

# *Designing and Manufacturing of Tooth Wheel Used in Metro Trains*

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**Abstract:** Some machining operation is so simple which are done quite easily, such as turning, the job is held in position in the chuck and turning operation is done easily. No other device is required to hold the job or to guide the tool on the machine in such an operation. Be that as it may, a few operations are such sort in which the instrument is required to be guided by methods for another gadget and furthermore a few employments are of such structures which are required to be held in position on the machine by methods for another gadget. Installations are exceptional reason device which are utilized to encourage generation when work piece is required to create inside a set up resistance.

A Tooth Wheel which is looked like a similarity to the transmission gears which had a higher amount of advantages than the reliable gears used presently, This type of gears are used at higher proportions rate of transmission where required like electrical locomotives(metros),flight breakers system. The main purpose of using Tooth Wheel is used to have higher transmission rates.

The aim of the project is to design and Manufacture of tooth wheel, As the process the Tooth Wheel are manufacturing by using circular face mill cutter there may chance of using fixture for tooth wheel. The 3D model of Tooth Wheel is created in NX-CAD software and NC program is generated on 3D model of Tooth Wheel. Generated NC program is fed to CNC machine through DNC lines. Finally, Tooth Wheel is manufactured using NC program which is generated in NX-CAM software.

## **I. Introduction**

The process the Tooth Wheel is manufacturing by using circular face mill cutter there may chance of using fixture for tooth wheel. The 3D model of Tooth Wheel is created in NX-CAD software and NC program is generated on 3D model of Tooth Wheel. Generated NC program is fed to CNC machine through DNC lines. Finally, Tooth Wheel is manufactured using NC program which is generated in NX-CAM software so we initially will go through the CAD/CAM.

**Literature Review:** The wheel profile design procedure described in the previous time has been applied to improve the performance of some RET metro trains. In 1999 the existing NP46 rails were replaced by S49 rails. The profiles looked similar and therefore no difference in the train performance was expected.

However, after the introduction of the new rail profile the metro trains started to experience severe lateral vibrations. At the same time a high level of wheel wear was observed after a relatively small mileage. In fact these vibrations were caused by the worn wheels.

Immediately after these vibrations had been observed the wheels were re-profiled to prevent derailment. Thus, due to the replacement of the rails the lifetime of the wheels was reduced from 120000 km (with the rails NP46) to 25000 km (with the rails S49). The Railway Engineering Group of TU Delft was commissioned a research work to improve vehicle stability and wheel life.

## **II. Unigraphical Description Of Wheel Profile**

It will provide the basics required to use any CAD/CAM package. You will learn the preliminary steps to start, to understand and to use the package for modeling, drafting, etc. To start NX, use the NX 7.5 shortcut under Start (or Windows button) -> (All) Programs->UGS NX 7.5. It may take a minute or so for NX to start the first time.

Once opened, you will be presented with the window shown in below Figure. / STARTING A NEW MODEL to begin displaying, you should first make a section document. NX part records utilize the expansion.prt for the two segments and congregations of segments. In this instructional exercise, a solitary part record will be utilized with one segment. To make another part, tap the new catch to open the new discourse. Until further notice, remain in the Model tab. Select Model from the Templates list (the default) and set the Folder to an area on your S: drive or desktop. The default area might be put away locally on the lab Computer you are utilizing and won't not be in your meandering profile (it won't not be available on an alternate Computer). Once an organizer has been chosen, set the name of the part document. Snap OK to begin displaying.

Once the new document has been made, the NX modeling interface will open (Figure 3). Like most current PLM tools, the interface for NX contains various symbols, records, content prompts and different elements that can be unfathomably overpowering. Until further notice, we will focus on the sketching tools, part navigator, viewer and menu. / DRAWING IN NX NX, as most current PLM apparatuses, is highlight based. That implies you develop a part from an arrangement of elements that are included grouping. This arrangement, and the subtle elements of each element, can be altered later if the design

should be changed. Elements can be evacuated, embedded or changed to make any strong part possible.

This permits an exact and profoundly editable strategy for portraying strong segments and contrasts from the 3D freestyle displaying approach (freestyle demonstrating is utilized as a part of Computer graphics applications, for example, making models for Computer games). The elements you add to a section will show up in the Part Navigator under Model History.

For another part, the main thing showing up in the model history ought to be a Datum Coordinate System. This is the 3D Cartesian facilitate framework utilized as a part of the part and incorporates an inception area and arrange tomahawks. These are appeared with the facilitate framework pointer at the focal point of the Viewer.

The general work process for making a section in NX is to make a draw, expel or spin the outline and afterward add components to the subsequent strong part. Representations are 2D (mostly) drawings that are utilized to characterize a cross area of the part. This cross segment can be expelled (pushed into or hauled out of the plane of the outline) and pivoted (wrapped around a hub) to frame a 3D strong part.

To make a Sketch, tap the Sketch catch in the drawing toolbar. This will open another exchange for characterizing the outline. The initial phase in characterizing the Sketch is to set the portraying plane. This is the 2D plane that the portray will be drawn onto. This plane can be gotten from the current Datum Coordinate System or can be founded on geometry from a current part (you can outline on the surface of a section).

Until further notice, begin the portray on the XY plane. To choose the XY plane, move the cursor to the skewed lines interfacing the x and y pivot lines on the arrange framework marker. // NX, also known as NX Uni graphics or usually just U-G, is an advanced CAD/CAM/CAE software package developed by Siemens PLM Software. It is used, among other tasks, for: Design (parametric and direct solid/surface modeling) Engineering analysis (static, dynamic, electro-magnetic, thermal, using the Finite Element Method, and fluid using the finite volume method). Manufacturing finished design by using included machining modules First release of the new "Next Generation" version of Uni graphics and Ideas, called NX.

This will eventually bring the functionality and capabilities of both Unigraphics and I-DEAS together into a single consolidated product. Expanding many-sided quality of products, advancement procedures and design groups is testing organizations to discover new instruments and strategies to convey more noteworthy development and higher quality at bring down cost.

Driving edge innovation from Siemens PLM software conveys more prominent power for the present design challenge. From creative Synchronous Technology that joins parametric and sans history demonstrating, to NX Active Mockup for multi-CAD get together design, NX conveys leap forward innovation that sets new models for speed, execution, and convenience. NX automates and disentangles configuration by utilizing the item and process learning that organizations pick up as a matter of fact and from industry best practices. It incorporates tools that

originators can use to catch information to robotized dreary errands. The outcome is diminished cost and process duration and enhanced quality.

In product design tolerances are specified due to inherent variability's of manufacturing processes. Tolerance specifications have significant implications on the quality and cost of the product for proper tolerance specifications, tolerance analysis must be performed. Prototyping is the only method available for the analysis of the geometric variations.

**Tolerance:** Allowance for a specific variation in the size and geometry of part, why is it needed: No one or thing is perfect. Hence, engineers have come up with a way to make things close to perfect by specifying Tolerances. – Since variety from the drawing is unavoidable the satisfactory level of variety must be indicated.

**Interchangeability:** If a replacement part is used it must be a duplicate of the original part within certain limits of deviation .The relationship between functionality and size or shape of an object varies from part to part. How are Tolerances Specified Size– Limits specifying the allowed variation in each dimension (length, width, height, diameter, etc.) are given on the drawing Geometry– Geometric Tolerance Allows for specification of tolerance for the geometry of a part separate from its size GD&T (Geometric Dimensioning and Tolerance) use special symbols to control different geometric features of a part Department of Mechanical Engineering and Mechanics Value of Tolerance. The tolerance for a single dimension may be specified with the dimension and then the tolerance. The tolerance is total variation between the upper and lower limits. The tolerance for a single dimension may be specified with the dimension and then the tolerance. The tolerance is total variation between the upper and lower limits.

### III. Identification of CNC Machine for Wheel Design

DMG 5-axis milling machine is used for manufacturing fin cage component. In DMG 5-axis milling machine X, Y, Z, B, C are 5 vectors, X & Y are tool movement and Z is for table upwards movement, B for spindle movement, C for table rotation. High rigidity with Integrated Spindle up to 12000rpm, Spindle is directly coupled with motor.

Vertical Operations, Integrated rotary table of 1200mm X 700mm with rotary dia 700mm. Horizontal Operations, With head tilting at 90deg. Angular and 5-axes simultaneous machining, Capable of machining from +30 deg to -120deg head tilting. Machine accuracies, Positional Accuracy +/- 0.005mm, Repeatability +/- 0.003mm FOR Toothed wheel 5-axis milling machine is used / SELECTION OF TOOLS Selection of tools plays an important role in manufacturing any component.

Proper tools must be selected otherwise in manufacturing process improper tools results in damage of work piece or damage to the tools, tool holders. Suitable tools for manufacturing fin cage are listed below /SPOT\_DRILLING This operation subtype allows th1e tool to pause at the tool tip or

shoulder depth of the tool by a specified number of seconds or revolutions.

**Drilling:** This operation subtype allows you to do basic point-to-point drilling /TAPPING this operation subtype creates a tap cycle that feeds into the hole, reverses the spindle, and feeds out of the hole /FACE\_MILLING FACE\_MILLING is the main Face Milling operation subtype. A milling cutter that cuts metal with its face. Face milling creates large flat surfaces.

**Face Milling:** AREA Face Milling Area is a Face Milling operation subtype that is customized to recognize a cut area and wall selection.

**End Mill:** A milling cutter that performs a mix of peripheral and face milling. End milling engages the bottom and edges of the milling cutter.

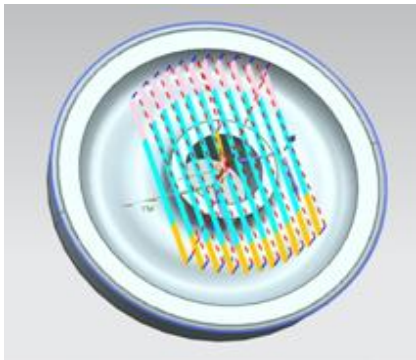


Fig 1: Tool path generation of face milling for toothed wheel

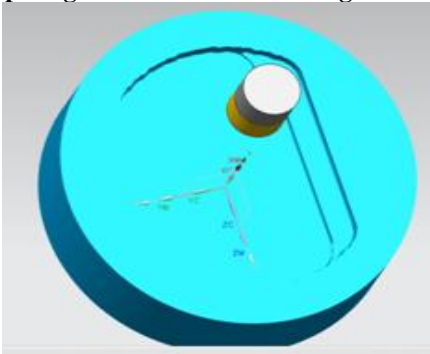


Fig 2: tool path verification of face milling for toothed wheel

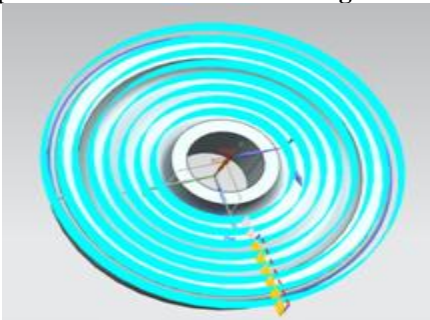


Fig 3: tool path generation of planar milling for toothed wheel

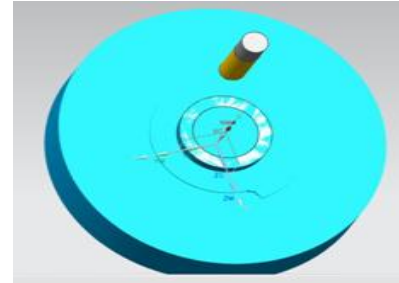


Fig 4: tool path verification of planar milling for toothed wheel

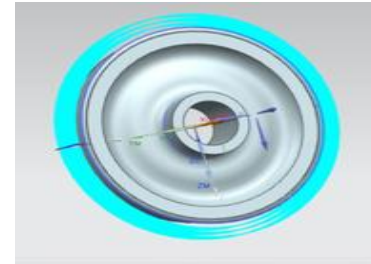


Fig 5: tool path generation of planar milling for toothed wheel

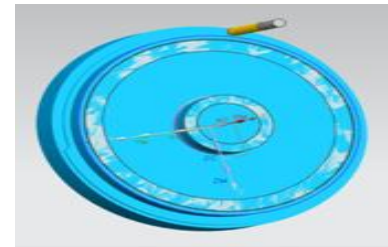


Fig 6: tool path verification of planar milling for toothed wheel

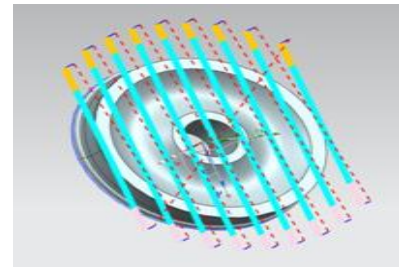


Fig 7: tool path generation of planar milling for toothed wheel

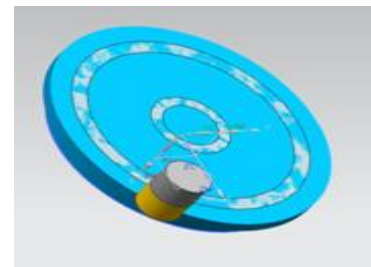
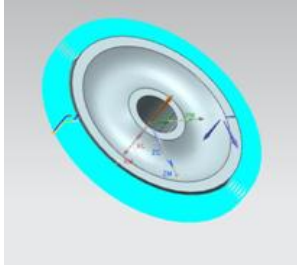
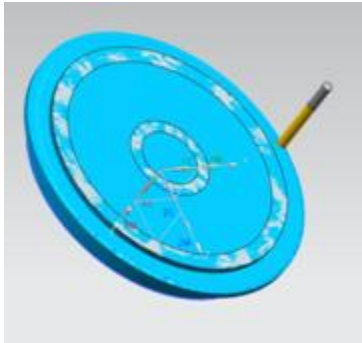


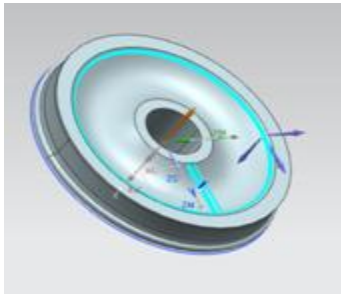
Fig 8: tool path verification of planar milling for toothed wheel



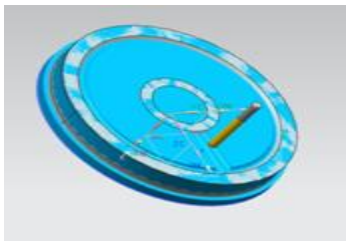
**Fig 9: tool path generation of planar milling for toothed wheel**



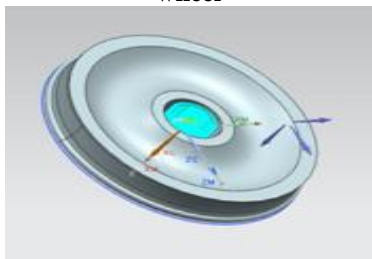
**Fig 10: tool path verification of planar milling for toothed wheel**



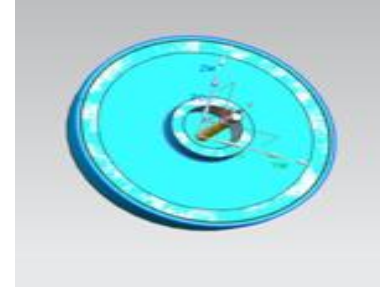
**Fig 11: tool path generation of planar milling for toothed wheel**



**Fig 12: tool path verification of planar milling for toothed wheel**



**Fig 13: tool path generation of planar milling for toothed wheel**



**Fig 14: tool path verification of planar milling for toothed wheel**

#### **IV. Conclusion**

3D model of Toothed wheels created using UGNX-7.5 – CAD. Tool path is generated for “Toothed wheel” component using CAM software (‘UGNX-7.5’ which is CAD/CAM software used to generate part program by designing and feeding the geometry of the component). The generated part program will be transferred to the required CNC machine with the help of DNC lines. Final component of toothed wheel is manufactured by above generated NC program.

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