

MINISTRY OF EDUCATION AND SCIENCE OF THE REPUBLIC
KAZAKHSTAN

Kazakh National Research Technical University named
after K. I. Satpayev

A. Burkitbaev Institute of Industrial Automation and Digitalization

Department "Industrial Engineering"

Arshynbek Marzhan Myrzalykyzy

On the topic: "Computer-integrated preparation of the manufacturing technology
of the "pulley" part in the CAD/CAM environment. Annual production program
3000 pieces"

DIPLOMA WORK

Specialty: 5B071200 - Mechanical Engineering

Almaty 2020

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ADMITTED TO THE DEFENSE

Head of the Department

PhD, associate Professor

_____ Arymbekov B. S.

« ____ » _____ 2020 y.

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Performed

Arshynbek M. M.

Reviewer

Scientific Adviser

PhD, tutor

Zhankeldi A. Zh.

« ____ » _____ 2020 y.

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CONFIRM

Head of the Department

PhD, associate Professor

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« _____ » _____ 2020 y.

TASK

to complete a thesis

Studying: Arshynbek Marzhan Myrzalykyzy

Topic: *Computer-integrated preparation of the manufacturing technology of the "pulley" part in the CAD/CAM environment. Annual production program 3000 pieces.*

Approved by the order of the University Rector № 762-b "27" January 2020 y.

Deadline for completion of work

"20" April 2020 y.

Initial data for the thesis: characteristics of CAD/CAM system, definition of the type of production, the term of computer-integrated technology.

Summary of the thesis:

a) analysis of pulley technology

б) the concept of computer-integrated mechanical engineering

в) description of the part

г) creating a model

List of graphic materials: detail drawing, tables of mechanical and physical properties, part route, attachments of creating process of material.

presented__ slides presentation of the work

Recommended main literature: *from*

a) Дипломное проектирование по технологии машиностроения : [Учеб. пособие для вузов / В. В. Бабук, П. А. Горезко, К. П. Забродин и др.]

CHART

preparation of a thesis

Name of sections, list of issues being developed	Deadlines for submission to the scientific supervisor	Note
Literature review Theoretical Part	04.02.20 – 20.03.20	
Examination of computer-integrated mechanical engineering	24.03.20 – 7.04.20	
Analysis of pulley technology. Creating a Model.	10.04.20 – 24.04.20	

Signature

consultants and standard controller for the completed thesis with an indication of the relevant sections of the work

The section titles	Consultants, full name. (academic degree, title)	Date of signing	Signature
Normcontrol	Phd, tutor, Zhankeldi A. Zh.		

Scientific adviser _____ Zhankeldi A. Zh.

The task was accepted by the student _____ Arshynbek M. M..

Date "___" _____ 20__ y.

АННОТАЦИЯ

В данной дипломной работе предоставлены решения вопросов проектирования «шкива» с эксплуатацией системы CAD/CAE.

Главной задачей дипломной работы является исследование систем автоматизированного проектирования шкивов с поэтапным его использованием в производстве, что предоставляет возможность упростить их проектирование, получить экономическую эффективность благодаря сокращению количества используемого материала без утраты долговечности и надежности всех частей шкива.

На стадии выполнения работы были рассмотрены общие сведения о шкиве, и его служебное назначение. Были произведены расчеты для определения типа производства. Также исследованы анализы технологичности конструкции детали шкива и технологии изготовления шкива.

Соответственно были анализированы основные характеристики и методики функционирования CAD/CAM систем. С помощью программного обеспечения КОМПАС 3D благодаря чертежам была произведена модель детали «шкив». Также интенсивно изучены инструментальные возможности детали в КОМПАС 3D.

ANNOTATION

In this diploma work are provided solutions projecting of «pulley» with CAD/CAE system operation.

The main task of the diploma work is to study systems for the automated design of pulleys with their phased use in production, which makes it possible to simplify their design, to achieve economic efficiency by reducing the amount of material used without losing the durability and reliability of all parts of the pulley.

At the stage of execution of the work, general information about the pulley and its assignment were considered. Calculations have been made to determine the type of production. Also analyzed the efficiency of the pulley part design and pulley manufacturing technology.

Accordingly, the main characteristics and methods of operation of CAD/CAM systems were analyzed. With the help of software COMPASS 3D the model of the part «pulley» was produced thanks to drawings. The instrumentation of the 3D COMPASS part was also intensively studied.

АҢДАТПА

Бұл дипломдық жұмыста CAD / CAE жүйесінің көмегімен «шкив» дизайнын шешуге мүмкіндігі қарастырылды.

Диссертациялық жұмыстың негізгі мақсаты - өндірісте кезең-кезеңімен қолдану арқылы дамытқыштардың автоматтандырылған дизайның зерттеу, олардың конструкциясын жеңілдетуге, оның барлық бөліктерінің беріктігі мен сенімділігін жоғалтпастан, пайдаланылатын материал мөлшерін азайту арқылы экономикалық тиімділікке қол жеткізуге мүмкіндік береді.

Жұмыс барысында шкив туралы жалпы мәлімет және ресми бұйымның қолданылуы қарастырылды. Өндіріс түрін анықтау үшін есептеулер жүргізілді. Сондай-ақ, доңғалақ бөлшегінің конструкциясының жарамдылығына және доңғалақ жасау технологиясына талдау жасалды.

Сәйкесінше, CAD / CAM жүйелерінің негізгі сипаттамалары мен жұмыс істеу әдістері талданды. КОМПАС 3D бағдарламалық жасақтамасын пайдаланып, сызбалардың арқасында «шкив» бөлік моделі құрастырылды. Сондай-ақ, КОМПАС 3D бөлігіндегі құралдардың мүмкіндіктері қарқынды зерттелді.

CONTENT

Introduction	9
1. Technological part	11
1.1 Definition of the type of production	12
1.2 Analysis of the manufacturability of the part design	13
1.3 Analysis of pulley technology	14
2. Research part	20
2.1 The concept of computer-integrated mechanical engineering	20
2.2 COMPASS-3D	21
2.3 Description of the Part	24
3. Creating a Model	27
Conclusion	31

Introduction

Mechanical Engineering in Kazakhstan is currently in a difficult situation. The reality of even the largest world-famous enterprises is unclear. The usual explanation for this situation is poor industrial performance, which is an inevitable consequence of the low competitiveness of domestic products compared to foreign products.

What is the success of Western mechanical engineering? Research shows that the main reason for the high efficiency of Western industry is the availability of modern information technologies for all production. Of course, one of the main tasks of the engineer is the design of the products, as well as the technological processes of their manufacture. Reflecting the established practice of the consistent implementation of the processes of design and development of manufacturing technology, the Computer Automated Process Planning (CAPP) in mechanical engineering is divided into two main parts: CAPP design of products and CAPP technology of their manufacture.

CAPP Design of products, which is called CAD (Computer Aided Design), performs volumetric and flat geometric modeling, specifying the types and sizes of the parts based on the creation of parametric series, Engineering analysis, evaluation of design solutions of the received drawings.

Configured process description is used in the Computer Aided Manufacturing (CAM) system. The mission of the CAM is to organize the operation of, for example, a flexible production system by developing the personnel of the CNC equipment control programme.

The most striking feature of CAD/CAM systems is the so-called solid modeling of the product exclusively on the computer screen, viewing the process of processing the parts and transferring the generated control programs over the computer networks to the equipment with CNC. Traditional drawing and paper documentation

become a by-product of design, production times are shortened, quality and competitiveness of products are improved.

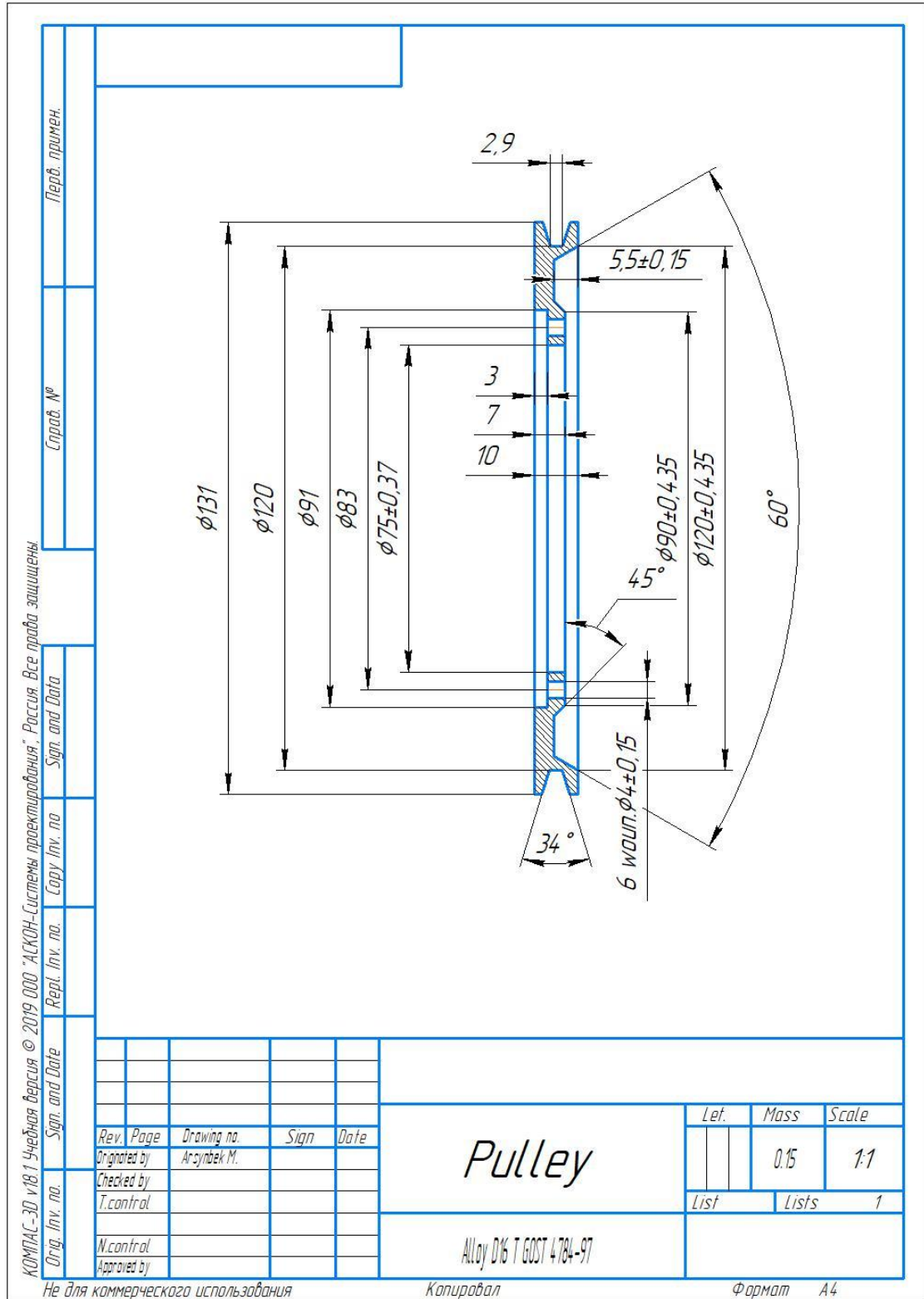
The 3D COMPASS is a versatile 3D design system that is easy to use and provides solid, surface and direct modeling that combines CAD and CAM modules. It provides a fairly complete set of tools for design, engineering analysis, drawing and development of management programs for various CNC machine tools.

The aim of this project is to make practical use of the knowledge acquired in the field of mechanical engineering for technological preparation of production on the basis of MCAD-systems.

In this work the application of COMPASS 3D for computer-integrated solution of project and technological tasks is considered:

- drawing and 3D model of the part «pulley»;
- verification of the routing processes of manufacture of the «pulley» part;
- creation of an efficient manufacturing process for a component
- development of an operating sketch of the technological process.

1. Technological part.



Detail drawing

1.1. Definition of the type of production

Annual product program $N = 3000$ PCs.

The actual annual Fund of equipment operating time is determined from the recommendations in table 1: $F_D = 4015 \frac{h}{cm}$

Defining the release time of parts:

$$tr = \frac{F_d \cdot 60}{N} = \frac{4015 \cdot 60}{3000} = 80.3 \left(\frac{min}{pieces} \right),$$

Where F_d - is the actual Fund of equipment operating time.

Data on an existing (similar) factory technological process or on an enlarged rationing of operations in table 1

Table 1. Duration of operations of an existing factory process

№	The name of the operation	T_{pieces} , min
1,1	Revolving lathe	0,978
1,2	Revolving lathe	5,028
1,3	Revolving lathe	3,168
2,1	Vertical drilling	1,083
3,1	Chemical-thermal	1,200

Number of operations $n = 5$.

Total unit time for all operations:

$$\sum T_{pieces} = 11,457 \text{ min.}$$

The average unit time is determined by the formula:

$$T_{av.unit} = \sum \frac{T_{pieces}}{n} = \frac{11,457}{5} = 2,2914 \text{ min}$$

The coefficient of seriality is determined by the formula:

$$k_s = \frac{tr}{T_{av.unit}} = \frac{30,1}{0,988} = 13,136$$

Since $10 \leq k_s \leq 20$, then the type of production is medium-scale production.

1.2. Analysis of the manufacturability of the part design

A **pulley** is a friction wheel with a rim or a small groove around the circumference. Its purpose is to transfer motion to a drive belt or rope. They are mainly used in hand-held transmissions.

D16t - is one of the most sought-after duraluminium alloys in the shipbuilding, aviation and space industries. Its main advantage is that the metal rolling produced from it has:

- stable structure;
- High strength characteristics;
- 3 times lighter weight than steel products;
- increased resistance to microscopic deformation during operation;
- good mechanical handling on lathe and milling machines, second only to some other aluminium alloys.

Chemical composition of D16T

Fe	Si	Mn	Cr	Ti	Al	Cu	Mg	Zn	Other	
Up to 0,5	Up to 0,5	0,3-0,9	Up to 0,1	Up to 0,15	90,9-94,7	3,8-4,9	1,2-1,8	Up to 0,25	Each 0,05	Sum 0,1

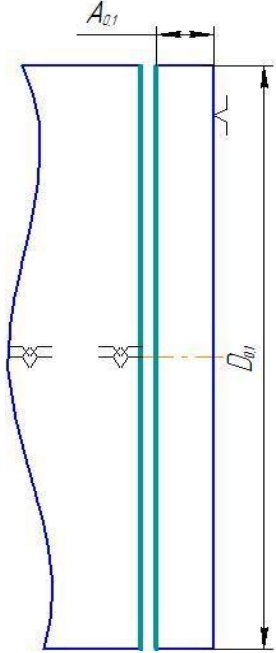
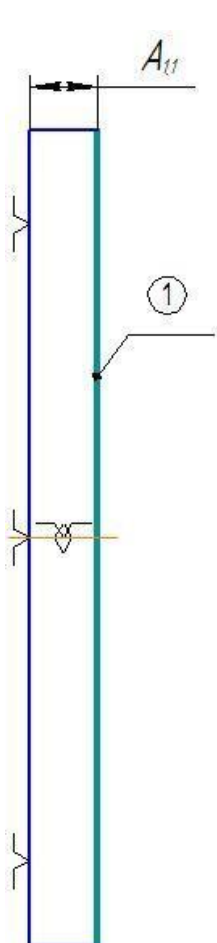
Physical and mechanical properties

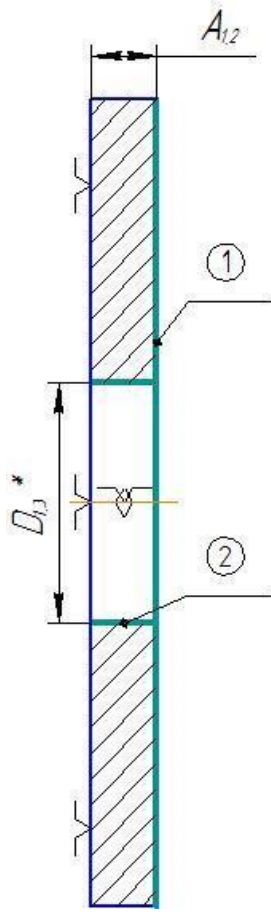
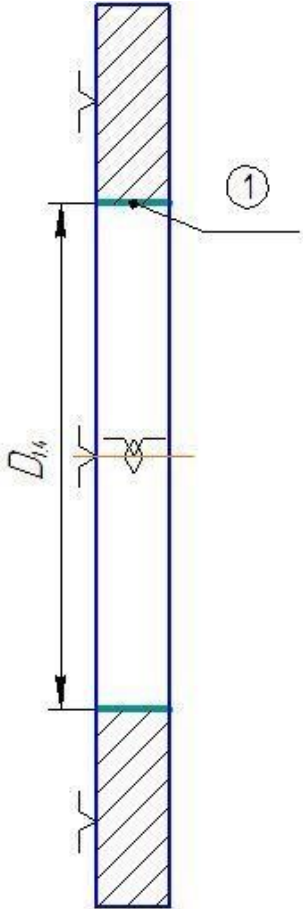
σ_B , MPa	HB	δ %	σ_{-1} , MPa
440	130	20	120

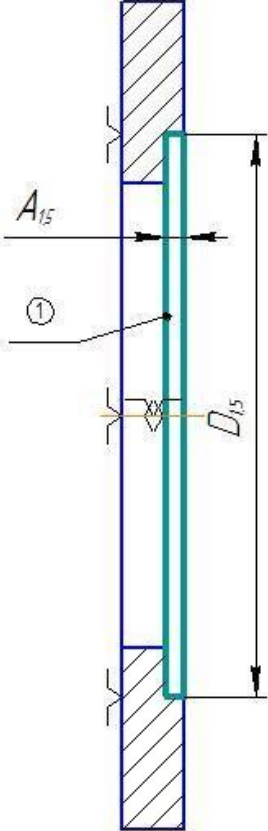
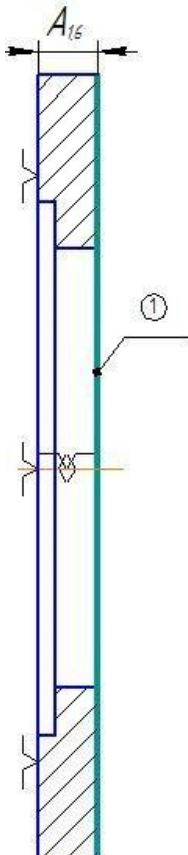
1.3. Analysis of pulley technology

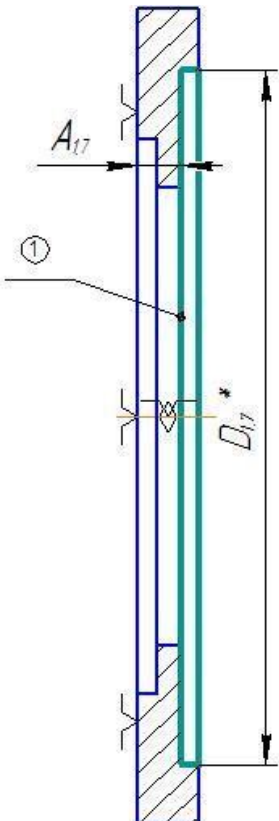
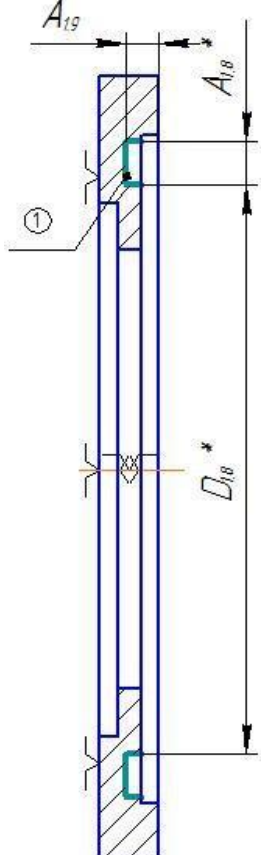
The route of the pulley technology is presented in table 1, where the technological bases are also indicated.

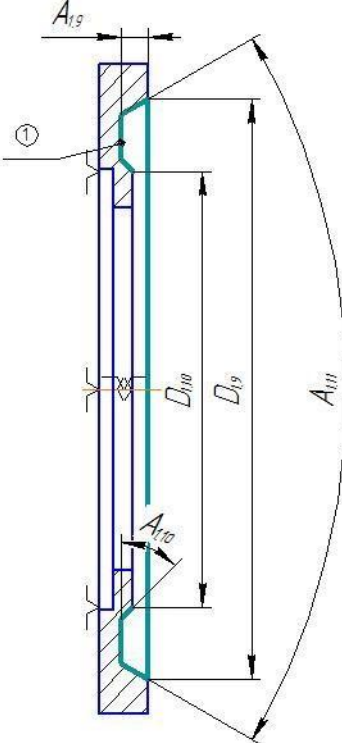
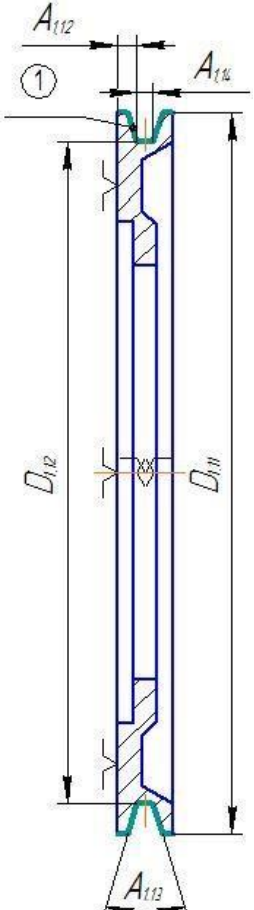
Table 2. Part Route

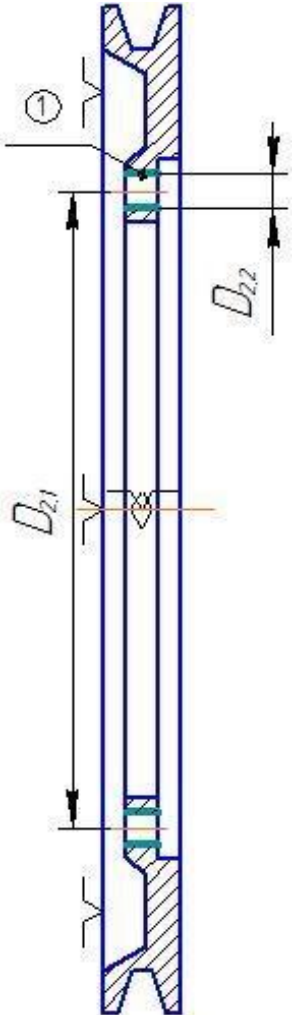
<p>0 Tape saw</p>	<p>1</p>			<p>Cut the finished product to withstand the size A01 ,D01</p>
<p>1 Power station with CNC</p>	<p>1</p>		<p>Outgoing Incisor</p>	<p>Install and brace Sharpen surface 1 to withstand size A1.1</p>

2		<p>Outgoing incisor</p> <p>Spiral Drill</p>	<p>Install and brace</p> <p>Sharpen the surface 1, withstanding size A1,2</p> <p>Drill hole 2, resistant to size D1,3</p>
		<p>Elongated bent incisor</p>	<p>Sharpen hole 1 to withstand size D1.4</p>

			<p>Persistent straight incisor</p>	<p>Sharpen the groove 1 to withstand the dimensions A1,5, D1,5</p>
	<p>3</p>		<p>Forward bent incisor</p>	<p>Install and brace</p> <p>Sharpen the surface 1, withstanding size A1,6</p>

			<p>Persistent straight incisor</p>	<p>Sharpen end groove 1, resistant to dimensions A1,7, D1,7</p>
			<p>End groove incisor</p>	<p>Sharpen end groove 1 to withstand dimensions A1,8, A1,9, D1,8</p>

			<p>Persistent straight incisor</p>	<p>Sharpen end groove 1 to withstand dimensions A1,9, A1,10, D1,9, D1,10, A1,11</p>
			<p>Outgoing incisor Ditch cutter</p>	<p>Sharpen D1.11 diameter Sharpen groove 1 to withstand dimensions A1,12, A1,13, A1,14, D1,12</p>

<p>2 Drill</p>	<p>1</p>	 <p>The diagram shows a cross-section of a spiral drill bit. It features two diameters: D_{21} for the main body and D_{22} for the cutting edge. A hole, labeled '1', is shown being drilled into a workpiece. The workpiece has a hatched outer surface and a central hole. The drill bit is positioned to drill this hole, with its cutting edge at the bottom. The hole is shaded in light blue.</p>	<p>Spiral Drill</p>	<p>Install and brace</p> <p>Drill hole 1, resilient D2,1, D2,2</p>
<p>3 Chemical-thermal</p>				<p>Cover anodized oxide of plated and chromatinated coating</p>

2. Research part.

2.1. The concept of computer-integrated mechanical engineering

Currently, the main trends in modern mechanical engineering are increasing the working parameters of machines and constructions, reducing their material expenses and energy intensity. In addition, the development time, quality, reliability and cost have a huge impact. In order to comply with today's rules and regulations, the design automation process must be seen as a system of interrelated design, computing and technology tools at all stages of the project.

It should be noted that the time of unrelated programs and systems that automate individual parts of the production chain, as it was at the dawn of the computer era, has passed. Now the user-professional requires the application software developers to complete solutions providing end-to-end technology within a single integrated system of automated design. This approach makes it possible to model the product on a computer and to produce ready-made optimal solutions by overhauling a large number of variants at the design stage and thus to reduce the production time of the finished product several times.

CAD of manufacturing technology, which in Kazakhstan is commonly referred to as the Automated System of Technological Preparation of Production (ASTPP), forms technological routes, chooses equipment, performs the development of technological processes and technological equipment. The narrower task of CAPP (Computer Automated Process Planning) is to design the technological documentation that is consistently brought to the workplaces.

The radical change in the processes of designing and manufacturing of products at machine-building enterprises of the West consists in the unification under the unified control of computer systems CAD, CAPP and CAM, which received the name «computer-integrated manufacture».

Software systems that support product design and manufacturing processes are commonly referred to as CAD/CAM systems. At the same time «CAM» indicates the presence of NC modules and, therefore, the receipt at the output of the control program system for various processing methods.

Integration of CAD and CAM modules in the same system allows specialists of different services at the enterprise to cooperate closely and efficiently among themselves. For example, design bureau engineers create component models, make the necessary strength calculations, then assemble the assembly units, check the possibility of assembly, and transmit model data to the technology department. Successively, engineers for ready-to-use models with the aid of CAM modules form control programs for CNC machine tools, check their accuracy and correctness, adjust and with the aid of a computer network, transmit to machines with CNC, where the resulting parts are processed.

The CAD/CAM system in the information system is supported by a single database (DB) in which the three-dimensional mathematical model of the product is stored, which means information about its structure and geometry (as a result of the design in the CAD system), as well as about the manufacturing technology (system of machine coordinates, type and parameters of cutting tool, trajectory of its movement during processing of different surfaces, cutting modes, etc.).

At present, the main trend in the development of Western enterprises is the transition from separate Saps and their partial integration to the full integration of the technical and organizational spheres of production.

Many domestic plants are just beginning to fully exploit CAD/CAM systems, facing financial difficulties and a shortage of specialists. To date, the most famous and common CAD/CAM systems include ADEM, T-FLEX, Compass, and Gemma. Imports include Solidworks, Cimatron, Inventor, Unigraphics, Pro/Engineer.

2.2. COMPASS-3D

The COMPASS-3D system is designed to create three-dimensional parametric models of parts and assembly units containing both typical and non-standard, unique structural elements.

The parametric technology makes it possible to quickly obtain model products on the basis of the designed sample.

A key feature of COMPASS-3D is the use of its own mathematical core and parametric technologies developed by ASKON.

The main tasks of the system are:

- product modelling to create the design and production documentation necessary for their production (assembly drawings, specifications, details, etc.),
- simulation of products to calculate their geometric and schematic characteristics,
- modeling products to transfer geometry to calculation documents, to control software packages for CNC equipment
- create isometric images of products (for example, for cataloging, illustrating technical documentation, etc.).

Details. A part model in COMPASS-3D is created by performing boolean operations on three-dimensional elements. Medium-sized elements are formed by moving a flat figure ("sketch") in space as specified by the user. The sketch is displayed on the plane by the standard COMPASS-GRAPH drawing and drawing editor. It is possible to transfer an image from an earlier graphical document, thus allowing the creation of a three-dimensional model to rely on existing drawing and design documents.

The system makes it possible to use rotational elements, pressing elements, cross-section elements and kinematic elements. There are also different construction options available for each of them.

Additional operations simplify the setting of parameters of common structural elements such as curves, round holes, stiffening ribs, casting slopes and curves. At any stage, which makes the work much easier, it is possible to form a thin-walled envelope, and also to remove a part of the body along a boundary that is a plane or a curved surface.

COMPASS-3D also offers a variety of ways of copying elements: copying over a grid, around a circle, along a curve, mirror copying, and creating "mirror" parts.

The sketch is displayed on the plane by the standard COMPASS-GRAPH drawing and drawing editor. All drawing and editing commands, parameterization commands and service capabilities are available. The only exception is the impossibility of entering certain process marks and design items.

You can transfer an image from an earlier drawing or fragment in the COMPASS-CHART. This allows the creation of a three-dimensional model to rely on the existing drawing and normative documentation. The sketch is usually located in one of the orthogonal coordinate planes, on a flat face of an existing body, or in an auxiliary plane whose position is given by the user.

To create 3D models of parts and assemblies, associative drawings and specifications, you need to rely on Operation commands. The following types of operations are available on the Compass system:

- the rotation of the sketch around the axis of the sketch plane;
- pushing out the sketch in a direction perpendicular to the plane of the sketch;
- kinematic operation - moving the sketch along the specified guide;
- the construction of the body according to several sections-sketches. What we will use in this work.

Each operation has additional options to vary the rules of the body. After the basic body is created, additional volumes are "glued" or "cut out". Each of them represents a body formed by the above operations on new sketches. If you select the type of operation, you must specify whether the created body will be subtracted from the base volume or added to it. When the volume is subtracted from the part, it is possible to achieve various openings, grooves, grooves, and when the volume is added - pulses, protrusions, ribs.

Slightly more options are available when you enter the cut or glue operation parameters than in the basic (very first) operation. Additional options make it easier to set parameters. The additional operations make it possible to simplify the parameters of common structural elements - the cladding, the rounding, the cylindrical opening, the slope, the stiffening rib. It is sufficient to specify a face, an edge or several ribs to construct the bonnet and enter its parameters - the size of the

legs or the size of the knuckle and the angle. Similarly, when constructing an opening, it is sufficient to select it and enter the appropriate parameters.

At any stage of operation, the body can be transformed into a thin-walled shell. But to do so would have to exclude one or more edges which should not be included in the shell). The order of work with the resulting shell will be the same - addition and subtraction of the bodies, formation of the cradles, curls and holes.

At any stage of operation, it is possible to remove a part of the body along a boundary, which is a plane or cylindrical surface formed by pressing an arbitrary sketch.

Auxiliary constructions. As mentioned above, the sketch can be constructed on a plane. Some operations require the indication of a guide. If the orthogonal planes, faces, and ribs that exist in the model are not sufficient to construct, the user can create auxiliary planes, axes, and spatial curves, setting their position to one of the modes provided by the system. Assembling. The COMPASS-3D assembly model consists of separate components - parts and subassemblies; each new part is modeled on the basis of already existing parts using parametric relationships.

Details and sub-sections can be created directly in the assembly or pasted into it from an existing file. In addition to user-developed models, the components of the assembly can be standard products such as fastening or supports of shafts, the library of which is part of the system supply package.

The mutual position of the components of the assembly will be determined by indicating the interfaces between them. Various types of conjugations are available in the system: coincidence, parallel or perpendicular to faces and ribs, arrangement of objects at a distance or at an angle to each other, concentric, or touch.

To create copies of components, will be used the same operations as for copying the forming elements of the component - copying over a grid, circle, curve, mirror copy. It is possible to perform different operations with assembly components: combining two parts, subtracting one part from the other. The component can also be divided into two parts. The assembly can be displayed in a "disassembled" form.

When working with a three-dimensional model, the entire construction sequence is displayed in a separate window as a "building tree". It lists all the auxiliary elements, sketches and completed operations in the model in the order in which they were created.

In addition to the tree representing the model history, COMPASS-3D stores the hierarchy of its elements. At any moment, you can view the hierarchy in a special dialog. It displays all the topological relations between the elements of the model.

There are two aspects to parameterizing a three-dimensional model in

COMPASS 3D. First, each sketch can be parametric. Its objects can be subject to various parametric connections and constraints (vertical and horizontal, parallel and perpendicular, alignment, symmetry, touch). Second, when creating a model, the system remembers not only the order in which it is formed, but also the relationships between the elements.

2.3. Description of the Part

The pulley part belongs to the disk class. The pulley serves to transfer torque from shaft to shaft. Pulley rim designs depend on belt types, which are divided into flat, wedge, polyclinic, toothed and round belts.

The surfaces of the rim of the pulley for flattened gears, except for cylindrical gears, are often shaped convex in the form of a spherical or combined surface. With this shape, the belt can be centered to prevent slipping with the pulleys. The needles or solid disks of the pulley are connected by the rim and the hub only when the pulley diameter is small. The discs may be continuous or shaped with openings for less mass.

The «pulley» part is a friction rotating part of the removable transmission, which is in the form of a wheel covered by a flexible bond. The function of a pulley is to use it as one of the main parts of a timely transmission. The working pulleys transmitting torque are fixed to the shafts by means of spun, toothed, pinned and other joints. Torque-free pulleys, as like blank pulleys or stretch rollers are freely rotated on shafts or axles. Pulverized structures vary widely. The pulleys of small diameters are embodied in the form of monolithic, medium and large diameters and have a hub and a rim connected by a disc or spokes. The large pulleys are sometimes made of two halves connected by bolts. The pulleys are made of cast iron, steel, light alloy, plastic, and sometimes wood.

The pulley for the flat belts has a cylindrical or slightly convex working surface for protecting the belt from escape. The pulleys sometimes provided with ribs for the same purpose. The wedge and polycline straps have grooves of the trapezoidal profile. The pulley under the round-section belt is provided with a groove with a rounded bottom. The pulleys of tooth-cutting gears have teeth moving in an axial direction and ribords. Stepped pulleys are used in gears with gear adjustment by transferring the belt from one stage to another. The sliding conical pulleys in continuous gears with a wide wedge belt are provided with one or both spring-loaded mobile cones and with the forced movement of one or both cones.

This piece of «pulley» is a wheel which is part of a time transfer. The function of the pulley is to transmit the torque through the belt to the shaft. The shaft is connected to the pulley by the landing with the tension, the puck is pressed and fixed with bolts. The parts are connected under pressure. During the operation of this mechanism, the rotation of the pulley on the shaft is transmitted to a toothed half-fold, after to a half-fold and to a shaft via a toothed sleeve.

The high quality of the casting ensures: the use of a casting alloy with high liquid fluidity and an optimal casting configuration, which makes it possible to use a simple casting. A form which involves the simultaneous or directional hardening of the alloy and the free extraction of casting from the mould. The casting design must also take into account the actual production possibilities: the presence of certain equipment for preparing the alloy and forming the casting at the given output volume.

The casting configuration is simple, does not require a complex shape, the casting alloy has the necessary casting properties and it can be concluded that the casting is technological. How do you choose how you get the stuff for the pulley? Sandy-clay casting is used for the required quality of the part, requirements for geometric dimensions, roughness of surfaces, economy of production, specificity of the material of the part, its good casting properties, Liquid fluidity and a small linear shrinkage select a method for producing a blank: casting into a sand-clay form.

In most cases, the blanks for piecemeal and small-scale production, most commonly used in the production of food processing equipment, are produced by casting, forging or stamping.

Castings made in sandy shapes shall be so designed as to ensure a minimum number of the surfaces of the model connector or shape to exclude detachable parts of the shape and where possible the rods. It is necessary to strive to use a single plane of the connector. It shall be capable of being created according to the rule of light shadows, according to which shadows shall not be present when the part is imagined to be illuminated by parallel rays in a direction perpendicular to the plane of the shape connector or of the rod box.

The routing process of manufacturing parts for serial production. In mechanical engineering the main types of blanks for parts are steel and cast-iron castings, castings made of non-ferrous metals and alloys, stamping and all kinds of rolling.

The method for producing the blank shall be the most economical with the specified volume of release of the part. The design and material of the component are of great importance for the choice of shape, size and method for producing the blank. The type of blank has a significant impact on the nature of the process and the economy of processing it. Metal blanks are: rolled steel and non-ferrous metals having the simple and complex profiles in the form of rods and cutting, sheet stamping, casting.

Most parts of shafts, bushes, washers and rings are made from blanks supplied in the form of round, hexagonal and square bars. Large and complex parts are prepared from piecemeal blanks produced by casting, forging or stamping. The blank has to be slightly larger than the finished part, i.e. a layer of metal to be removed during machining, which is called the peeling for processing. The amount of the dispensing shall be the smallest (i.e. shape and size of the workpiece) but a

suitable part shall be obtained. Taking into account the need for maximum approximation of the shape and size of the blank to the parameters of the finished part, a method such as press rolling is used. The result of rolling is a blank of a circular section.

3. Creating a Model

Create a three-dimensional model of the pulley according to the blueprint

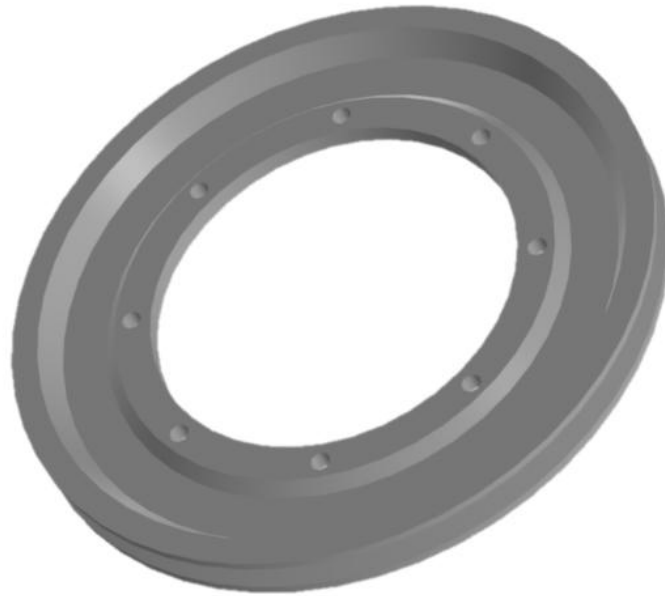


Fig. 1. 3-d model of pulley.

Before starting to develop the 3d model of pulley, you have to define the scheme as in figure 2 to make it easier to construct the model with a single squeeze command.

When you start COMPASS-3D for the first time, the Application View window appears in front of you. Click OK. The configuration is not needed yet. After running the program, you will see the start page. To create a detail, select the Detail icon on the start page.

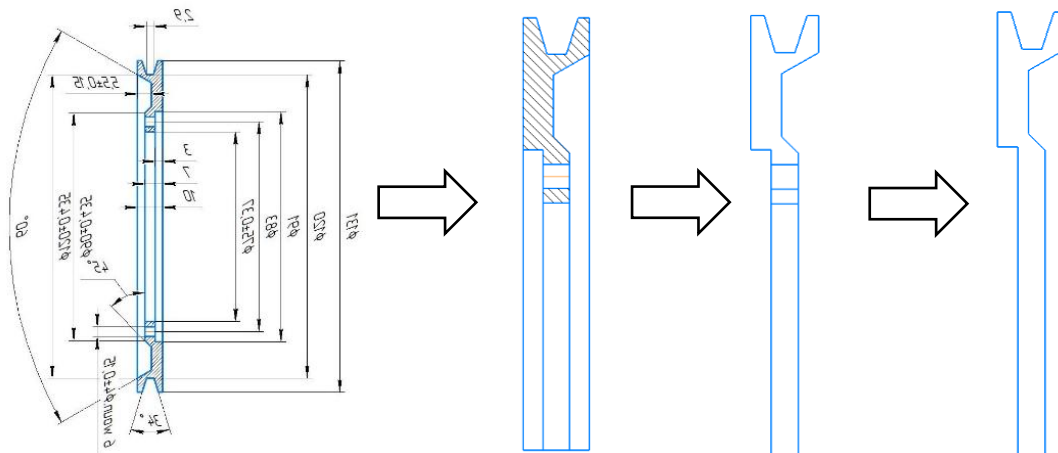


Fig. 2. Creating a pulley base sketch

1. Click the **Create** button on the Standard Panel. In the window that opens, select the type of the new document **Detail** (Figure 3, a).
2. On the **View** panel, click the list button to the right of the **Orientation** button and enter the **YZX isometry** (Figure 3, b). In the **Tree window** of the model, select **Plane XY** (Figure 3, in). The basis of any operation is the sketch. The sketches are located on the planes or faces of the model. To construct a thumbnail, click the Thumbnail in the Current State panel and select the desired plane. Go to thumbnail mode. The image is unfolded on the plane of the screen. A thumbnail icon appears in the right corner.
3. Run the axis line. Using the **Continuous Input** command, approximate the outline. Go to Thumbnail mode. In the Geometry panel, select the Segment command. Draw the cursor to the origin and specify the starting point of the segment. Position the cursor so that the angle is zero, and under the cursor the inscription «Alignment» appears. The length of this segment does not matter, so click with the mouse as soon as the line becomes horizontal. Click Abort Command or Esc on the keyboard to exit the Segment command. Point the cursor at the completed segment and click on it. The Context Pane appears. In the list of styles, select the Axis style.

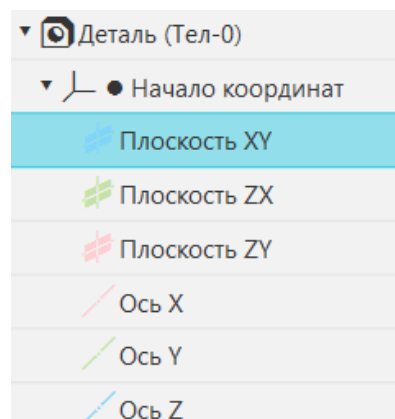
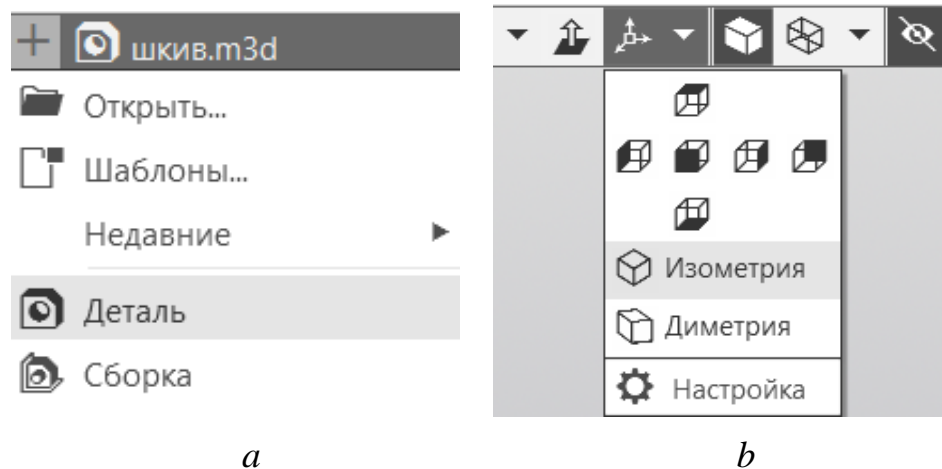


Figure 3. Command selection panels

You have created a centreline. The centreline is the axis of symmetry for the rotation operation and the main element. All further constructions will be executed relative to the axis of rotation. Construct an sketch of the rotation model from the origin, as in rice, to facilitate the construction of the model on a single push command.

4. On the **Detail Construction** panel, select in the **Operations** drop-down menu the type of action to be performed **Rotation operation** (Fig. 4, a).
5. On the Properties Panel, select the **Spheroid** object, set the straight direction, angle 360° , and on the thin wall, select **No**. Click the **Create object** button. Get the model shown in Fig. 4, b.

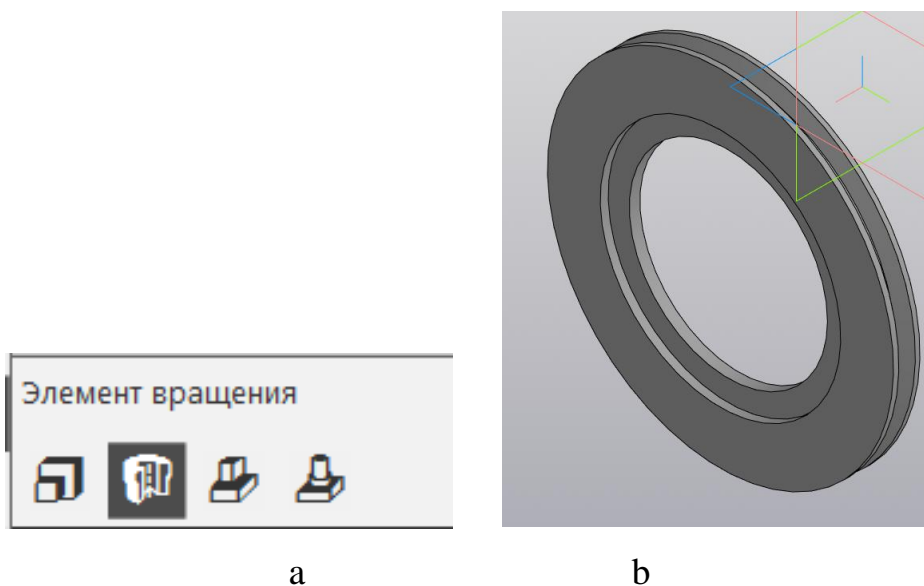


Figure 4. Command selection panels

6. Click the Create Object button. Next we will create cursors for our pulley. Click the Array button in the menu tab. We set the diameter, the number and the degree.
7. Next to get slots Click Cut By Pressing . On the Options tab, select Direction (Straight) and Distance (40 mm). Get the model shown in figure 5.

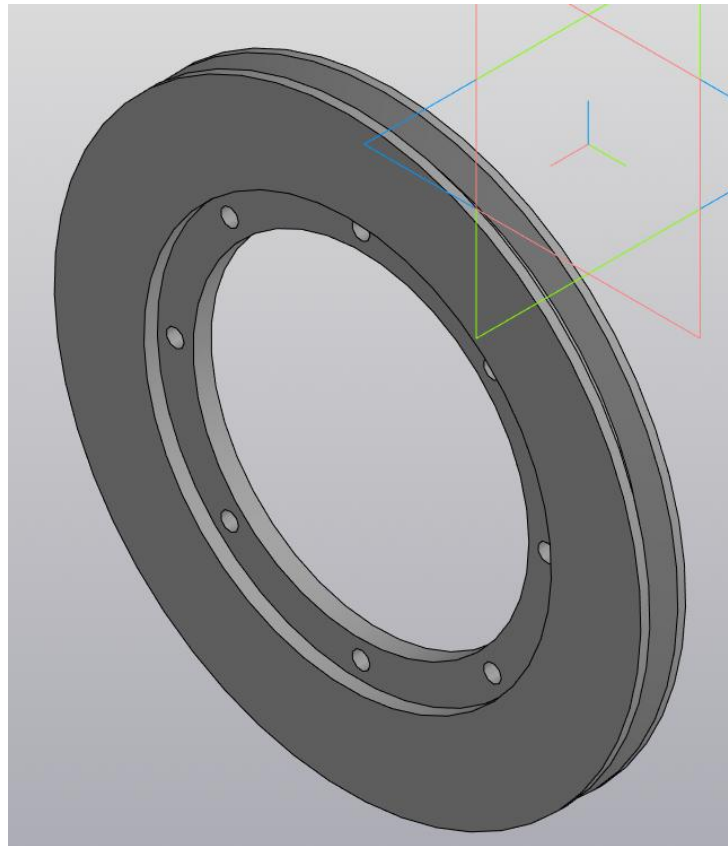


Figure 5. The model of pulley

8. Save the file with the name of “A Pulley”.

CONCLUSION.

As a result of the implementation of this diploma project, the stated goal was achieved, i.e. the technological process of manufacturing the «pulley» part in serial production was developed and the main tasks were solved accordingly. The drawing of the part «pulley» in the CAD-editor of the COMPASS on the basis of the primary paper design documentation has been created a solid model of the complex part of the pulley, on the drawing all necessary sizes, deviations and qualities for the manufacture of the part are shown.

Also the route process of manufacture of the «pulley» part has been developed by choosing the technological operations, which are useful for production of the given part in serial production, as well as the stages of operations.

Operational sketches of the CAD-edited COMPASS process have been developed sequentially through a step-by-step review of the process from the workpiece to the finished part. The sketches have the necessary sizes for processing on each operation, the base symbols, the fixing and the necessary special requirements. The main characteristics and methods of operation of different CAD/CAM systems were considered. Also, the tools of assembling in CAD/CAM COMPASS 3D have been studied in depth.

LIST OF USED LITERATURE

1. <http://cad.ru> – site about CAD/CAM systems;
2. Анурьев А.В. Справочник конструктора машиностроителя: В 3-х т. Т.1.-5-е изд., перераб. и доп.-М.: Машиностроение, 1978.-728р.
3. Горбачевич А.Ф., Шкред В. А. Курсовое-проектирование по технологии машиностроения: [Учеб. пособие для машиностроительных специальностей вузов].-4-е изд. перераб. и доп. Мн.: Выш. Школа, 1983.-256 р.
4. Большаков В. П.Б79 Создание трехмерных моделей и конструкторской документации в системе КОМПАС-3D. Практикум. — СПб.: БХВ-Петербург, 2010. — 496 с.:ил. + DVD — (Учебное пособие)
5. Ицкович Г.М. и др. Курсовое проектирование деталей машин.-М.: Машиностроение, 1965.-438 р.
6. Справочник технолога-машиностроителя. В 2-х т. Т. 2/Под ред. А. Г. Косиловой и Р.К. Мещерякова. -4-е изд., перераб. и доп. -М.: Машиностроение, 1985. 496 р.
7. Основы технологии важнейших отраслей промышленности. Учебное пособие для вузов. Под ред. И.В. Ченцова, В.В. Вашука, -Мн. 1989г