- All the other parameter values are acquired form the list the parameter value that corresponds to the *model_no*.

Another possible way is to link the parameters in the NX with a excel file which calculates all the required parameters by using the following steps

- From one main parameter (here maximum size of the bolt head that can be hold) parameter **A** all other parameters are derived by Using the table value and fitting a polyline of order 2 all the other parameters can be calculated. The derived relation between variables is shown in figure(4).
- These parameters are then can be extracted to the NX expression

This process has following advantages

- Instead of making only for particular set of variant types we can have infinite number of variants. So this model is better if a design analysis need to be done.

- By this method we can involve expression relation between parameter which are more than order1 (NX expression does not allow for Quadratic or cubic expression if the two parameters are of same dimension(example length of the part) .

Figure(3) shows the how parameters are included in the part drawing.

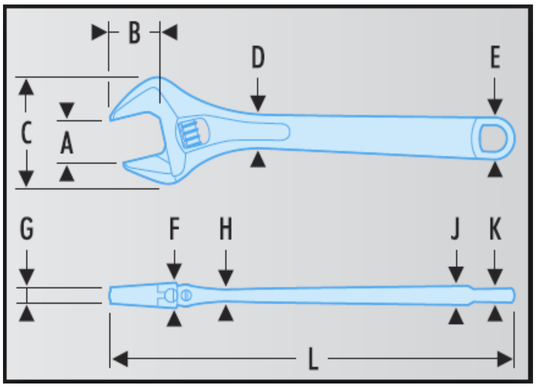


Figure 1: Datasheet showing measurement location of various parameters [1]

Item code	A...	B...	C...	D...	E...	F...	G...	H	J...	K...	L...	Mass (
113A.4C	13	13	34	11	14	9.5	4.5	5	6.5	5	114	65
113A.6C	20	19	45	14.8	18.5	11.8	5	6	8.3	5.8	155	125
113A.8C	27	24	61.5	18.5	24	13.5	6.5	9	11	8.5	206	275
113A.10C	30	29	69.5	21.5	28.5	15.5	9	9.5	12.7	10	255	480
113A.12C	34	34	80	25	31	18	10	10.2	13	10	306	700
113A.15C	44	44	97	28	35	20	13.5	12	14.8	11.5	380	1146
113A.18C	53	53	115	31	38.5	22.5	16	14.3	17.8	13.5	456	1792
113.24C	62	61	132	42	38	31	17	17	15	15	612	3265

Figure 2: Database showing various parmeter values [1]

	Name	Formula	Value	Units	Dimensionality	Type	Source
1				mm	Length	Number	
2	a	{13, 20, 27, 30, 34, 44, 53, 62}	{13,20,27,30,34...			List	
3	b	{13,19,24,29,34,44,53,61}	{13,19,24,29,34...			List	
4	c	{34, 45, 61.5, 69.5, 80,97,115,132}	{34,45,61.5,69....			List	
5	curr_width	nth(model_no, a)	13	mm	Length	Number	
6	d	{11,14.8,18.5,21.5,25,28,31,42}	{11,14.8,18.5,2...			List	
7	e	{14,18.5,24,28.5,31,35,38.5,38}	{14,18.5,24,28....			List	
8	f	{9.5,11.8,13.5,15.5,18,20,22.5,31}	{9.5,11.8,13.5,1...			List	
9	g	{4.5,5,6.5,9,10,13.5,16,17}	{4.5,5,6.5,9,10,...			List	
10	h	{5,6,9,9.5,10.2,12,14.3,17}	{5,6,9,9.5,10.2,...			List	
11	j	{6.5,8.3,11,12.7,13,14.8,17.8,15}	{6.5,8.3,11,12.7...			List	
12	k	{5,5.8,8.5,10,10,11.5,13.5,15}	{5,5.8,8.5,10,10...			List	
13	l	{114,155,206,255,306,380,456,612}	{114,155,206,25...			List	
14	Length	nth(model_no,l)	114	mm	Length	Number	
15	max_width	nth(model_no,c)	34	mm	Length	Number	
16	model_no	1	1		Constant	Number	

Figure 3: Setting up of Parameter values in the NX environment

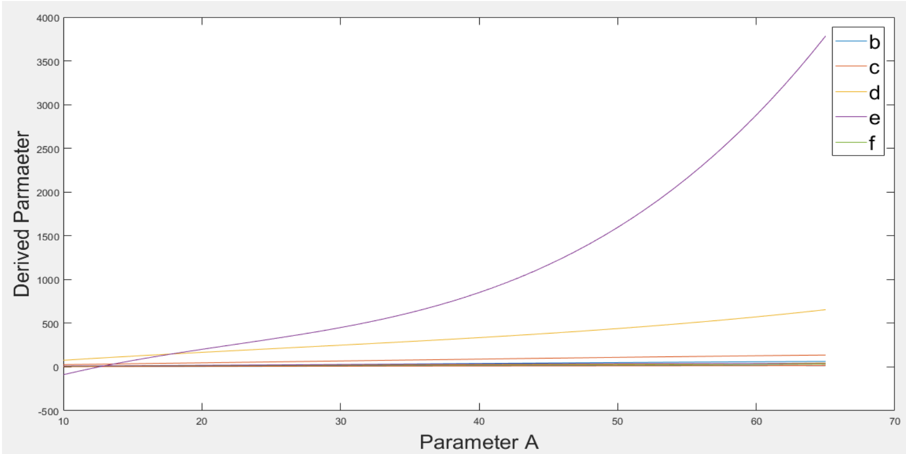


Figure 4: Relation between variables

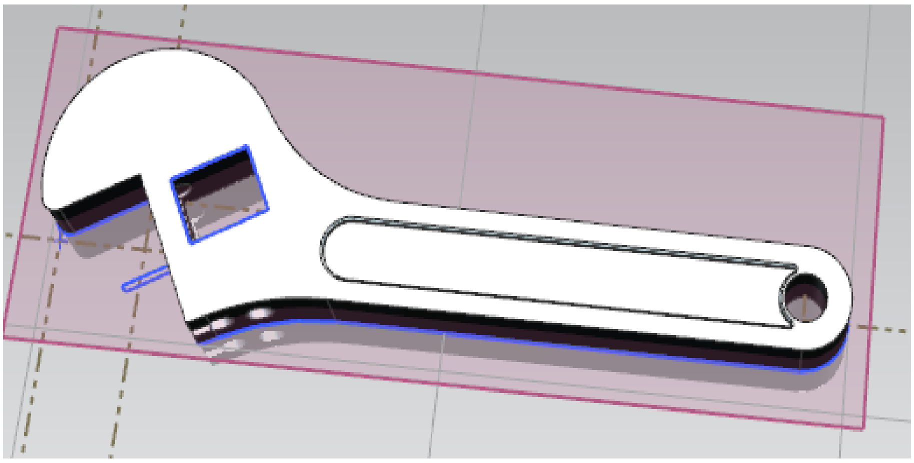


Figure 5: Completed part1

figure(5) shows the completed view of part 1. While making part 1 following things are made to ensure the parametric modelling

- All dimensions are made in reference with the parameters defined.
- While making any feature in the component the parts features are used thinking of manufacturing process behind it. for example instead of drawing circle and extrude, *hole* feature is used.
- All the extrude option the end point is given with reference to the surface end.
- While making any sketch it is ensured that the sketch is fully constrained and there is no auto constrained available and the dimensions for constraining is also a parameter.

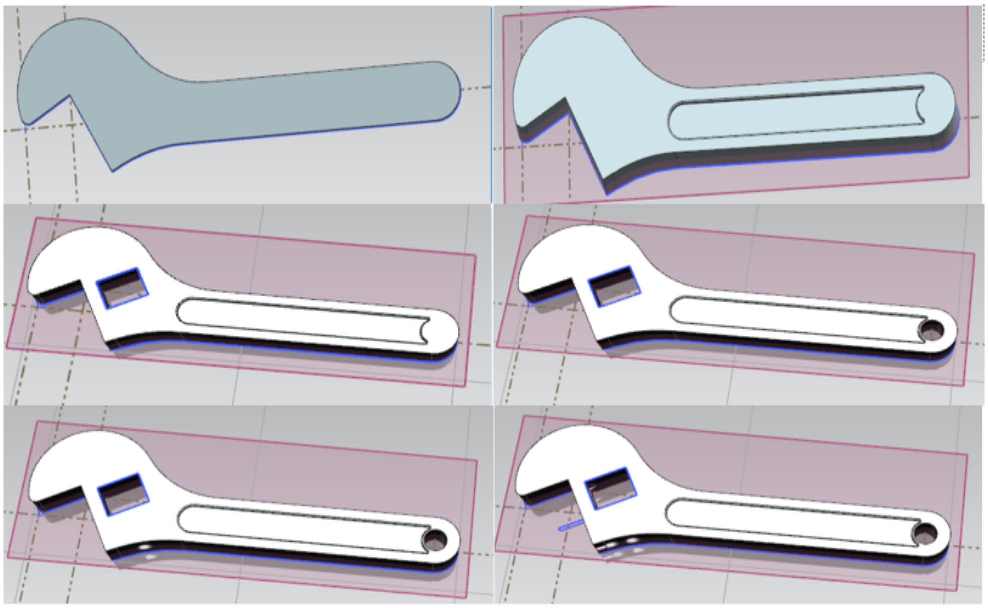


Figure 6: Steps followed in making part1

The steps followed in making part 1 are shown in the figure(6)

- First from a square block piece the part is cut in the shape all the dimensions in the figure (including the dimensions for constraints) are made from parameters.

- For the drill at the handle **drill** feature is used diameter is selected form the parameter list and the drill depth is given till the face instead of specifying the drill depth this is required for parametric modeling as this thickness changes for each and every model.This is shown in figure(7). And also the centre point of the drill is constrained with a distance which is a parameter which depends on the model number of the part.

- Cut is made for the screw by extrude and subtract cut depth is given till the bottom face.

- After drawing section it is cut for moving jaw movement while extruding it is extruded till the face.

- Drill is made for the pin insertion till the side face of the main part.

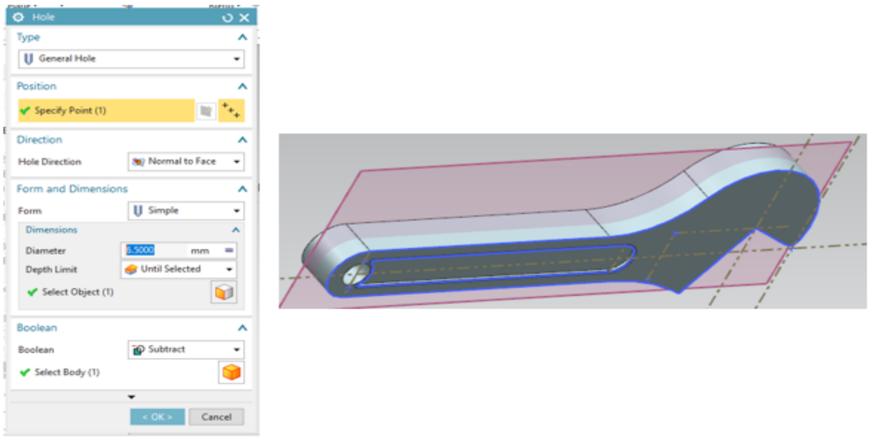


Figure 7: Method followed in Extrude process

2.1.2 Part 2 - Moving jaw

Moving jaw is the part which moves when the screw(shown in section 2.1.3) For making other parts the expression and parameters are linked with the main part by using link parameters from other parts option in NX. This is shown in figure(9). The part moving jaw is shown in figure(8). Here also all the dimensions are referred with parameters.

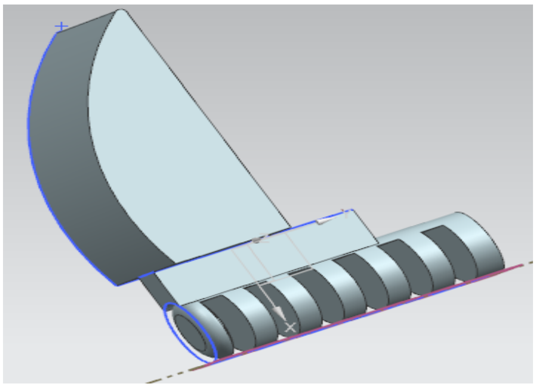
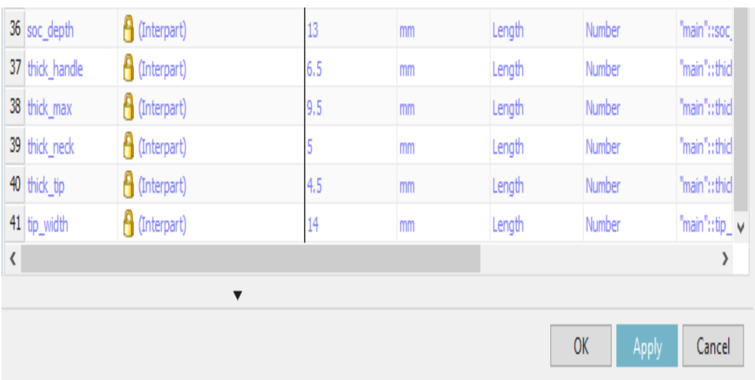


Figure 8: Moving jaw - Part 2



36	soc_depth	(Interpart)	13	mm	Length	Number	"main"::soc_
37	thick_handle	(Interpart)	6.5	mm	Length	Number	"main"::thic
38	thick_max	(Interpart)	9.5	mm	Length	Number	"main"::thic
39	thick_neck	(Interpart)	5	mm	Length	Number	"main"::thic
40	thick_tip	(Interpart)	4.5	mm	Length	Number	"main"::thic
41	tip_width	(Interpart)	14	mm	Length	Number	"main"::tip_

Figure 9: Inter part expressions to link expression between parts

2.1.3 Part 3 - Screw

Screw is the part which moves the moving jaw. This part is shown in figure(10). Screw pitch is fixed as 2mm but the length and dia is made as a parameter which varies as per the model. Internal diameter is also fixed at 4.05mm for the pin insertion.

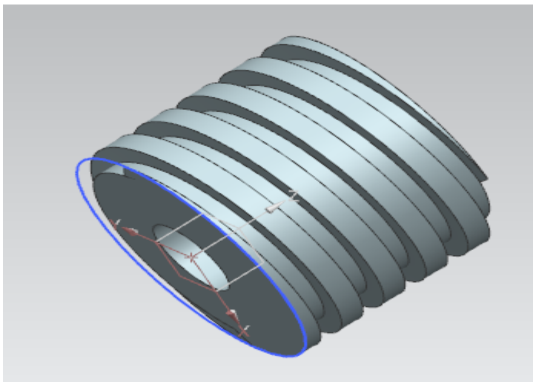


Figure 10: Screw - part3

2.1.4 Part 4 - Pin

Pin is the part which holds the helical screw with the main body part. The part is shown in figure(11). The diameter of the pin is fixed to be 4 mm. While the length is made as a parameter referred from the main body part.

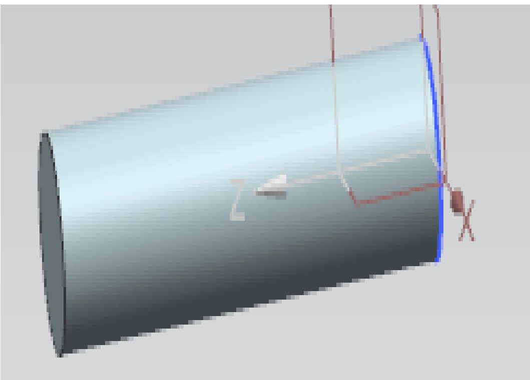
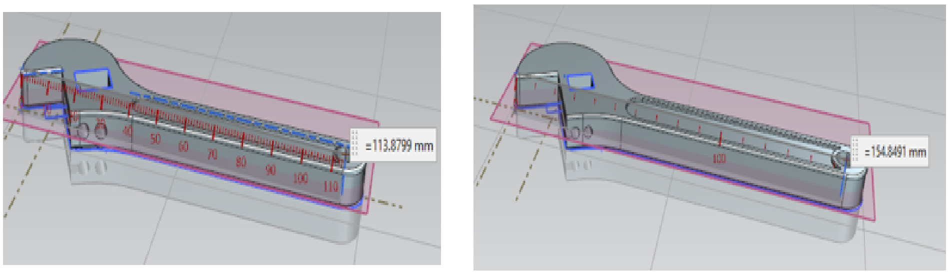


Figure 11: Pin - Part 4

2.1.5 Testing of parametric modelling

To check whether the parametric modeling is working the variable *model_no* is changed and then looked if the model is generated without any errors or warnings. It is seen that for the given types of variants it works fine. The picture(12) shows two different variant(model_no 1&2) of part1 made by parametric model.



Model1 whose length is ~114

Model2 whose length is ~155

Figure 12: Part1 variant 1 and 2 made by parametric modeling

2.2 Assembly of parts

For assembly of parts first the base part is loaded in the assembly environment and then other parts are imported into the environment and constraints are applied.

- First main body of the part is loaded in the assembly environment.
- Moving jaw is loaded and aligned to the main body by applying constraints *touch align* option on the bottom face of the moving jaw and the top surface of the main body(as shown in figure(13)) and then using *align and lock* option the axis of the screw in the moving jaw is made concentric with the hole diameter in the main body.
- The pin is aligned such that the hole and the pin diameter are concentric and the end face of the pin touches the main body end face.
- The screw is placed such that the internal diameter is concentric with the hole present in the mainbody and then the face of the screw thread is mating with the thread present in the moving jaw.

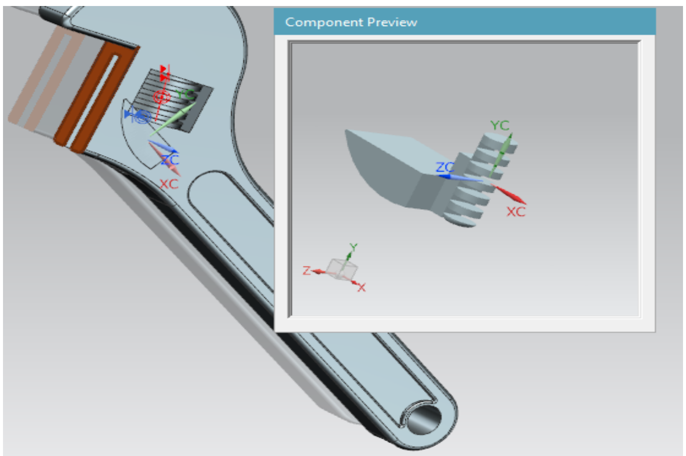


Figure 13: Assembly of moving jaw with the main body

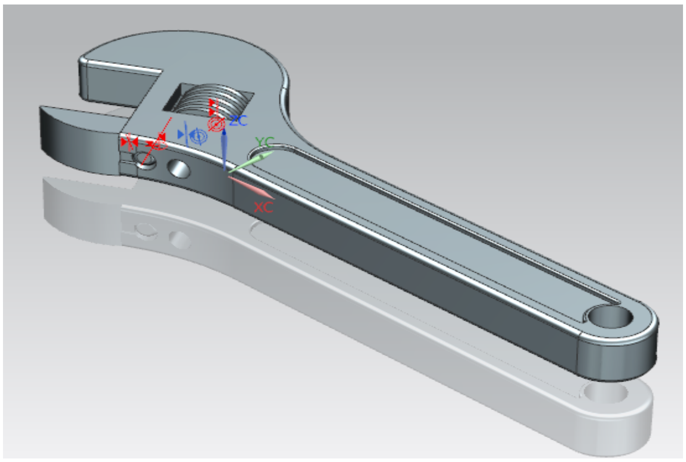


Figure 14: Adjustable Wrench after completion of assembly

3 Manufacturing of part

This section talks about how to select the primary manufacturing process for one of the part of the component (the main body). To decide on the manufacturing process of the product/component some of the assumptions need to be made. The number of parts that is required, the tolerance of each part dimensions and surface finish requirements. It is assumed that the company needs to make 2000 pieces of this part and since the tool is used on bare hands most of the time the surface finish of the tool has to be atleast 1.6 μm. The pin which fixes the screw and the hole in the main body is in interference fit. And also it need to be considered since the part is a tool which is used for fastening purpose it should not wear out easily so part need to be hardened or hard materials should be used.The material used for making wrench is mostly steel (chrome-vanadium steel).

3.1 Selection of Primary production process

Since the part has lateral features (hole for insertion of pin and moving jaw slot) which are also hollow the part is considered as **very - complex** part.
From the table (see table shown in figure(15)) for selecting primary production process based on shape complexity, the process for very complex part can be solid deformation and the second priority of the process is joining of pieces.

Table 5.1 Selecting basic forming techniques as a function of shape complexity and quantity

Shape complexity	Mono		Open		Complex		Very complex		Preference
Quantity	< 180	> 1000	< 150	> 2000	< 50	> 1500	< 100	> 1000	
	D	B	C	B	C	A	E	B	high
	E	E	D	A	D	B	D	D	
	B	D	B	D	B	C	C	E	
	C	C	E	C	E	D	A	C	
	A	A	F	E	F	E	B	A	
	-	-	A	-	A	-	F	F	low

Code A = forming from liquid (casting, moulding)
Code B = forming from solid by deformation
Code C = forming from solid by material removal
Code D = forming by joining parts
Code E = forming by assembly
2-14 Code F = forming by material increase

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Figure 15: Primary Process selection based on shape complexity

In solid deformation process there is two possible methods Bulk processing or sheet metal forming[2]. Out of these possible methods technologically possible methods is forging. Since the part need to be have more strength they are made by forging. First the part shape is cut from raw material and then it can be forged. Different types of forging include Drawn out, upset forging or compression die forging. Here **closed die forging** operation is selected followed by trimming process.
One of the critical dimension in this part is the hole for insertion of the pin this is a interference fit which may be H7/s6 [3] for a pin of 4 mm diameter the tolerance 4 s6 corresponds to+19 to +27 and hole of 4 H7 corresponds to 0 to +12 micron but the tolerance achieved by impression die forging is given in the table shown in figure(16) it is clearly seen the required tolerances is more than the tolerance achieved by the current selected process. So this requires some subsequent operations to be done on the forged component. Drilling is done for this process on a CNC machine (*refer fig(17)*). And slot milling is required.

Dimensional tolerances for impression-die forgings[14]

Mass [kg (lb)]	Minus tolerance [mm (in)]	Plus tolerance [mm (in)]
0.45 (1)	0.15 (0.006)	0.46 (0.018)
0.91 (2)	0.20 (0.008)	0.61 (0.024)
2.27 (5)	0.25 (0.010)	0.76 (0.030)
4.54 (10)	0.28 (0.011)	0.84 (0.033)
9.07 (20)	0.33 (0.013)	0.99 (0.039)
22.68 (50)	0.48 (0.019)	1.45 (0.057)
45.36 (100)	0.74 (0.029)	2.21 (0.087)

Figure 16: Tolerance Achievable by Closed die Forging process[4]

Minimum Recommended Position Tolerances for Location of Hole Features in (mm) Less than 1" (25.4mm) in Diameter *		
Method	Normal Tolerance inches (mm)	Tight Tolerance inches (mm)
Manual location techniques (center punch and drill)	.080 (2)	.020 (0.5)
Drill fixture using bushing	.025 (0.635)	.008 (0.2)
Precision milling or CNC machine with fixture	.016 (0.41)	.008 (0.2)
Precision milling or CNC machine with optical or precision orientation	.005 (0.13)	.003 (0.076)
Jig boring with optical or precision orientation	.002 (0.025)	.0005 (0.013)

Figure 17: Tolerance Achievable by Drilling process[5]

4 File Exchange Tests

This section talks about various file format conversion for exchange of file for closed loop for exchange within same CAD systems and between two different CAD systems.

4.1 Close loop test - within same CAD system

Closed loop test is done to test for the data processing losses when transfering form a CAD system to use in some CAD system in other place. The part file from NX *.prt* format is converted into IGES and STEP format and then again imported in NX. Following things are noticed

- In IGES format all the features are converted into a surface features and in STEP format the part is imported as one solid part there are no features present (this is shown in figure(20&21))
- The quality of the curves are poor in IGES format file *see picture(18)*.
- The imported part doesn't have any expression this means Parametric modeling feature is lost.
- All rendering details are deleted.

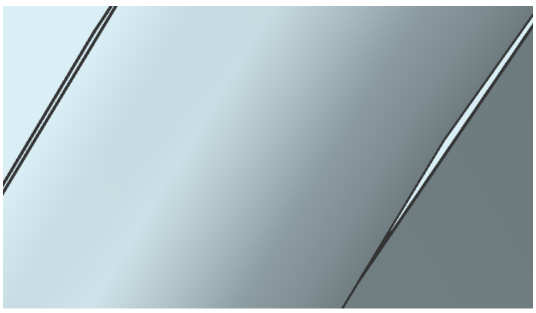

Figure 18: Zoomed view of edge in IGES file

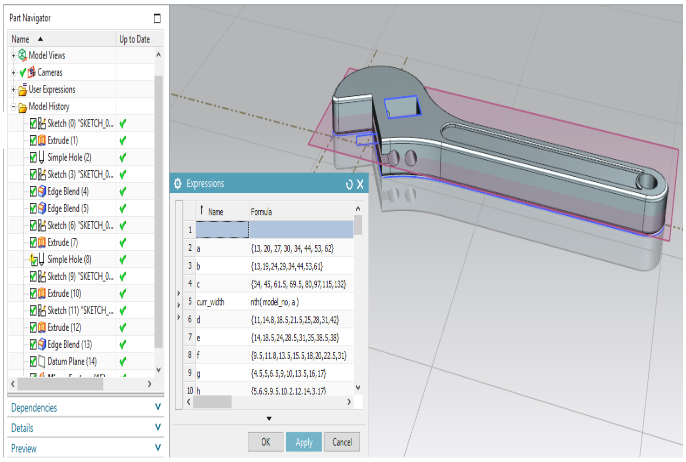

Figure 19: Part before conversion

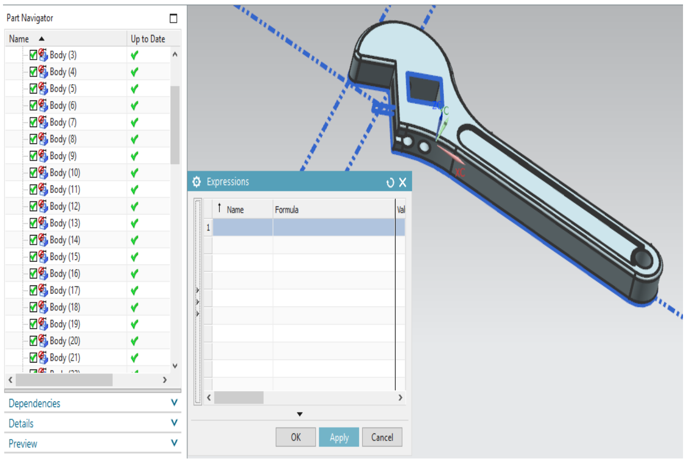

Figure 20: Part after IGES conversion

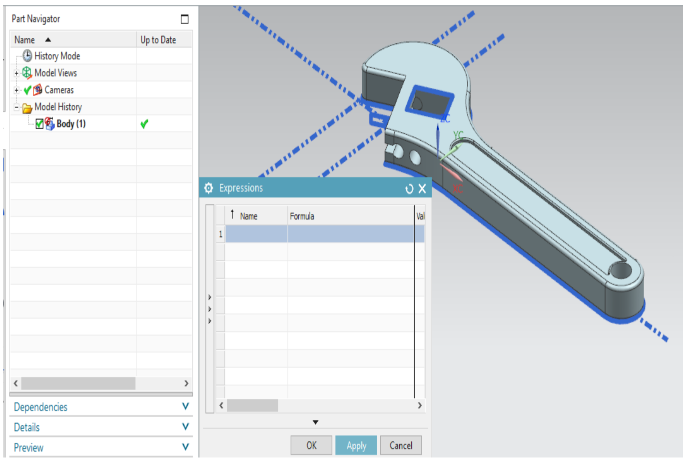

Figure 21: Part after STEP conversion

4.2 Testing - in other CAD system

In this test part file from NX is converted into a standard (IGES and STEP) and opened in solid edge and following things are noticed

- All the expressions are not available so no parametric modeling.
- The model is imported as 2d surfaces and editing of these surfaces were not possible.
- All the constraints are not available anymore.

References

[1] *113a.c chromed adjustable wrenches*. [Online]. Available: https://www.traceparts.com/en/product/facom-chromed-adjustable-wrench-maximum-opening-of-13-mm?CatalogPath=TRACEPARTS:TP02006003003&Product=10-27022007-124785&SelectionPath=1%7C1%7C1%7C1%7C1%7C1%7C1%7C1%7C1%7C1%7C1%7C1%7C1%7C1%7C.

[2] *Manufacturing*. [Online]. Available: https://sites.esm.psu.edu/courses/emch13d/design/design-tech/manufacturing/manuf_2.html.

[3] *Tolerances and fits*. [Online]. Available: http://www.mitcalc.com/doc/tolerances/help/en/tolerancestxt.htm.

[4] *Forging*, Jun. 2018. [Online]. Available: https://en.wikipedia.org/wiki/Forging.

[5] L. E. Edge, *Machinist drilling mechanical tolerance capabilites chart - ansi size drilled hole tolerance, iso metric drill sizes*. [Online]. Available: https://www.engineersedge.com/manufacturing/drill-mechanical-tolerances.htm.