Laboratory Manual for TA202A

Manufacturing Processes II

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January 2023

TA-202: Manufacturing Processes II – Schedule of labs for 2022-23, 2nd Semester

	Introduction to machines and processes; Lab exercises; Formation of project groups; Finalization of ideas; Preparation of detailed project models and drawings.					Project work Project drawing report must be submitted in week 6 to begin project work. Mid-term project evaluation during week 9.								Final project evaluation period
Day / Week	1	2	3	4	5	6	20th -25th Feb: Mid Sem Exam	7	4th -12th Mar: Mid Sem. Recess	8	9	10	11	12
Mondays	9/1	16/1	23/1	30/1	6/2	13/2		27/2		13/3	27/3	3/4	10/3	17th – 21st April; during lab hours
Tuesdays	10/1	17/1	24/1	31/1	7/2	14/2		28/2		14/3	28/3	MU2	11/4	
Wednesdays	11/1	18/1	25/1	1/2	8/2	15/2		1/3		15/3	29/3	5/4	12/4	
Thursdays	5/1	12/1	19/1	2/2	9/2	16/2		2/3		16/3	MU1	6/4	13/4	
Fridays	6/1	13/1	20/1	27/1	3/2	10/2		17/2		3/3	17/3	24/3	31/3	

MU1: In lieu of the lab you will miss on account of a holiday on 30th March for Ram Navami there will be a make-up lab for the Thursday batch on Saturday 1/4/2023 from 9 am – 12 pm. And this is not an April fool's joke.

MU2: In lieu of the lab you will miss on account of a holiday on 4th April for Mahavir Jayanti, there will be a make-up lab for the Monday batch on Saturday 8/4/2023 from 9 am to 12 pm.

Other make up labs will be conducted as necessary to make up for any labs we might miss on account of student festivals.

General set of instructions for TA202 labs – read these very carefully

1. Safety first: since this course involves laboratory exercises with machines, students must always practice the following safety procedures:
    a. All students must come to the laboratory in shoes.
    b. No student must wear loosely fitting clothes – these can get caught in rotating machinery.
    c. Students with long hair must tie their hair.
    d. Students must take permission of the Lab Staff/Tutor before handling any machine.
    e. Students must ensure that the floor around the machine they are operating is dry and not oily to avoid slipping.
    f. Students must not lean on any machine in operation.
    g. When operating machines, students must wear safety glasses and gloves as necessary. All students will be issued an apron, a set of safety glasses, and a set of safety gloves. Students not wearing aprons will not be permitted to work on any equipment.
    h. No unauthorized use of mobile phones by students is permitted. Those found using their phones will be marked absent for that lab day and will accordingly loose that part of the grade.
2. Discipline also first: Lab runs from 2 – 5 pm. Please make sure you are in the TA202A labs by 1:50 pm. Those students who turn up after 2:05 pm will be marked absent from the day's exercise and will be given a '0' for that exercise. Students will also loose a component of the grade reserved for lab attendance if they are being marked absent on account of being late. Guides/TAs/Tutors reserve their right to also deny students entry if the students are late.
3. Printing a copy of this manual is not necessary and is discouraged. Copies can be issued from the stores in the lab.
4. Students must handle all equipment in the lab with care.
5. While working on their respective projects, students may issue instruments and tools from the tool room by depositing their ID card. ID cards will be returned on the safe return of tools/instruments borrowed.
6. Power to the machines will be turned off 10 minutes before the end of the laboratory session to allow students to return the tools/instruments they may have borrowed.
7. Students are required to clear off the chips from the machine they were using/operating and lubricate the guides etc. at the end of the lab session.
8. Lab exercises 4 – 5 will require at least three students in every group to bring their laptops to the lab. These exercises are to be conducted on the student's own laptop during the lab. For further instructions, please read the 'Instructions for lab exercises 4 – 5' section.
9. Students must pre-install the software necessary to complete the lab exercise 4 – 5. Links to download the software to be used are provided in the 'Download software for lab exercises' section.
10. In the case any student misses any of the lab exercises 3 – 5 for valid reasons (duty leave, medical leave, other valid reasons), those students must make up for the exercise within the same week by joining another section. Please discuss this with the instructor. If making up for the exercise is not possible in the same week, students must contact the instructor to work out alternatives. If the student does not reach out to the instructor to make up for a missed exercise(s), even with valid reasons for missing the exercise, the student will be awarded a '0' grade for the exercise(s) missed. In the case that a student has missed a lab exercise and has no valid reason for missing the exercise(s), that student will be awarded a '0' for that exercise. No make-up of the missed exercise(s) will be permitted to such students.

Organization of this manual

Part A of this manual provides an overview of the lab exercises and part B details the exercises.

Please note

Since the manual spans 300+ pages, lets save some trees. Do not print the manual. Students are not expected to possess a hard copy of the manual. Copies of this manual will be provided to you in the labs, which must be issued from and returned to the stores in the labs.

Part A

Contents

Overview of the lab and lab exercises for TA202A

TA202A is a lab-based course with an emphasis on experiential learning, i.e., learning by doing and making. Students, on being introduced to the working of machines, and on completion of certain lab exercises, are expected to make a project in the lab during the semester. There will be five lab exercises dedicated to introducing the working of machines and equipment to be used by students in the lab during the semester. Each exercise will take place in one turn of the lab, i.e., in one week. Hence, the first five weeks will be spent in exercises and preparatory work for the project, and the remaining six weeks will be dedicated solely for project work.

The project that students are expected to make in groups, consists of conceptualizing a device/product/task, followed by designing the parts and making drawings for manufacturing those parts, followed by planning of machining and other operations, and fabrication of the components, including 3D printing some parts, followed by controlling some parts using a programmable board and a motor, and finally, assembly of the device/product. A detailed project report must be submitted during the final evaluation of the completed project. This report should contain general descriptions of the completed project, design details, detailed drawings, and assembly drawings, followed by suggestions for improvements, if any, and any other information the students think is important.

Each project must necessarily have at least one part that is 3D printed and one part that is made on the CNC machine. Each project must also use the Arduino kit supplied to the group.

An outline of the week-wise lab schedule is:

Lab	Activity	Milestone	Grades*
1	Intro. to machines and machining processes	-	-
2	Intro. to machines and machining processes. Assign groups & guides	Groups formed. Groups to brainstorm about ideas as homework	-
3	Assemble + Disassemble 3D printers.	Discussion on feasibility of project ideas conceptualize	5% for the exercise + 1% for the project ideas
4	CAD exercise + project discussions	Finalization of project idea	5% for the exercise + 1% for the quality of the project idea
5	CAM exercise + Arduino exercise + project discussions	Groups to demonstrate progress in modelling and drafting	10% for the exercises + 3% for showing at least six 3D models of the parts to be used in project, with three part drawings of components
6	Start of project work	Submit project files	5% for complete project file
6 – 11	Project work	There will be a mid-term project evaluation during week 9	10% for demonstrable progress made in the project.
12	Final project evaluation	See details provided later	22.5%

* Lab grades will also include 2.5% for attendance, and 10% based on your guide's evaluation of your individual progress

An overview of the five lab exercises is provided below, and detailed instructions for all exercises are provided in subsequent sections of this manual.

Lab exercise 1 and 2: Introduction to machines and machining processes

Emphasis in this lab and course is to use secondary manufacturing processes such as machining, traditional and not, to make parts for the respective projects. Traditional manufacturing processes that students will be introduced to in the labs are:

- turning (to make round parts)
- milling (to make parts of other shapes)
- drilling (to make holes)

Above processes will be introduced to students on manually operated machines, and on machines controlled through computers, called Computer Numeric Controller machines, or CNC machines. Students will also be expected to operate machines on their own to make parts necessary for their projects. Students will be provided training before being allowed to use the machines on their own. Detailed instructions on what this exercise will constitute is provided later in this manual. The objective of this exercise to introduce machines and machining processes to you by having you make/observe how to make a spur/bevel gear and a L-clamp assembly. These parts are representative of things you might use in your projects.

Lab exercise 3: Assembling, printing with, and disassembling a 3D printing machine

In addition to using conventional machine tools, manual and/or CNC, students will also be introduced to additive manufacturing, i.e., 3D printing. In one lab exercise, each group of students will be given a machine in a box and students will be expected to assemble the machine, and then again disassemble the machine. This will constitute a lab exercise and is aimed at introducing concepts of assembly following a detailed set of instructions. Students will also be expected to print at least one or more parts for use in their respective projects. Parts to be printed will be made of PLA. Detailed instructions are provided later in the manual.

Lab exercise 4: CAD – computer aided design

Since the goal of this lab-based course is to make something, and since students are expected to make 3D CAD models of their parts and assemblies, and since students may or not have been introduced to CAD before, an exercise on CAD is planned in the labs. More on this later in the manual.

Lab exercise 5: CAM – computer aided manufacturing + Arduino board-based DC motor control

This will be a three-part exercise.

Parts 1 and 2: Since each project is required to have at least one part made on a CNC machine, and at least one part made by 3D printing, a lab exercise on generating CNC codes (G codes) for representative parts to be made on the CNC machine and on the 3D printer will be conducted. This will be a two-part exercise, and detailed instructions are provided later in the manual.

Part 3: Since this is the era of DIY (do-it-yourself), and since a lot of DIY projects involve use of programmable boards to control motors and other things, students will be introduced to controlling a DC motor using a programmable Arduino board. A kit including an Arduino board, a motor driver, a battery pack, and a DC motor will be issued to each group of students. A lab exercise will instruct the use of these things to drive the motor. Each group is expected to use the kit provided to them in their projects to drive whatever the group deems fit. Detailed instructions on the exercise are provided later in the manual.

Lab exercise 1 and 2: Introduction to machines and machining processes

For this exercise, the students will be divided into two sections (A and B) as below. Each section will conduct one activity in one week, and the other activity in the week after. Each section will have approximately 45 students divided into groups of 7 or more students in every group – depending on the number of students doing the exercise on that day. Each group will be assigned a guide. Guides are trained technical staff in the TA202A labs. Guides will introduce working of the turning, milling, and drilling machines in the labs along with the kind of possible machining operations on each of the machines. The objective of this exercise to introduce machines and machining processes to you by having you make/observe how to make a bevel gear and a L-clamp assembly. These parts are representative of things you might use in your projects. Hence, we think it best to demonstrate machines through making parts that you will also make during your projects.

Week	Activity	
	Making a gear	Making a L-clamp assembly
1	A	B
2	B	A

Students will not be trained to operate machines during this orientation, instead the objective is simply to introduce the machines that the students will use when they will make parts to execute their respective projects. Training to operate these machines will be provided by guides assigned to groups when the respective groups must use the machines, i.e., week six onwards, when actual project work will begin.

CNC machines will not be introduced in this lab exercise. Working of CNC machines will instead be discussed in class. Moreover, since each group must necessarily make at least one part of their project on a CNC machine, its working will be explained to students by their respective guides, as and when the group needs to use the machine in the labs. This will happen week six onwards, when actual project work will begin.

In general, machining is a secondary manufacturing process, and it is subtractive in nature, i.e., the part is given its final form by removing material. The mechanics of metal removal are complex in nature and studying cutting mechanics is beyond the ambit of this course. However, the general principle of material removal is by providing a relative motion between the work material being cut, and the cutting tool. This relative motion can be obtained by a combination of rotary and translational movements of either the tool or the workpiece, or both. Machine tools, in general, provide two kinds of relative motions. The primary motion is responsible for the cutting action and absorbs most of the power required to perform the machining action. The secondary motion of the feed may proceed in steps or continuously and absorbs only a fraction of the total power required for machining. When the secondary motion is added to the primary motion, machine surfaces of desired geometric characteristics are produced. The tool shears away material, i.e. material transitions from elastic to plastic and shears/chips away resulting in a new surface. The kind of surface that is produced by the operation depends on the shape of the tool and the path it traverses through the materials.

Having outlined the general principle of material removal in machining, an overview of the machines, their parts, and the possible machining operations on each of the machines is discussed below for the benefit of those interested. Though there are plenty of types of machines and machining operations, this lab, and hence this manual, will primarily involve the use only of turning, milling and drilling processes, and as such, only these are outlined below.

Turning on a lathe machine

A schematic of a typical manually operated turning machine, also known as a lathe machine, is shown below. Your guide will explain to you the working principle of this machine and will also outline the function of different major components of the machine.

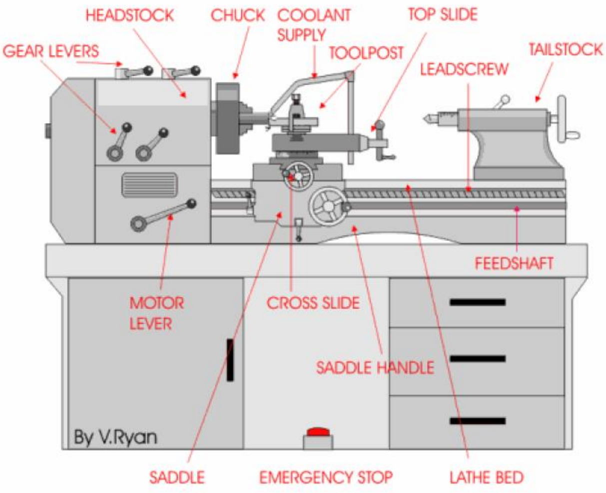


Fig. 1 Schematic of a lathe machine

Lathe machines are primarily used to make round parts. When the workpiece is rotated about an axis and the tool is traversed in a definite path relative to the axis, a surface of revolution is generated. When the tool path is parallel to the axis, the surface generated is a cylinder - as in straight turning or boring operations. Turning machines can also be used for hole making, i.e., drilling. An overview of some possible machining operations on the lathe machine is shown below. Your guide will provide you an overview of some of these machining operations.

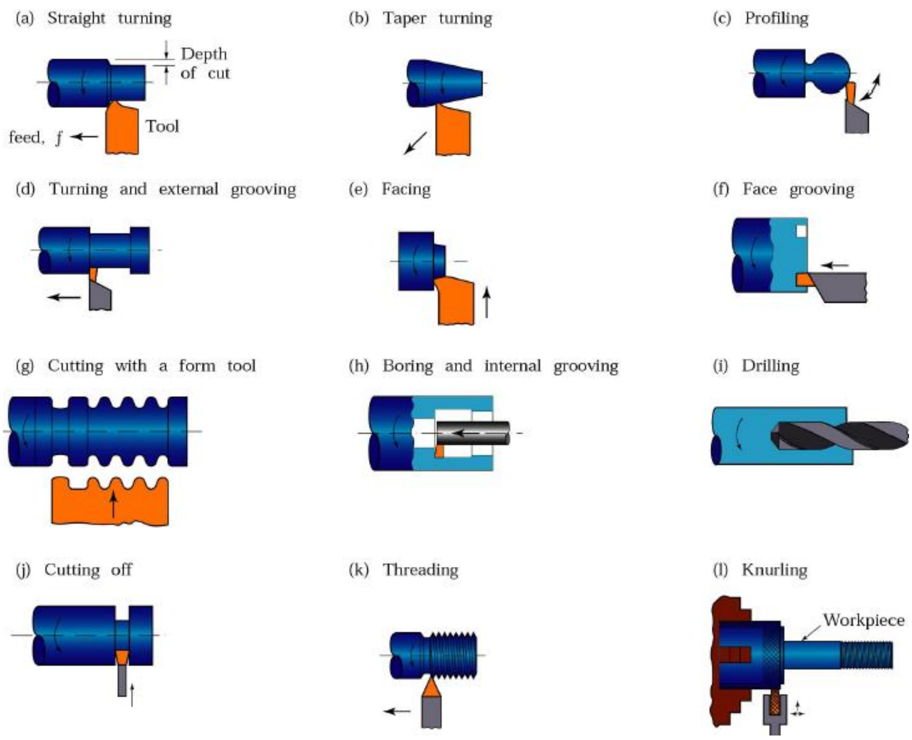


Fig. 2 Schematic of some possible machining operations on the lathe machine. Figure source: Kalpakjian and Schmid's book

Of all the possible machining operations on a lathe machine, straight turning, shown above is the most common, and the tool used to perform this operation, with all its geometry is shown in the schematic below. Your guide may demonstrate this geometry to you in the lab.

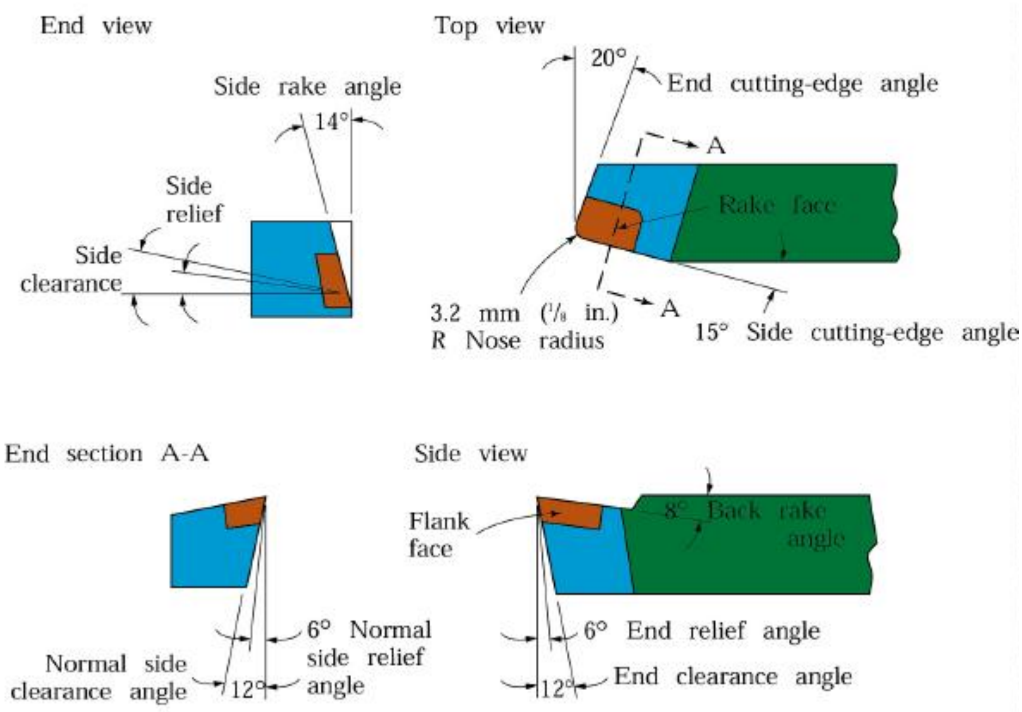


Fig. 3 Typical tool geometry for a turning tool. Figure source: Kalpakjian and Schmid's book

Lathe machines, as well as all other machines can be operated over a range of feeds, speeds, and other cutting parameters. Selection of these parameters is a function of the material being cut. More about this will be discussed in class.

Milling machine

A schematic of a typical manually operated milling machine is shown below. Your guide will explain to you the working principle of this machine and will also outline the function of different major components of the machine.

Milling machines are usually used to make parts with varying shapes. Milling is also a multipoint cutting operation as opposed to turning, which is usually only a single point machining operation. In plane milling, the cutter revolves and moves over the work piece as shown. The axis of the cutter is parallel to the surface generated.

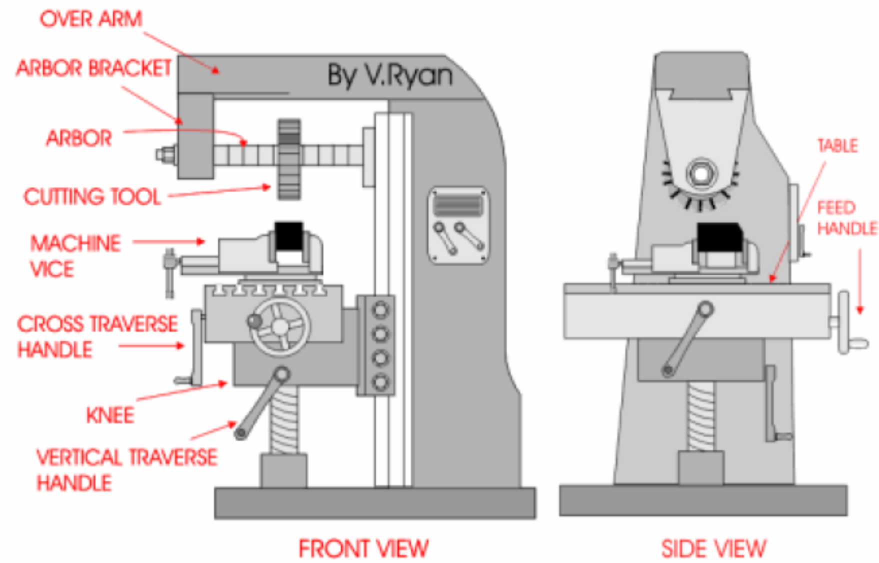


Fig. 4 Schematic of a milling machine

An overview of some possible machining operations on milling machines is shown below. Your guide will provide you an overview of some of these machining operations. The milling machine will also be used in this lab for making gears using form cutters as shown below.

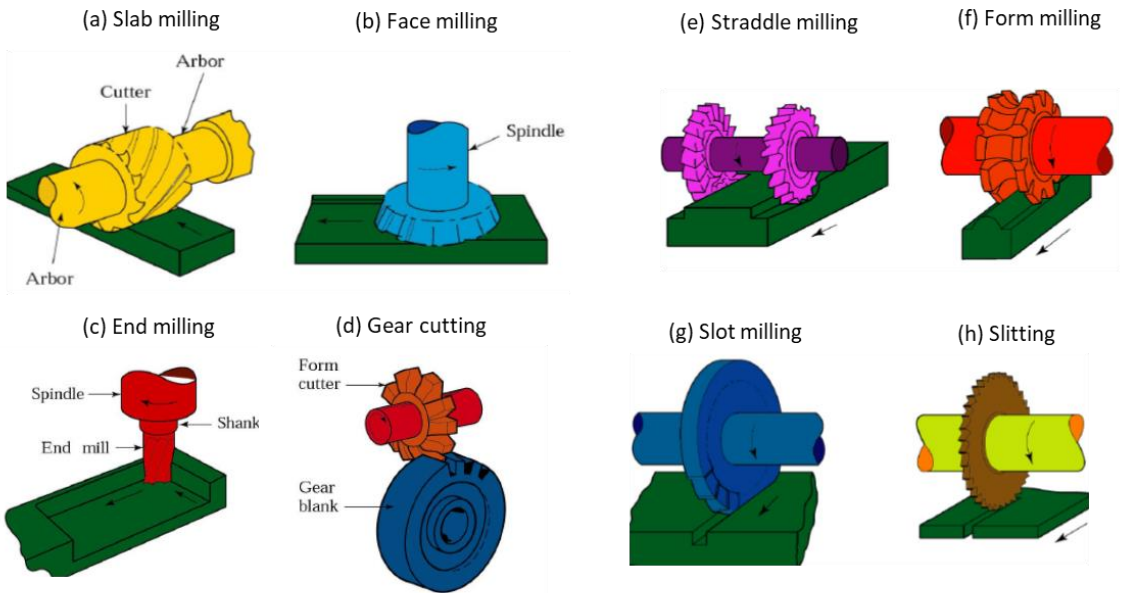


Fig. 5 Schematic of some possible machining operations on the milling machine. Figure source: Kalpakjian and Schmid’s book

Drilling machine

A schematic of a typical manually operated drilling machine is shown below. Your guide will explain to you the working principle of this machine and will also outline the function of different major components of the machine. Pay attention to the continuously variable speed transmission mechanism of this machine. It is interesting and different than the speed changes in the turning and milling machines.

In drilling, the drill may turn and be fed into the workpiece that is clamped in the vice on the machine. Drilling may also be performed on a lathe machine, in which the workpiece may revolve while the drill is fed into it.

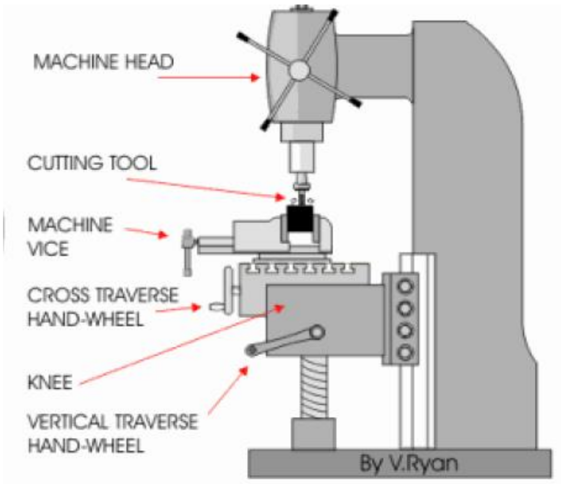


Fig. 6 Schematic of a drilling machine

An overview of some possible machining operations on drilling machines is shown below. Your guide will provide you an overview of some of these machining operations, and will also show you the geometry of typical drills.

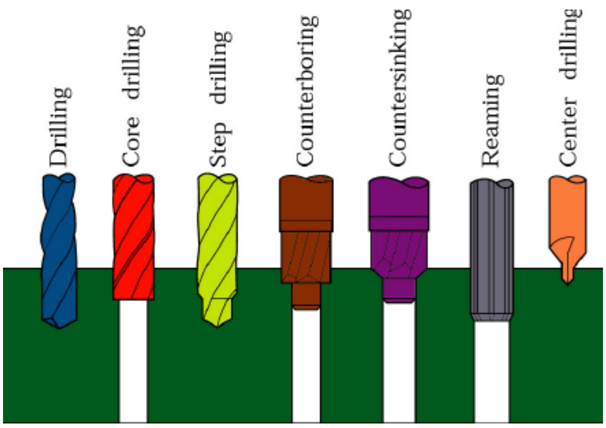


Fig. 7 Schematic of some possible machining operations on the drilling machine. Figure source: Kalpakjian and Schmid's book

Having introduced the general operations possible on the turning, milling and drilling machines, specific procedures and the steps involved in making a spur/bevel gear and the steps involved in making a L-clamp assembly are detailed in part B of this manual.

Lab exercise 3: Assemble + disassemble 3D printers (5% of grade)

The goal of this lab exercise is to assemble a 3D printer followed by disassembling the same. Each group of students will be given a machine in a box and students will be expected to assemble the machine, and then again disassemble the same machine – all within three hours. This will constitute a lab exercise and is aimed at introducing concepts of assembly following a detailed set of instructions.

<u>Please read these instructions carefully before start of this exercise:</u>

1. This exercise is worth 5% of your grade.
2. For a full grade, the assembly and disassembly must be successfully completed with the stipulated three hours of lab time. No extra time will be provided.
3. DO NOT TURN ON THE MACHINE. This is written in CAPS for a reason. If machines are turned on with the wrong electrical connections made by the students, it may damage the machine. Hence students should not turn on the machine. If any group turns on the machine, that group will be awarded a '0' for the exercise.
4. On successful assembly of the machine, the students must have the assembly checked by their assigned guide/TAs/Tutor.
5. Once the guide assesses the assembly and is satisfied that the assembly is complete, students must proceed with carefully disassembling the machine and putting back all components in their respective pouches/boxes.
6. Successful completion of the exercise also includes:
    a. no damage to any of the parts
    b. no missing screws or any other parts
7. The guide/Tutor will asses the disassembled machine in the box, and:
    a. if any screw is found to be missing, 1% will be deducted from all members of the group for every missing screw
    b. if any part is found to be damaged due to mishandling etc., depending on the criticality of the part, the guide/TAs/Tutor may penalize all students in the group anywhere between 0% - 5% of the grade.
8. If the group cannot complete this task in time but has completed the assembly process – a grade of 2.5% will be assigned to all students in the group, i.e., 2.5% will be deducted.
9. 3D printing will be demonstrated to the students on a separate pre-assembled machine during this exercise. Each group will have the opportunity to print parts for their respective projects and may operate the machines at a later stage during the semester.

Students are requested to be careful during this exercise. The disassembled machine in the box will serve as the starting point for the lab exercise to be conducted on subsequent days. Furthermore, the same 3D printers will be used during the lab for printing parts necessary for projects. Hence, further care is encouraged during the exercise.

Common instructions for lab exercises 4 and 5

Please carefully note the following:

1. Lab exercise 4 is a fully computer-based exercise, and lab-exercise 5 is partly computer-based.
2. Lab exercise 4 is worth 5% of your grade, and lab exercise 5 is worth 10% of your grade. These exercises must be done within lab hours only, i.e., the exercise must be started and completed during the allotted three lab hours. Students who may have attempted the exercise before coming to the labs, must repeat the exercise afresh in the lab. The guides/TAs/Tutors will monitor this.
3. At least three students from every group are expected to bring their personal laptops with pre-installed software for conducting these exercises in the labs.
4. Three plug points per group will be provided to help charge laptops as necessary.
5. If, for whatever reason, software necessary to conduct labs has not been pre-installed on the laptops, i.e., pre-installation should be complete before coming to the labs, and if it is not, students will be penalized at the rate of 2% of their grade.
6. Exercises have been designed such that they can be easily completed within the allotted three hours. Yet, if student groups are not able to complete the exercise, student groups will be graded based on what they have successfully completed within the allotted three hours. Details for grading for each exercise is detailed in the relevant section for that exercise.

Download software for lab exercises

Please download and install the following software before coming to the labs

Lab	Software	Download link	System requirements
4	CAD: Autodesk Inventor Professional 2020*	https://www.autodesk.in/products/inventor/overview?referrer=%2Fproducts%2Finventor%2Foverview#0	https://knowledge.autodesk.com/support/inventor-products/troubleshooting/caas/sfdcarticles/sfdcarticles/System-requirements-for-Autodesk-Inventor-2020.html
5	CAM for CNC: Autodesk Fusion 360*	https://www.autodesk.com/products/fusion-360/students-teachers-educators	If you can install the above software, your system is fine to handle these too
	CAM for 3D printing: Craftware	https://craftbot.com/craftware/	
5	Arduino	https://www.arduino.cc/en/Main/Software	

*Select the free trial version, and the user category of a student or a teacher. Follow the instructions. Make sure to check for compatibility of your laptop with the system requirements.

Lab exercise 4: Computer aided design - CAD (5% of grade)

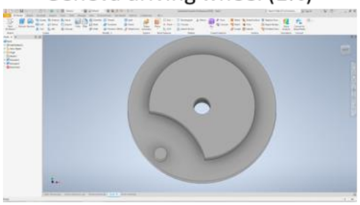
Since the goal of this lab-based course is to make something, and since students are expected to make 3D CAD models of their parts and assemblies followed by making part drawings to manufacture things, and since students may or not have been introduced to CAD before, this exercise on CAD is planned in the labs to introduce and orient students in the use of CAD.

Since students have already taken the TA101 course, this exercise will assume students are familiar with ideas of sketching, projections, dimensioning, sectional views, etc., and as such this exercise will not review those ideas.

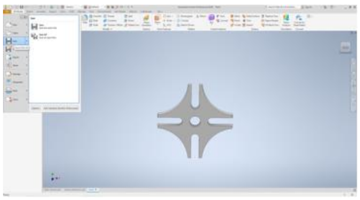
This exercise is more like a tutorial, with explicit and detailed instructions provided below for students to be able to successfully complete five tasks (i) make a Geneva driving; (ii) make a Geneva driven wheel; (iii) Assemble the mechanism in an support bracket with shafts; (iv) in the assembly also make a pair of bevel gears; (v) make a part drawing. Students should have all tasks assessed by the guide/TAs/Tutor. Grades will be assigned for each task.

An overview of the parts, assembly and drawing to be made is shown below:

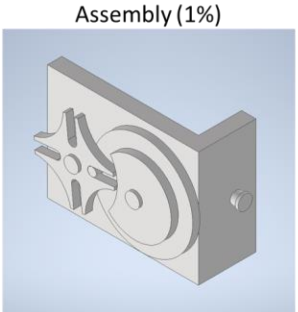
Geneva driving wheel (1%)



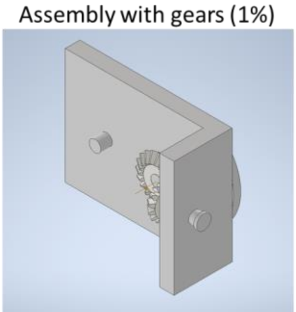
Geneva driven wheel (1%)



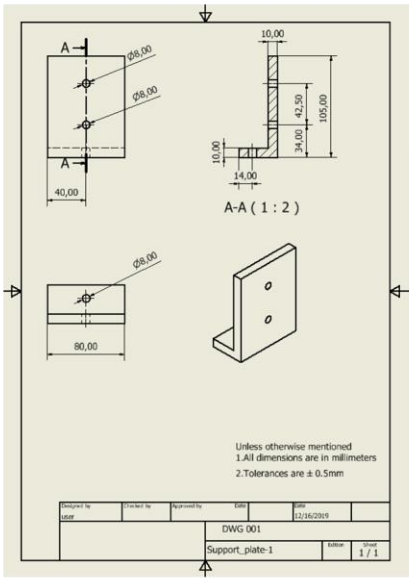
Assembly (1%)



Assembly with gears (1%)



Part drawing (1%)



Grading:

1. Geneva driving wheel (successful completion is worth 1%),
2. Geneva driven wheel (successful completion is worth 1%),
3. Assemble the mechanism in a support bracket with shafts (successful completion is worth 1%),
4. in the assembly also make a pair of bevel gears (successful completion is worth 1%), and
5. Making a part drawing (successful completion is worth 1%)

This exercise is designed such that it can be easily completed within the three hour lab duration. If any group finishes before time, that group is expected to work on their respective project. If any group of students is unable to finish this lab exercise in time, grades will be assigned for the tasks completed. No extra time will be permitted.

The tutorials provided in part 'B' are self-explanatory. Please follow all steps shown for successful completion of this exercise.

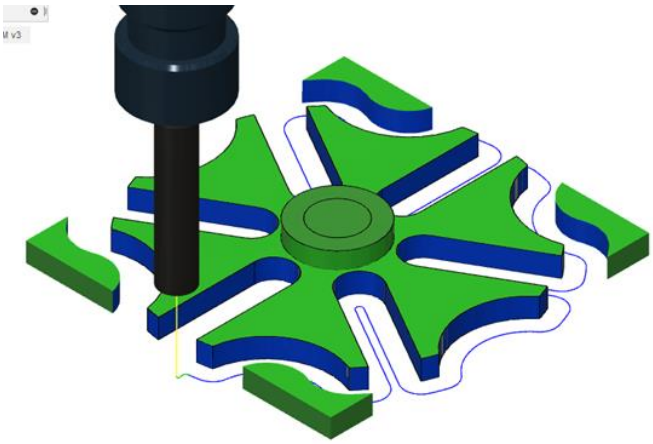
Lab exercise 5 – Part A: Computer aided manufacturing – CAM (5% of grade)

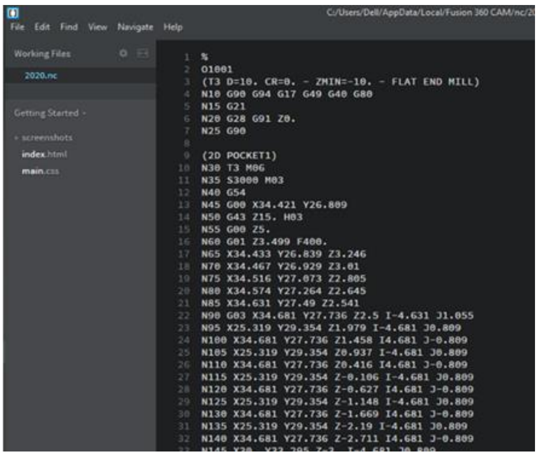
Since each project is required to have at least one part made on a CNC machine, and at least one part made by 3D printing, a lab exercise on generating CNC codes (G codes) for representative parts to be made on the CNC machine and on the 3D printer will be conducted. This will be a two-part exercise. One part will involve a CAM exercise for milling the driven wheel of the Geneva mechanism, and the other part will involve generating a G code to 3D print a spur gear.

Numerical controlled machines are more accurate than manually operated machines, and can produce components more uniformly, faster and in the long-run tooling costs are smaller but the initial investment is higher. Numerical Control (NC) involves a system in which actions are controlled by direct insertion of numerical data. The system automatically interprets symbolic instructions to control machine tools and other manufacturing systems. Symbolic instructions or the numerical data required to produce a part is called a 'part program'. The CNC machine, and the 3D printer, are both numerically controlled. Hence, this exercise is geared towards making such ‘part programs’ to control these machines to make the desired parts.

<u>CAM for CNC machines</u>

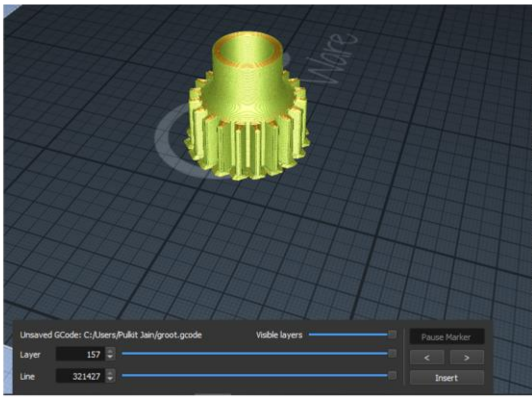
Making a part program starts with a CAD model for the part. In the case of CNC machines, this is followed by selection of the tools required to make the part, defining cutting parameters, and then selecting a tool path to be followed. Modern ‘part-programming’ methods use advanced software tools to do all this. Since your project will require you to make a part on the CNC machine, this lab exercise will guide you through the requisite steps in making a part program using ‘Autodesk Fusion 360’ – a modern CAM software. The part for which the G code is to be generated is shown below. Detailed steps are provided in the tutorial. The tutorial provided is self-explanatory.

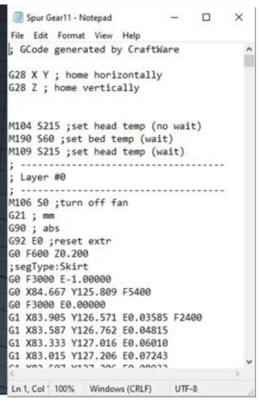




<u>CAM for 3D printing</u>

In the case of controlling 3D printing machines, we will also start with the CAD model of the part to be printed. This will be followed by defining parameters for printing speed, layer height, and also the internal constructional details of the part to be printed. The part for which the G code is to be generated in this lab exercise is shown below. There are several mechanisms for 3D printing. In this lab we will use the ‘Fused deposition modelling – FDM’ type. An overview of additive manufacturing processes will be discussed in class. The lab exercise is simply to outline the detailed steps involves in generating G codes for the part to be printed. The tutorial provided is self-explanatory.





Grading

This lab-exercise is worth 5% of your grade:

- 2.5% for the CAM for the CNC, and
- 2.5% for CAM for the 3D printer.

This exercise is designed such that it can be easily completed within the three-hour lab duration. If any group finishes before time, that group is expected to work on their respective project. If any group of students is unable to finish this lab exercise in time, grades will be assigned for the tasks completed. No extra time will be permitted.

Lab exercise 5 – Part B: Arduino board-based DC motor control (5% of grade)

Since this is the era of DIY (do-it-yourself), and since a lot of DIY projects involve use of programmable boards to control motors and other things, students will be introduced to controlling a DC motor using a programmable Arduino board. A kit including an Arduino board, a motor driver, a battery pack, and a DC motor will be issued to each group of students. This lab exercise will instruct the use of these things to drive the motor. Each group is expected to use the kit provided to them in their projects to drive whatever the group deems fit.

An overview about Arduino, sourced from: https://www.arduino.cc/ is provided below.

<u>What is Arduino?</u>

Arduino is an open-source electronics platform based on easy-to-use hardware and software. It's intended for anyone making interactive projects.

<u>What is an Arduino board?</u>

Arduino senses the environment by receiving inputs from many sensors, and affects its surroundings by controlling lights, motors, and other actuators.

<u>Which board will you use?</u>

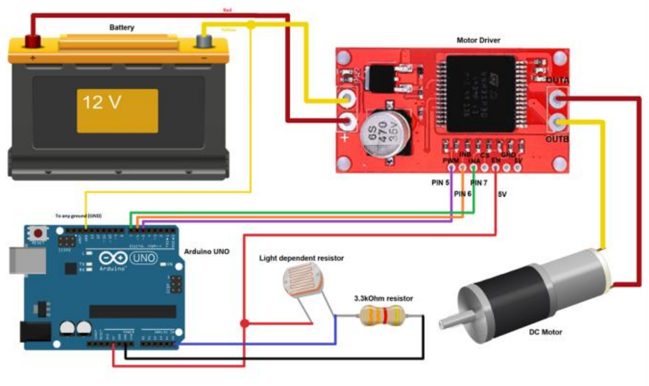
Arduino Uno. This is a microcontroller board. It has 14 digital input/output pins (of which 6 can be used as PWM - pulse width modulation outputs), 6 analog inputs, a 16 MHz quartz crystal, a USB connection, a power jack, an ICSP header and a reset button. It contains everything needed to support the microcontroller; simply connect it to a computer with a USB cable or power it with a AC-to-DC adapter or battery to get started. You can tinker with your UNO without worrying too much about doing something wrong, worst-case scenario you can replace the chip for a few hundred rupees and start over again.

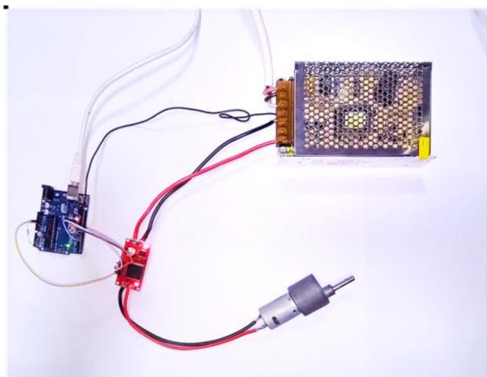
<u>What does the Arduino software do?</u>

You can tell your Arduino what to do by writing code in the Arduino programming language and using the Arduino development environment.

<u>Overview of the lab exercise:</u>

A schematic of what is expected in this lab exercise is shown below. You will write a program (detailed instructions in the tutorial later). That program will instruct the Uno board, which will drive a DC motor through a motor driver as shown. The motor driver will be powered by a 12 V DC battery pack as shown below.





Grading

This lab-exercise is worth 5% of your grade:

- 1% for successfully making all electrical connections as per the schematic shown in the tutorial
- 2% for successfully using a light dependent resistor (LDR) and turn the motor ON and OFF by interrupting the light falling on the LDR
- 1% for successfully controlling the speed of the motor, and
- 1% for successfully controlling the spin direction of the motor.

This exercise is designed such that it can be easily completed within the three-hour lab duration. If any group finishes before time, that group is expected to work on their respective project. If any group of students is unable to finish this lab exercise in time, grades will be assigned for the tasks completed. No extra time will be permitted.

Note

Kits must be issued from the tool room and must also be returned at the end of the exercise.
Students are liable to replace any item that they may damage in this kit.

Instructions for submitting your project reports to start your project work

General instructions for your projects:

1. All projects must have at least one 3D printed part, and up to a maximum of five 3D printed parts. If specific designs call for the use of more 3D printed parts, how many and which parts to print is something you and your guide will negotiate.
2. All projects must have at least one part made on the CNC machine(s) - turning and/or milling. Again, how many and which parts to make on the CNC machine(s) is something you and your guide will negotiate.
3. All other parts to be made for your projects must be made on the conventional machines available in the TA202 labs.
4. All projects must use the Arduino kit to be provided. You may use this to do whatever you like in the project. The kit includes:
   - one DC motor (max speed of 30 rpm, and max torque of 1 Nm);
   - two motor driver (one has a 1.5 Amp current rating, and the other is of 15 Amp);
   - one 12 V batter pack, one SMPS power supply, and
   - one Arduino UNO board, and several cables and wires
5. If you like and need to, you may use additional motors and sensors as you deem necessary. We will not provide these. You will have to make your own arrangements. Please have your guides approve any additional items you may wish to use.
6. All groups are expected to have finalized their project ideas by week 4. By week 5, we expect you to show us substantial progress in modelling and drafting of parts for your projects. Substantial progress includes showing us (your guides) 3D isometric views of your projects made in CAD. Remember that your guides are continually evaluating individual performance in all groups, as well as overall group progress.

Specific instructions for your project reports to be submitted before start of work

For your interim project reports to be submitted in the labs during week 6 please note the following:

1. You are expected to make two copies of the reports. One to be submitted to the lab stores, and another to be with you and your guides.
2. All reports should be spiral bound or in a flat file with a transparent front cover.
3. All reports should be structured as follows:
   - A cover page introducing your group number, project title, names of all group members, and your guide's name.
   - The second page must necessarily be a detailed isometric view of the entire project assembly. Please make sure to label all main parts.
   - The third page must necessarily be the bill of materials - which is essentially a list of the raw materials, sub-assemblies, intermediate assemblies, sub-components, parts, and the quantities of each needed to make your project. The more detail, the better.
   - Following pages must have details of sub-assemblies as appropriate and part drawings - each on a separate page. All part drawings must be dimensioned appropriately with tolerances and scales and materials to be used. You may also specify on these drawings which parts you plan to make traditionally, which on the CNC machine, and which on the 3D printer.
   - You may include any sizing calculations, as appropriate.

Timely submission of complete reports carries a 5% grade. Groups without reports will be marked absent for the day, and hence will receive a ‘0’ grade.

Instructions for preparing and submitting final project reports

Final project evaluation metrics:

22.5% of your final grade will be decomposed into:

- Successful working demonstration of project objectives: 7.5%. If some part of the project is not functional and/or demonstrable, you will be penalized accordingly.
- Everyone in the project group knowing how everything in the project was made: 5%. This will be gauged by questions from the Tutors.
- Quantum and complexity of project work: 5%. This will be gauged by how many parts were made in the project, how many were recycled, how difficult it was to make the marks that were made, and how difficult it was to control the motor(s) to drive the mechanism(s) of interest.
- Completeness of project report: 5%

Project report expectations:

All groups will submit their final reports to us. Report should be compulsorily structured as follows:

1. Cover page: Should include the project title, group number, list of group members, name of your project guide, and an updated isometric view of the project or a photo of the completed project.
2. Project overview: This is the second page of the report. The overview should have only four sentences. The first sentence should state the project objective. Second sentence should state how many parts were manufactured. The third sentence should summarize the cost of the project (see below for how to estimate the cost). The fourth sentence should suggest improvements that could be made to the project. Word count for these four sentences should not exceed 150 words.
3. This is to be followed by a table of contents.
4. This should be followed by isometric assembly drawings with part labelling.
5. This should be followed by detailing how you sized and selected the motor(s) you may have used. This section may also include any other design calculations you may have carried out.
6. Then a list of parts. List of parts should list in a five-column format: part name; the quantity of each part; material used for each part; if the part was manufactured/bought; if manufactured, the final column should list the machining operations to make the part.
7. This is to be followed by: detailed part drawings for all parts that were manufactured. Drawings should be with dimensions and tolerances, as necessary, and should state who made it, what material the part was made of, and what the scale of the drawing is. If any of the drawings are found to not meet our expectations and/or do not reflect the real manufactured parts, 1% of your grade will be deducted.
8. Finally, discuss cost analysis for your project. For this, you may assume that the material cost for steel used in your projects is Rs. 100/kg, for aluminium it is Rs. 350/kg, and for Delrin it is Rs. 400/kg. You must also estimate how long it took to make parts on the drilling, milling, turning, CNC, and 3D printing machines. The rates for these machines are: drilling - Rs. 75/hour; milling - Rs. 250/hour; turning - Rs. 150/hour; CNC - Rs. 500/hour; 3D printing - Rs. 100/hour. In your cost analysis, you may assume that the electric kit (that includes the battery, SMPS, motor, board, drivers, etc.) we have supplied costs Rs. 1000. If you have bought additional motors/sensors, include those costs too.

Log Sheet - To be submitted by Individual Students

Group # Sec.

Name:

Job Assigned:

(1) No. of parts fully completed from the planned and approved main drawing. (As per index submitted earlier)

S.N.	Part Name	Work done
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		

(2) No. of parts assembled together

S.N.	Names of parts assembled	
1		
2		
3		
4		
5		
6		
7		

List of material available in the labs for you to use for your projects:

M. S. ROD :- 6, 8, 9.5,12, 12.7, 16, 20, 22, 25, 32, 36, 40, 53, 63, 70, 90, 100,120 (diameter in mm)

M.S. FLAT (in mm) :- 12X3,25X6,50X6,50X10,50X20,100X20

M.S. SQUARE (in inches) :- ½"X ½" , ¾"X ¾", 1" X 1", **16X16 mm**

M.S. ANGLES (in mm) :- 25X25X3, 50X50X6

M . S. CHANNEL (in mm):-75X35

M.S. SHEET/PLATE(Thickness in mm) :- 3M, 6, 8, 10, 12, 15

G.I. PIPE OD (ID) in mm :- 34(**27**), 49(**41**), 114(**105**), 88(**78**), 162(**150**)

G. I. SHEET :- 22 gauge

GEAR CUTTER (Module) :- 1,1.25,1.5, 2, 2.5, 5

NUT, BOLT & WASHER: - 1/8 , 3/16" , ¼" , 5/16"

(1 inch=25.6 mm)

Hand tools and measuring equipment you might encounter in the lab

Fitting Shop Equipment

BENCH VICE

HAND VICE

PIPE VICE

File

File Card

Hammer

Hack Saw

Pitches of Hack Saw

Hand File

Flat File

Half-round File

Round File

Square File

Three Square File

Fitting Shop Equipment

Twist Drill

Counter bore Drill

Countersink Drill

Hand Reamer

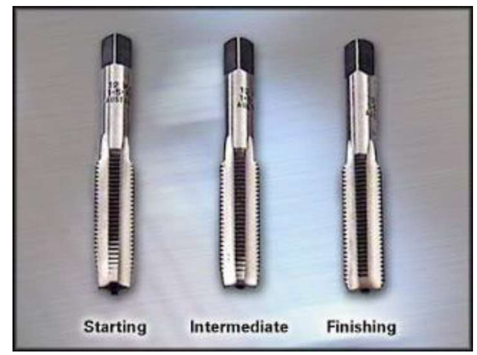
Machine Reamer

Adjustable Reamer

Chisels

Die

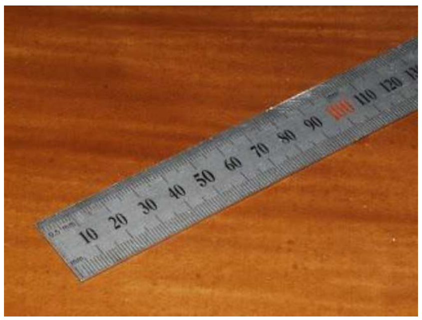
Die Nut



Taps

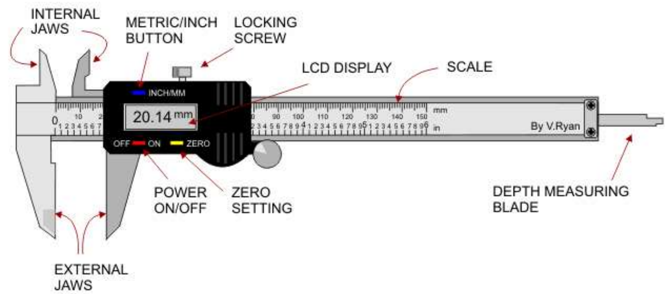
Tap Wrench

Measuring Equipment

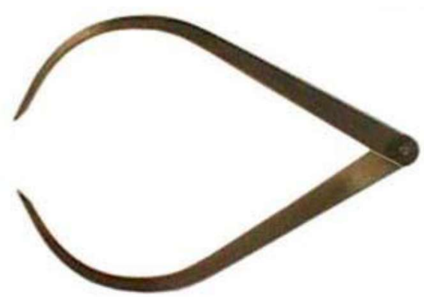


Ruler Scale

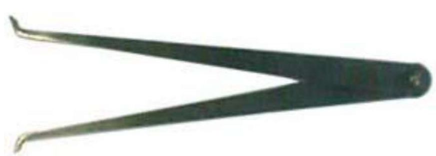
Dial Gauge



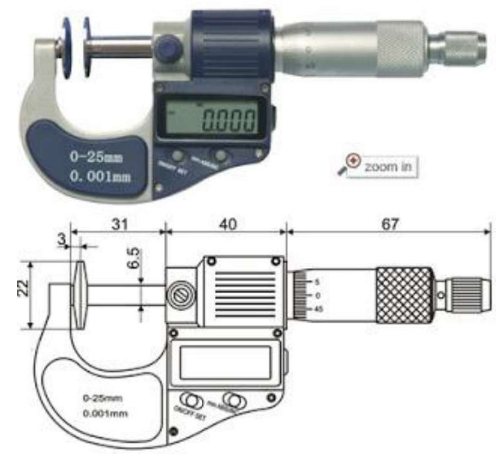
Vernier Caliper



Outside Caliper



Inside Caliper



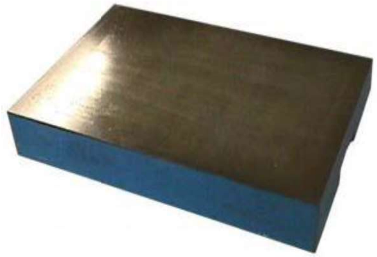
Outside Micrometer

Inside Micrometer

Depth Micrometer

Measuring & Marking Equipment

Divider



Surface Plate

Bevel Protector

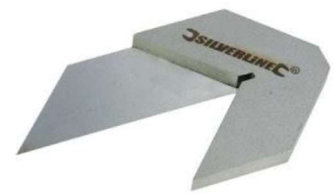
Scriber

Engineer's try Square

Combination Set

V-Block

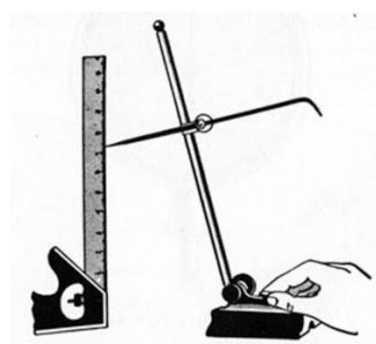
Wire Gauge



Center Square

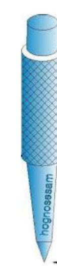
Striking Tools/Hammers

Angle Plate



Universal Marking Surface Gauge

Center Punch



Dot Punch

Part B

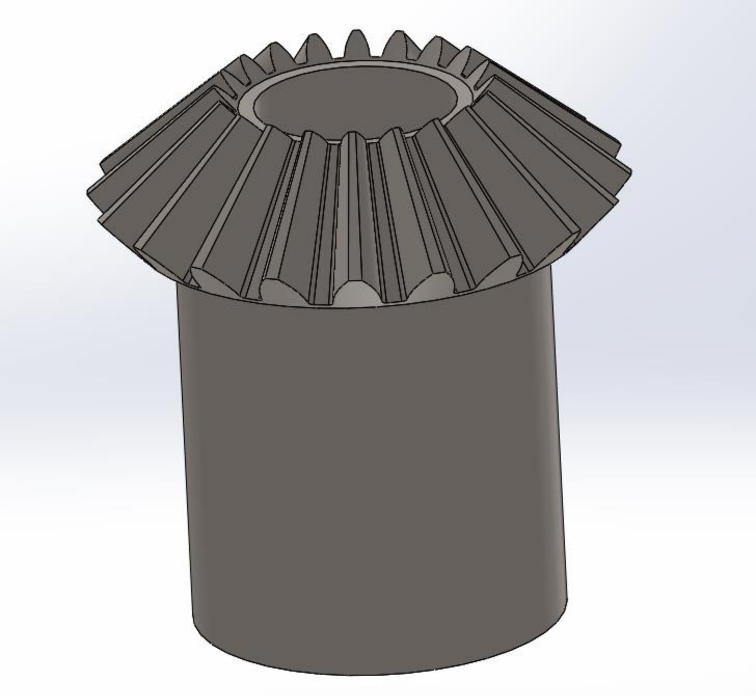
Lab exercise 1 and 2
Part A and B

Objective: to introduce machines and machining processes to you by having you make/observe how to make a bevel gear and a L-clamp. These parts are representative of things you might use in your projects.

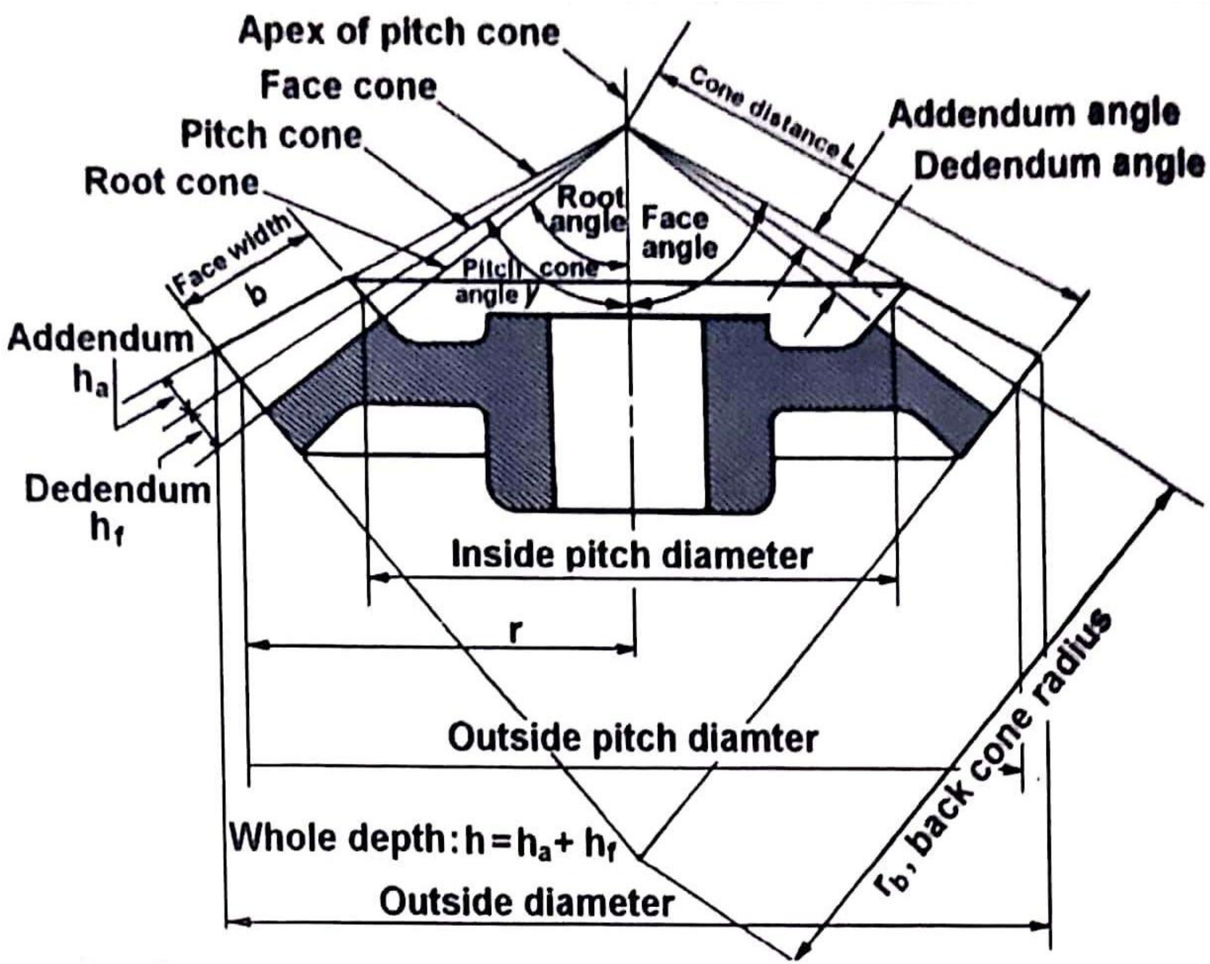
Structure of the lab: half of you will conduct one exercise in one lab turn and then the other in the next lab turn.

Part A: Making a spur/bevel gear.

This section first describes the procedure to make a bevel gear, followed by description of making a spur gear.



Nomenclature of bevel gear

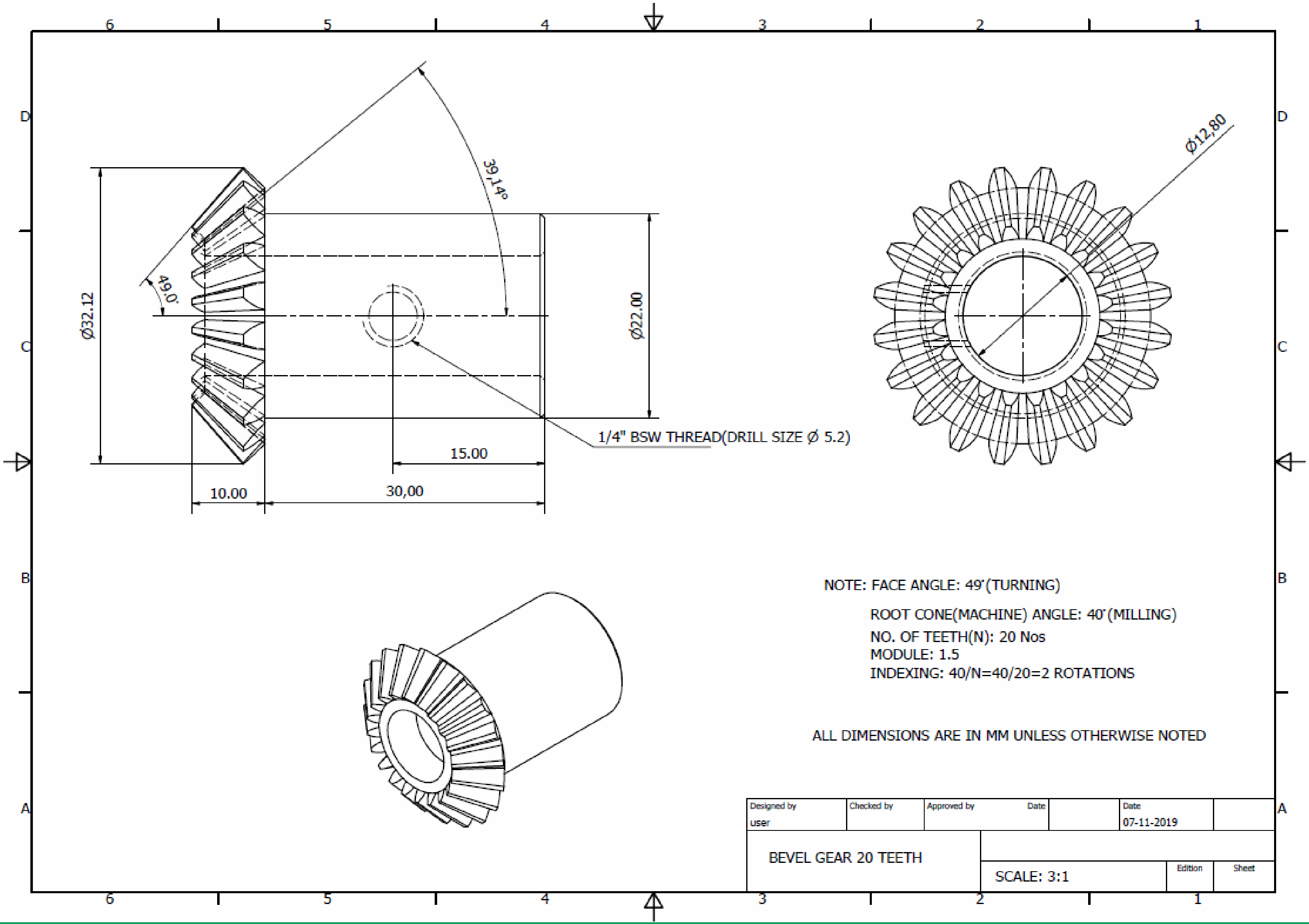


Calculation of standard straight bevel gear

1. Shaft angle, Σ
2. Module, m
3. Pressure angle, α
4. Number of teeth, $z_1 \; and \; z_2$
5. Pitch diameter, $d = z\,m$
6. Pitch cone angle, $\delta_1 = \tan^{-1} \frac{\sin \Sigma}{\frac{z_1}{z_2} + \cos \Sigma}$,

   $\delta_2 = \Sigma - \delta_1$
7. Cone distance, $R_e = \frac{d_2}{2 \sin \delta_2}$
8. Face width, $b \leq \frac{R_e}{3} \, or \; 10m$
9. Addendum, $h_a = m$
10. Dedendum, $h_f = 1.25\,m$
11. Dedendum angle, $\theta_f = \tan^{-1}\left(\frac{h_f}{R_e}\right)$
12. Addendum angle, $\theta_a = \tan^{-1}\left(\frac{h_a}{R_e}\right)$
13. Outer cone angle, $\delta_a = \delta + \theta_a$
14. Root cone angle, $\delta_f = \delta - \theta_f$
15. Outside diameter, $d_a = d + 2\,h_a \cos \delta$
16. Pitch apex to crown, $X = R_e \cos \delta - h_a \sin \delta$
17. Axial face width, $X_b = \frac{b \cos \delta\alpha}{\cos \theta\alpha}$
18. Inner outside diameter, $d_i = d_a - \frac{2b \sin \delta\alpha}{\cos \theta\alpha}$

Production drawing of a typical bevel gear



Steps in making a bevel gear

Step 1: Identify the raw material

Step 2: Mount the cylindrical work piece in the chuck

Step 3: Measure the diameter and turn it to size

Step 4: Drill a through hole

Step 5: Move to the milling machine

Step 6: Mount the gear blank in indexing head

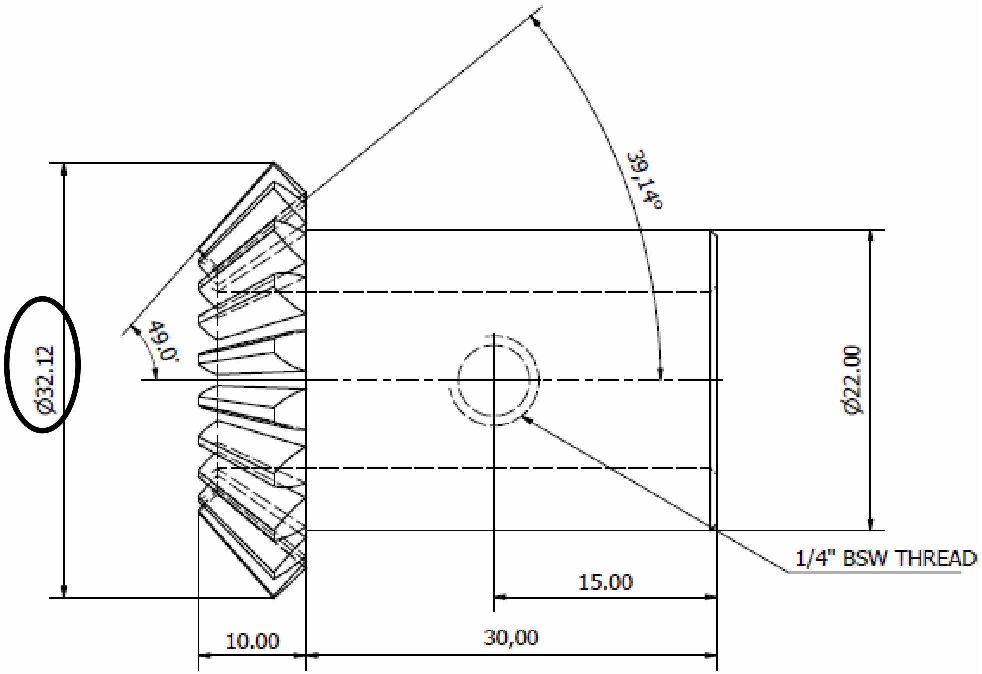
Step 7: Adjust the angle of the indexing head

Step 8: Mount the cutter on to arbor shaft

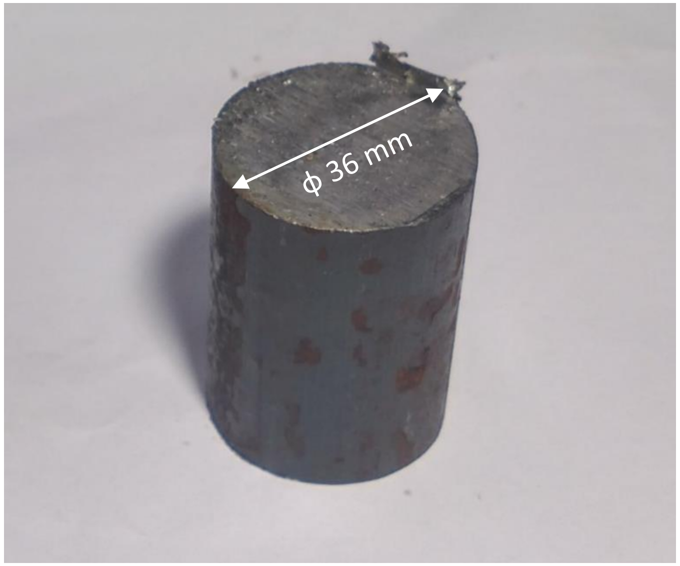
Step 9: Cut the gear teeth one at a time

… and, finally:

Step 1: Identify the raw material – a cylinder in this case



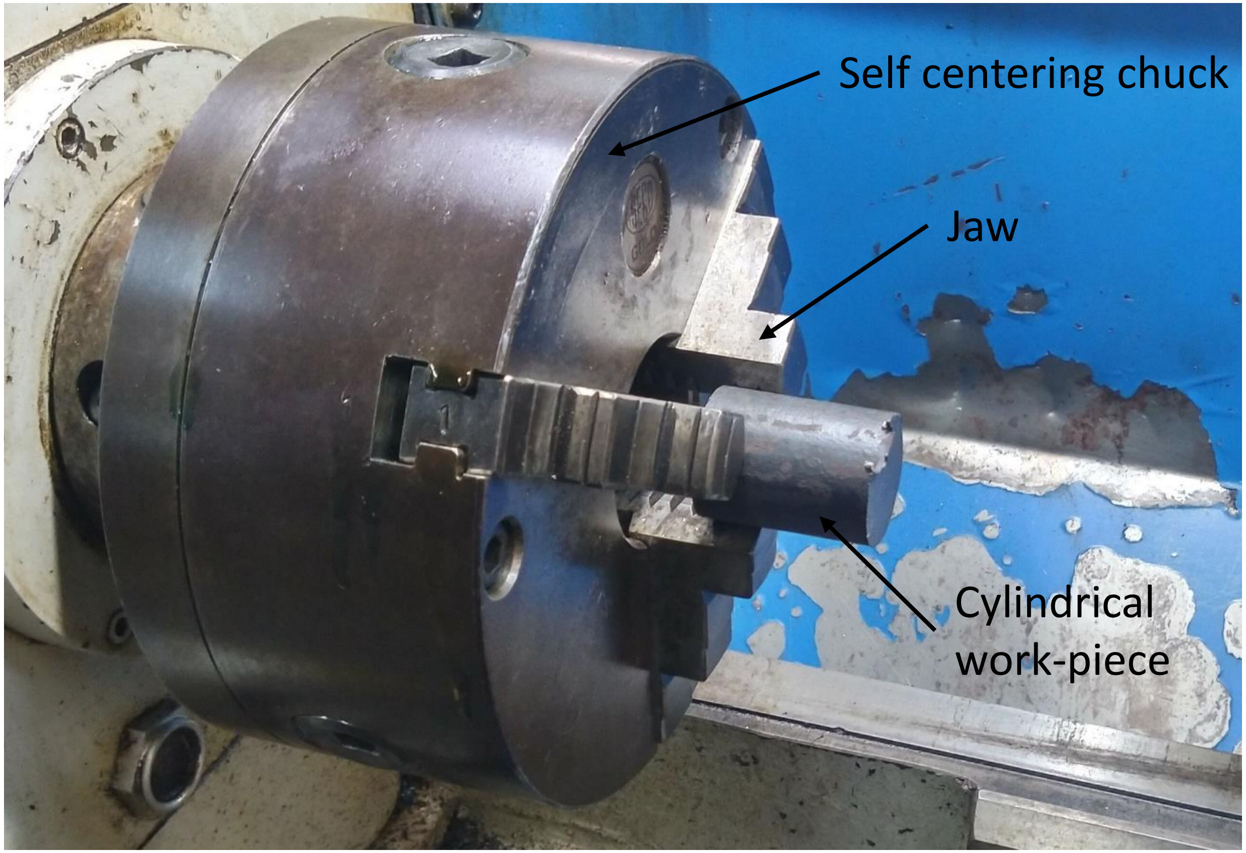
Drawing of the bevel gear



Select the diameter of the blank slightly more than the maximum diameter of the gear (please refer to the drawing)

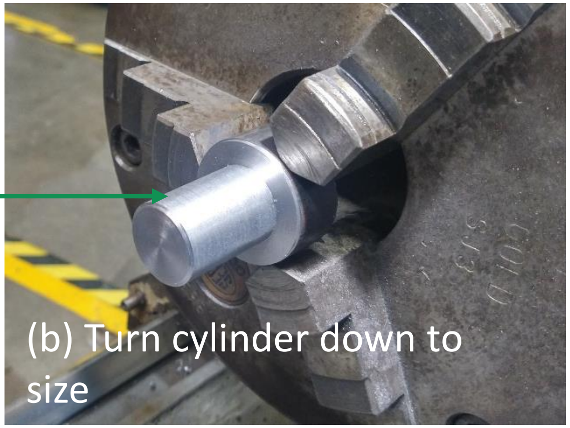
Step 1 continued: select a lathe machine

Step 2: mount the cylinder in the chuck

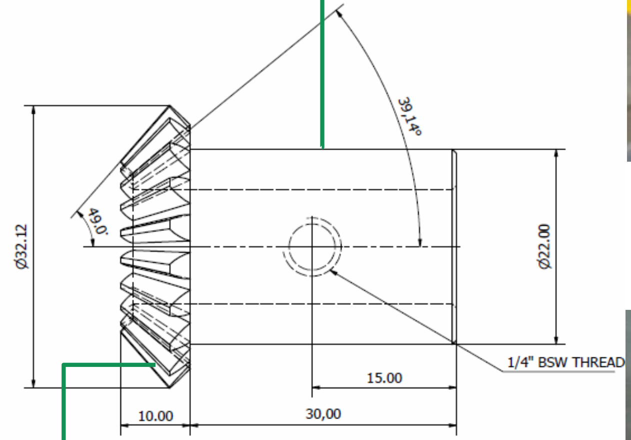


Step 3: Measure the diameter, and turn it to size

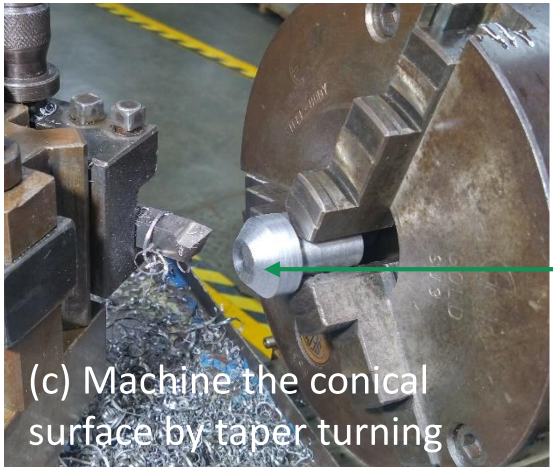
(a) Measure the diameter with a Vernier caliper



(b) Turn cylinder down to size



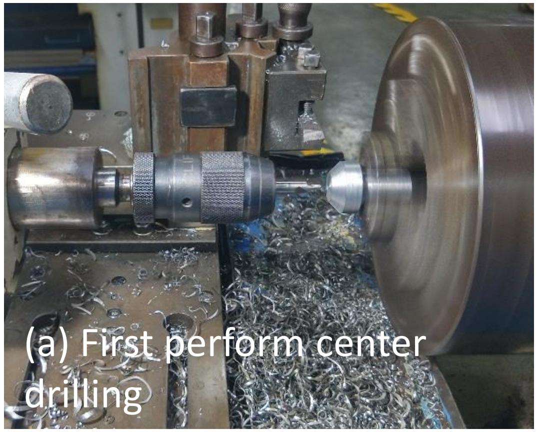
Drawing of the bevel gear

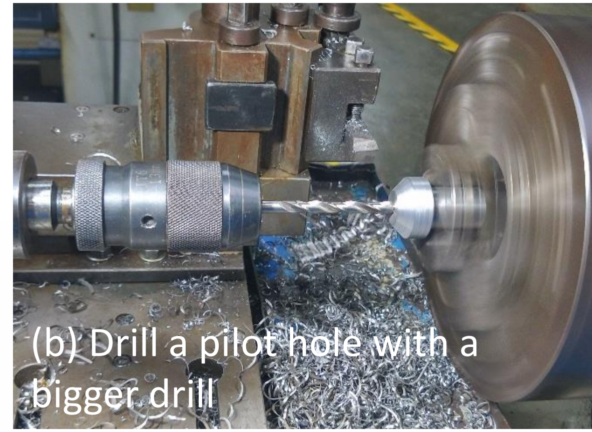


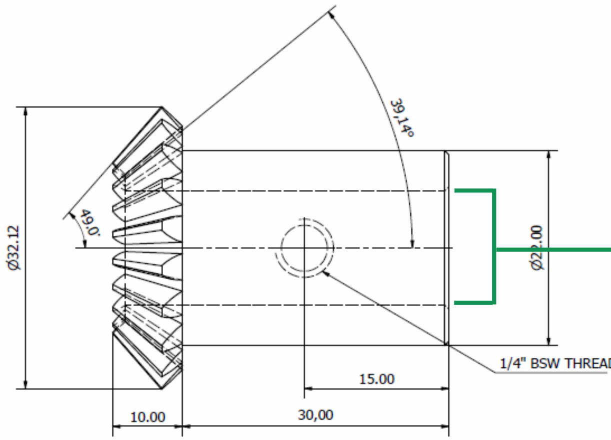
(c) Machine the conical surface by taper turning

(d) Gear blank

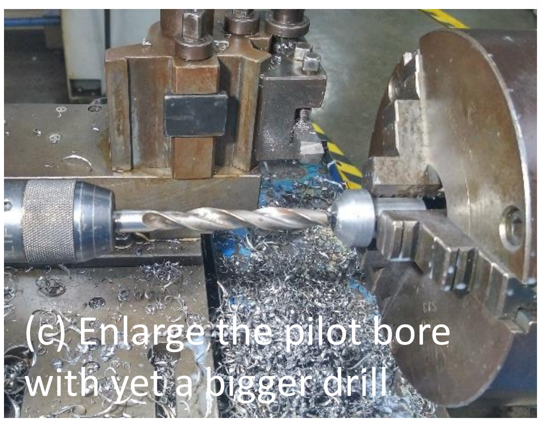
Step 4: Drill a through hole


(a) First perform center drilling


(b) Drill a pilot hole with a bigger drill



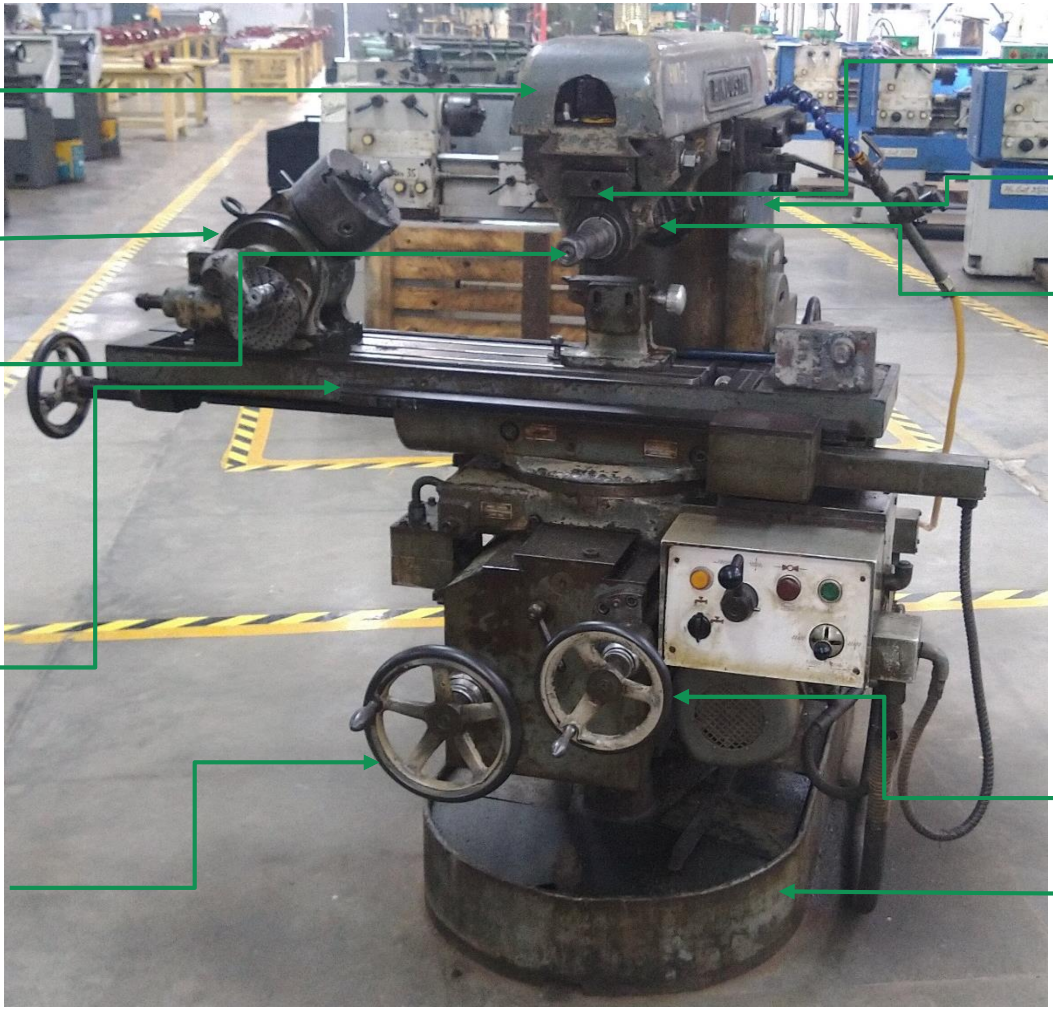
Drawing of the bevel gear


(c) Enlarge the pilot bore with yet a bigger drill

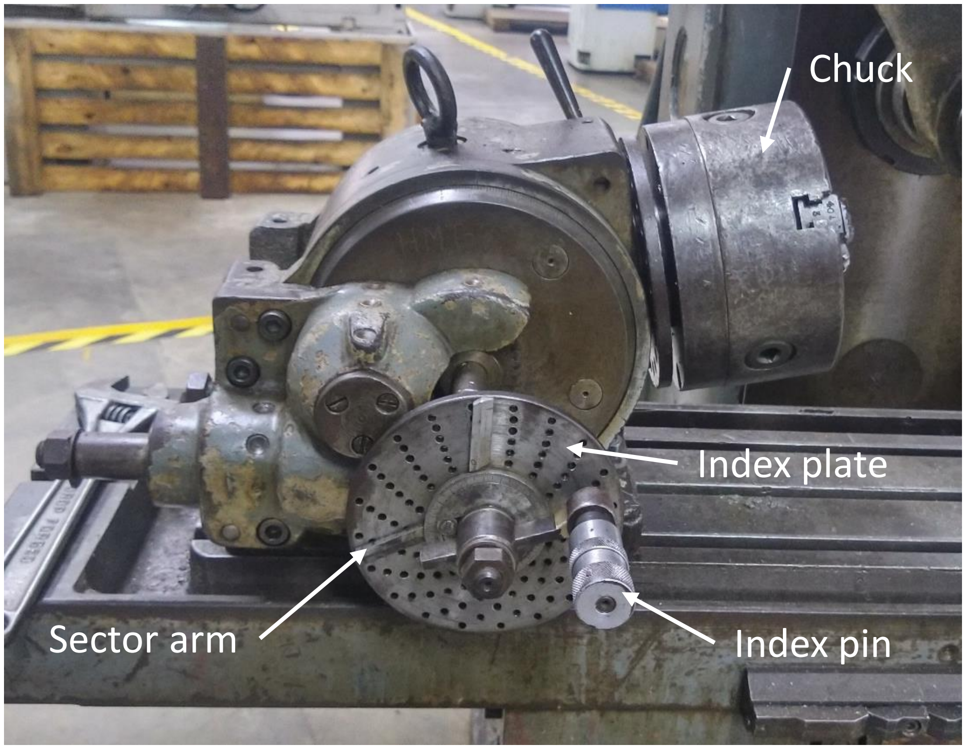

(d) Gear blank is ready to be loaded on the milling machine

All three drills are mounted in a drill chuck that is mounted in the tail stock of the machine

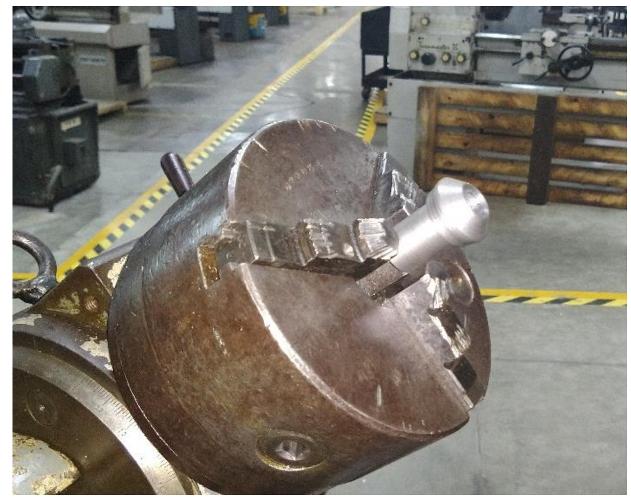
Step 5: Move to a milling machine



Step 6: Mount the gear blank on the Indexing head



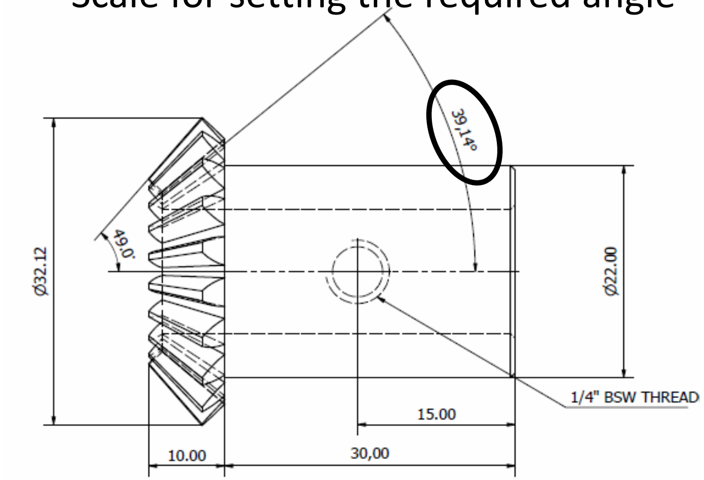
Indexing head



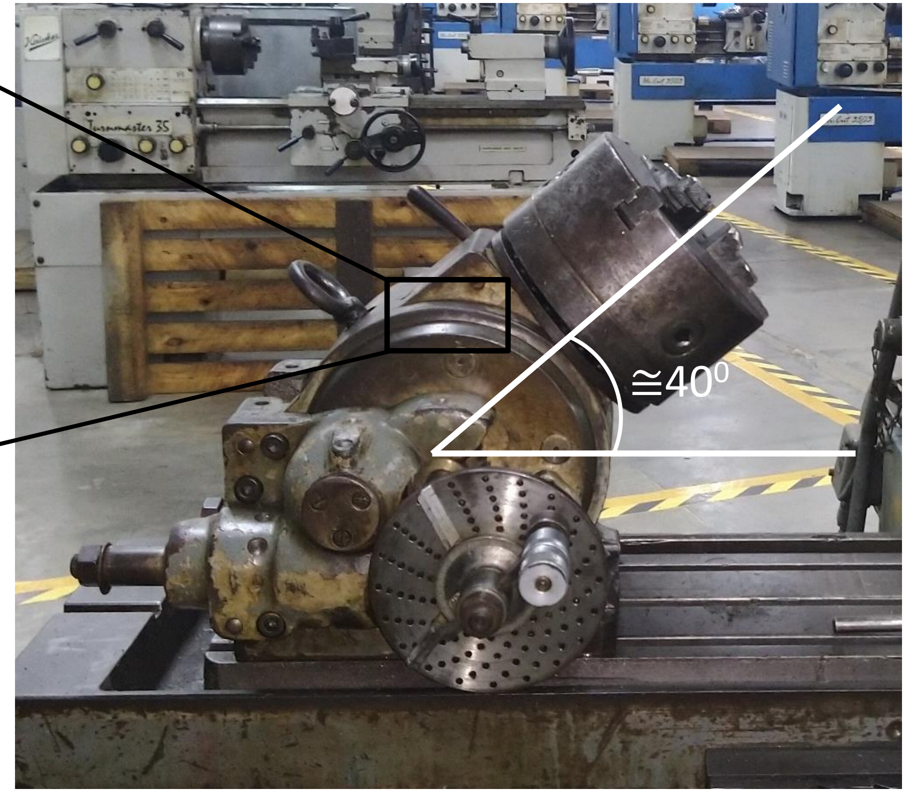
Gear blank mounted on chuck

Step 7: Adjust the angle of indexing head

Scale for setting the required angle

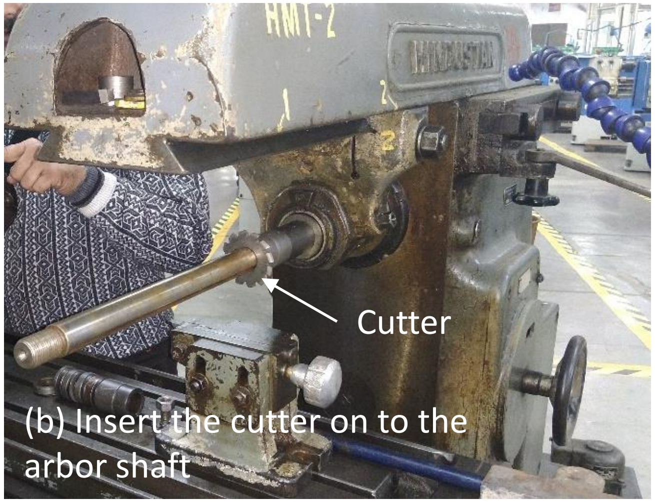


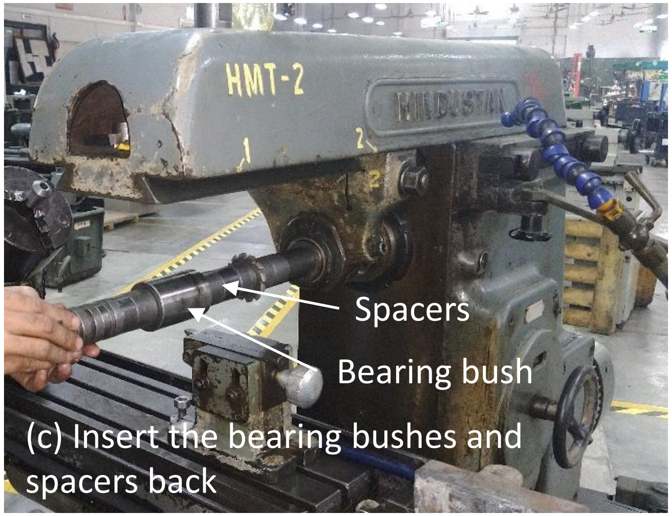
Drawing of the bevel gear



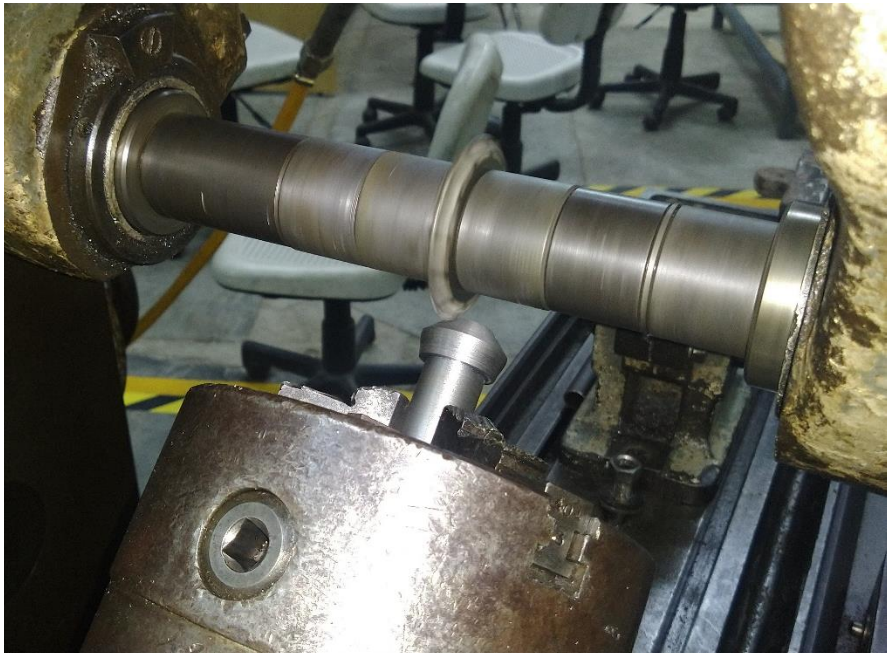
Indexing head inclined at ≅40^0

Step 8: Mount the cutter on the arbor shaft





Step 9: Cut the gear teeth one at a time

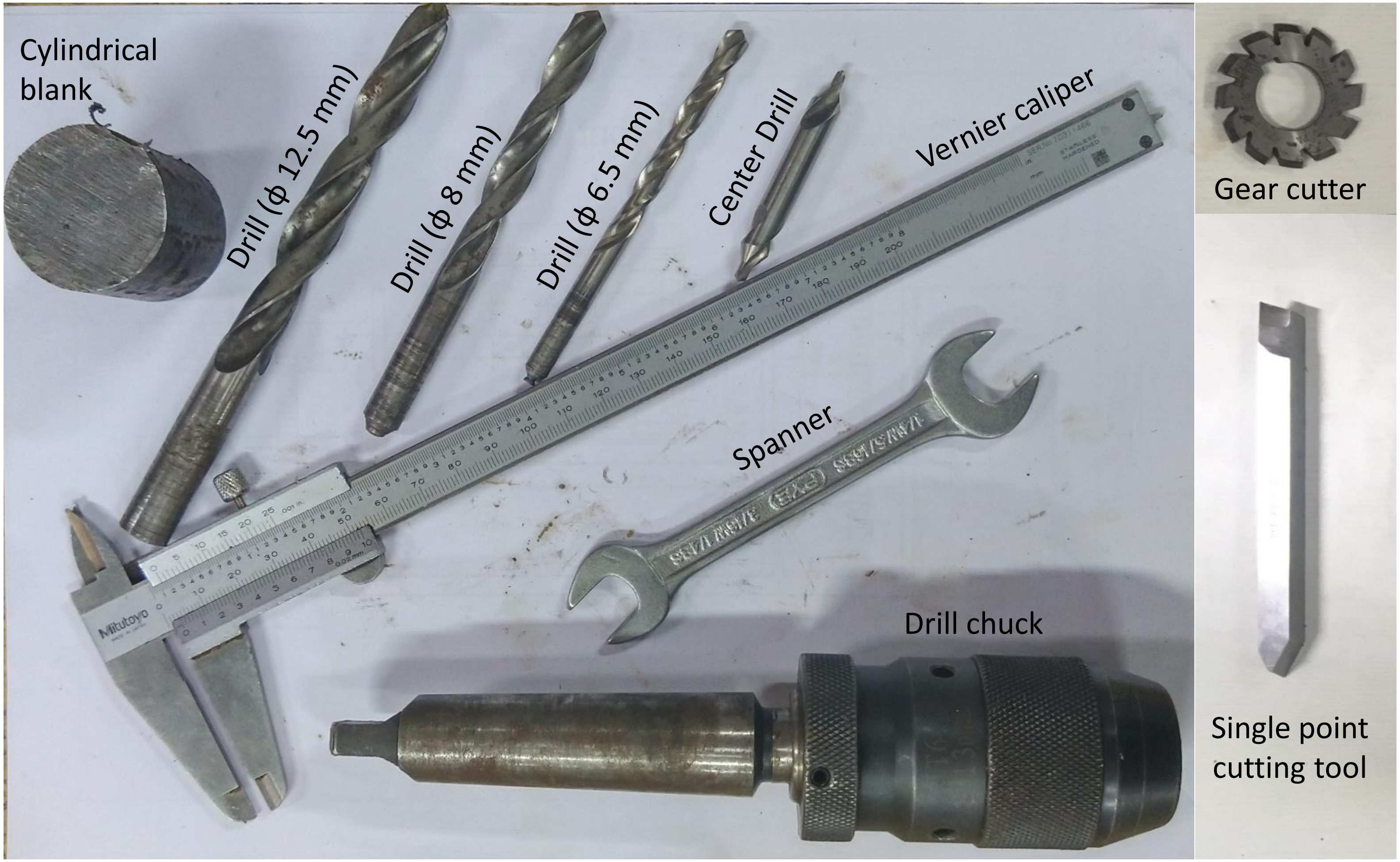


Index →

Index and cut all teeth similarly

Finally, Your bevel gear should look like this:

You will have used



Recounting all the steps in making a bevel gear

Step 1: Identify the raw material

Step 2: Mount the cylindrical work piece in the chuck

Step 3: Measure the diameter and turn it to size

Step 4: Drill a through hole

Step 5: Move to the milling machine

Step 6: Mount the gear blank in indexing head

Step 7: Adjust the angle of the indexing head

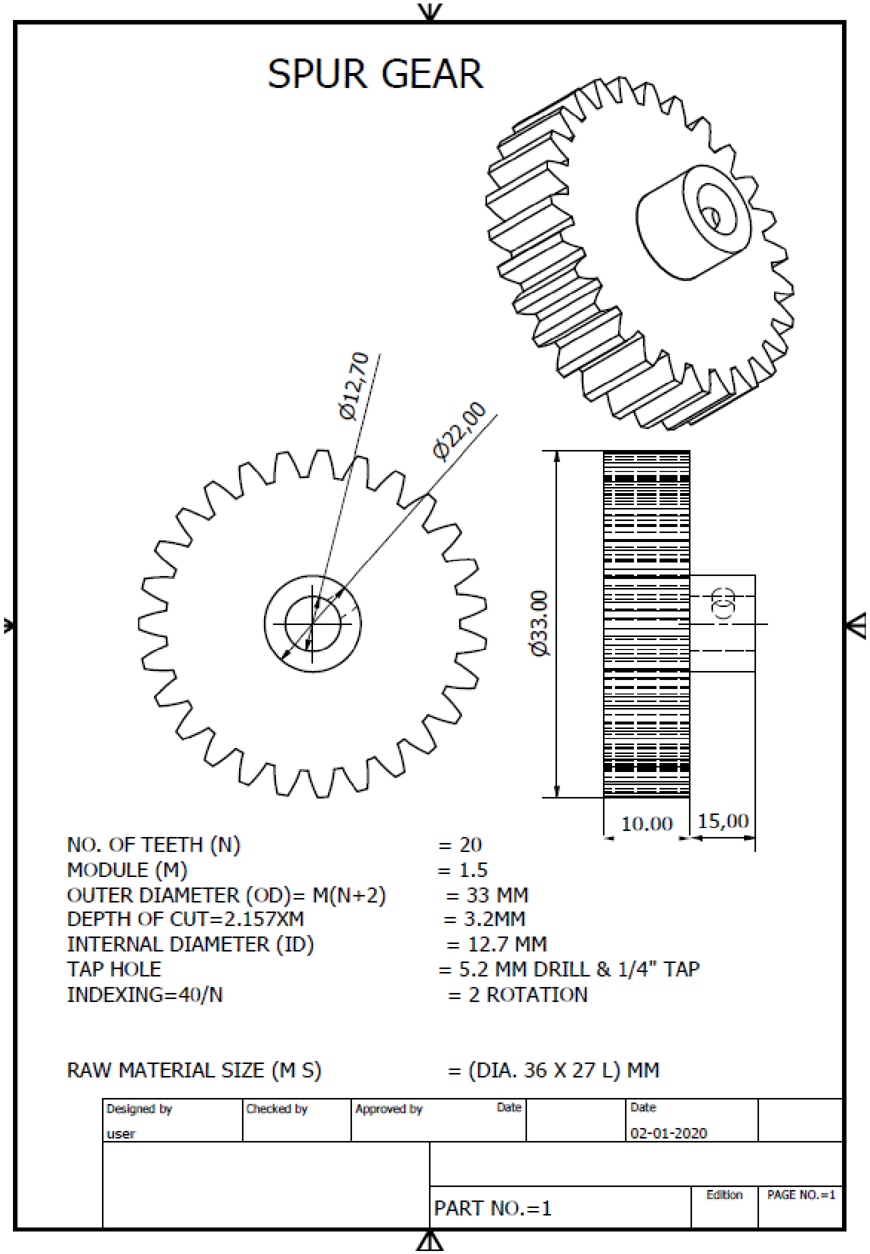
Step 8: Mount the cutter on to arbor shaft

Step 9: Cut the gear teeth one at a time

… and, finally:

Part A: Making a spur gear.
This section describes the procedure to make a spur gear.

Drawing of the gear



Steps in making a spur gear

Step 1: Identify the raw material

Step 2: Mount the cylinder in the lathe

Step 3: Measure the diameter, and turn it to size

Step 4: Drill a through hole

Step 5: Assemble the gear blank in a mandrel

Step 6: Mount the mandrel in the machine

Step 7: Turn the gear blank down to size

Step 8: Move to the milling machine

Step 9: Mount the mandrel and the cutter

Step 10: Mount the driving dog for indexing

Step 11: Cut the gear teeth one at a time

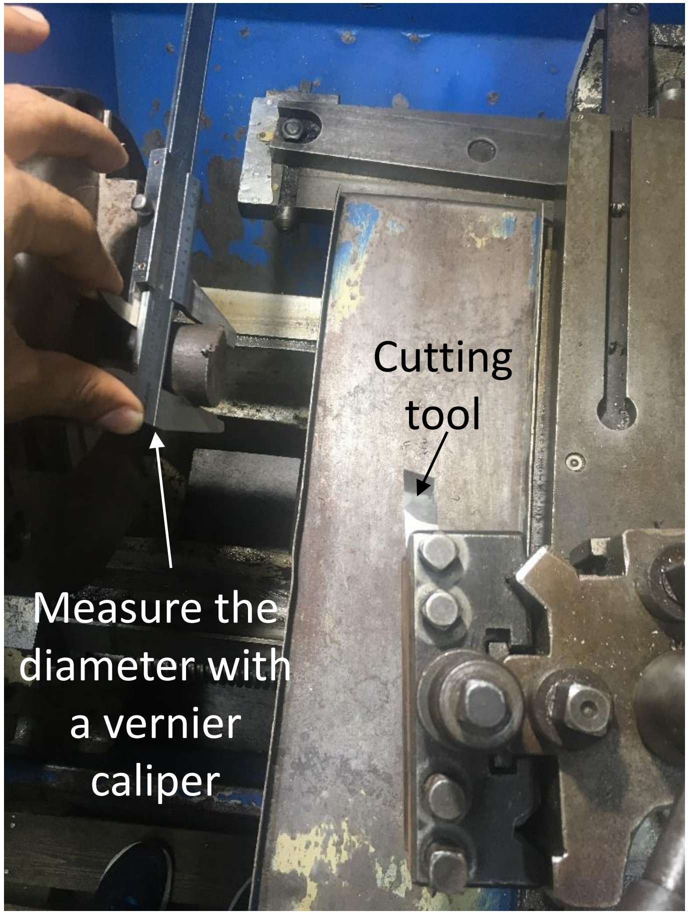
… and, finally:

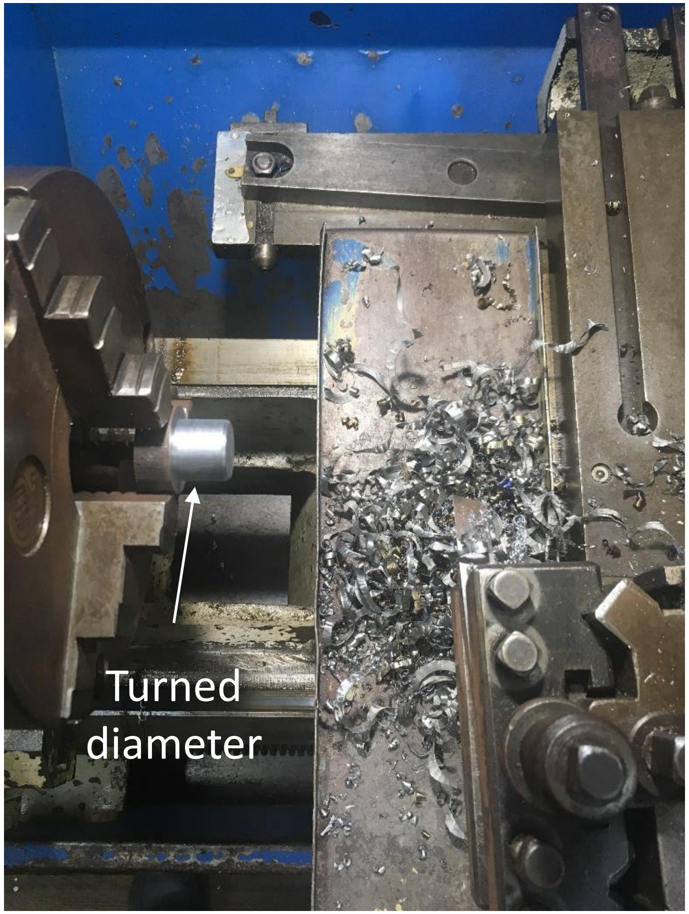
Step 1: Identify the raw material – a cylinder in this case

Then, the machine – a lathe

Step 2: Mount the cylinder in the lathe

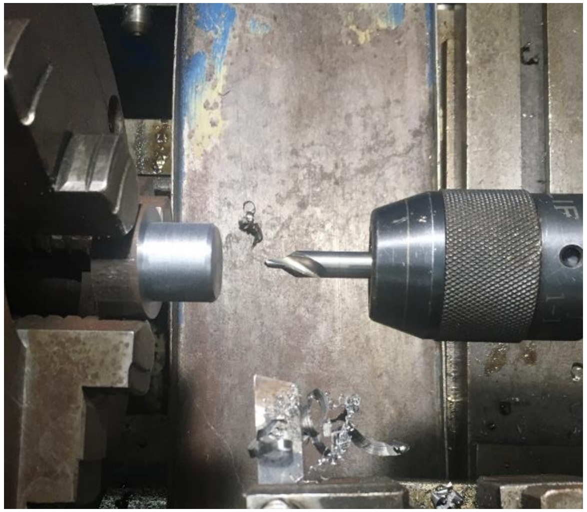
Step 3: Measure the diameter, and turn it to size



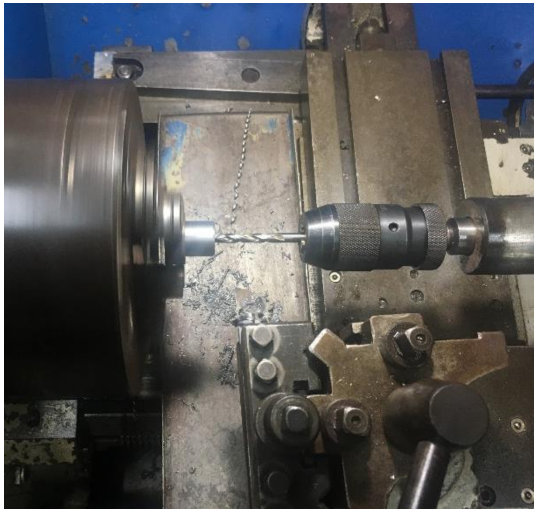


Step 4: Drill a through hole

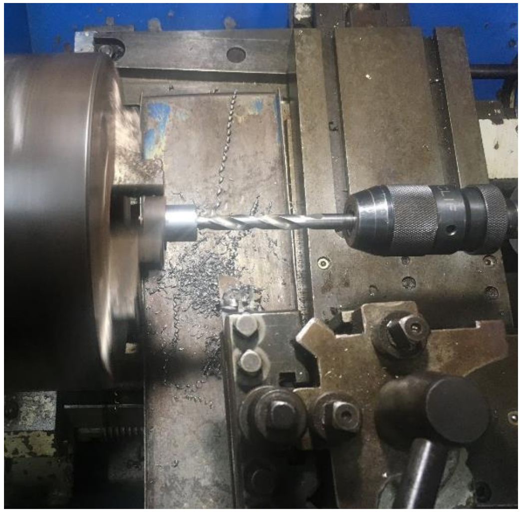
(i) Center drill



(ii) Drill a pilot hole

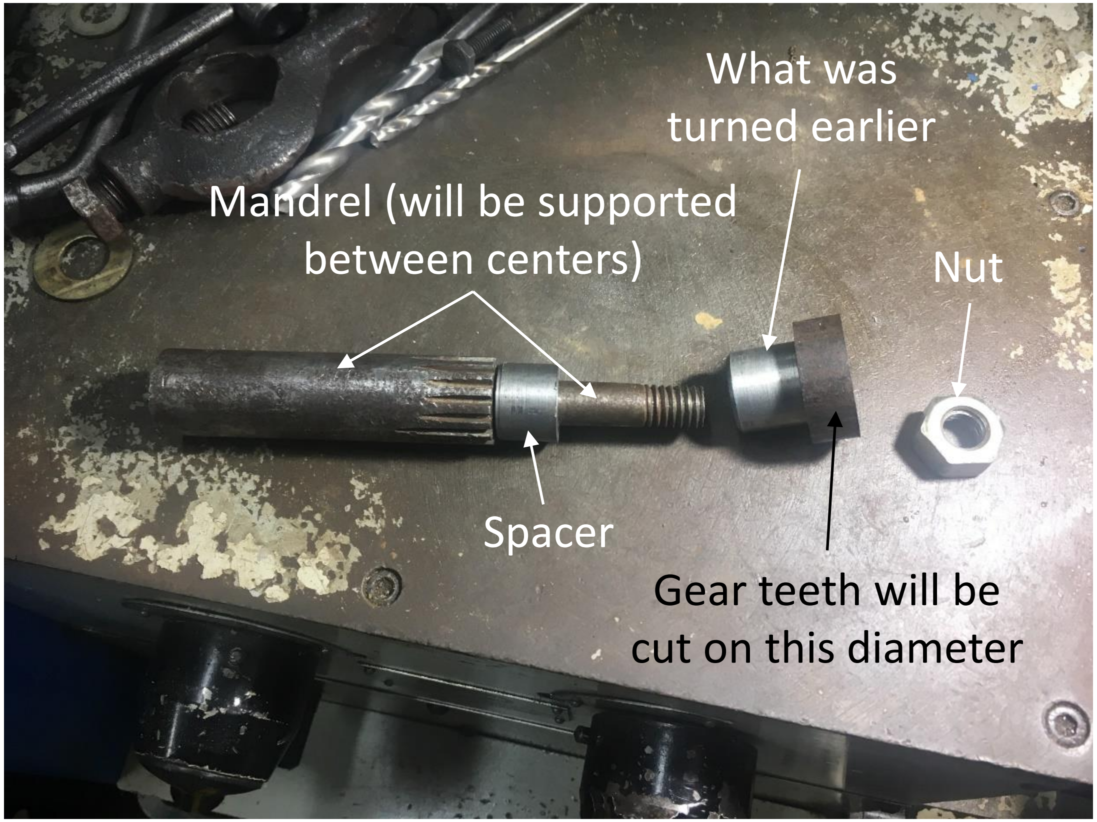


(iii) Enlarge the pilot bore with a bigger drill

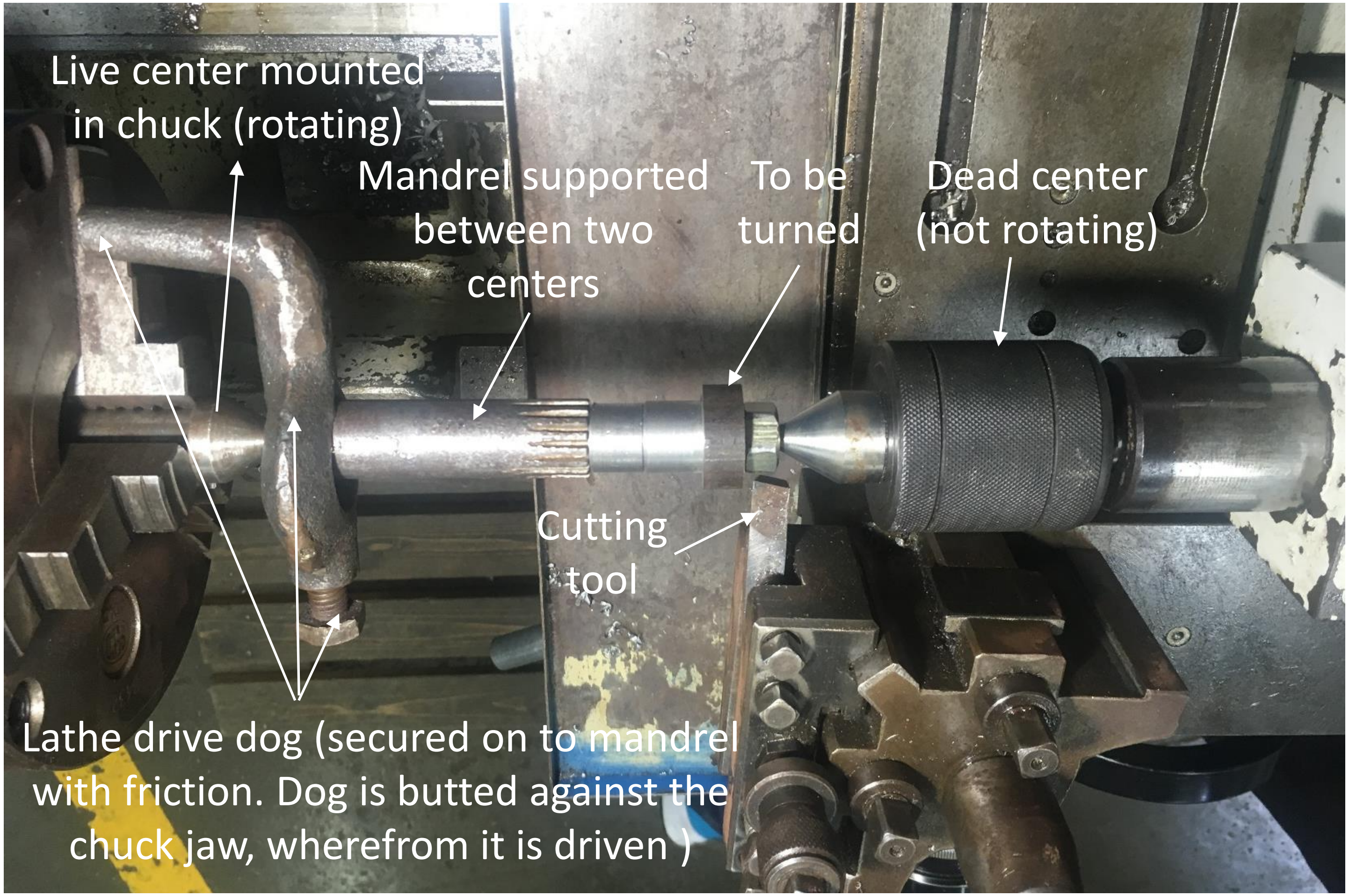


All three drills are mounted in a drill chuck that is in turn mounted in the tail stock of the machine

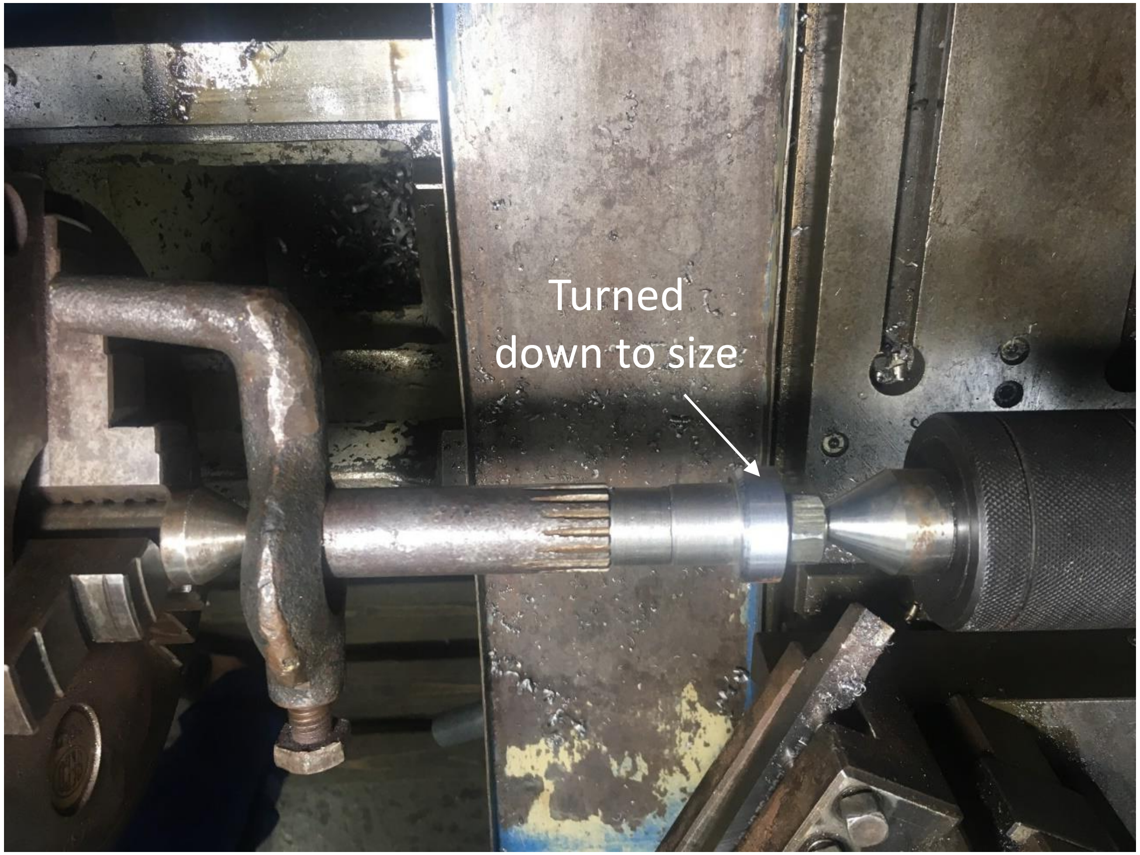
Step 5: Assemble the stepped gear blank in a mandrel



Step 6: Mount the mandrel in the machine

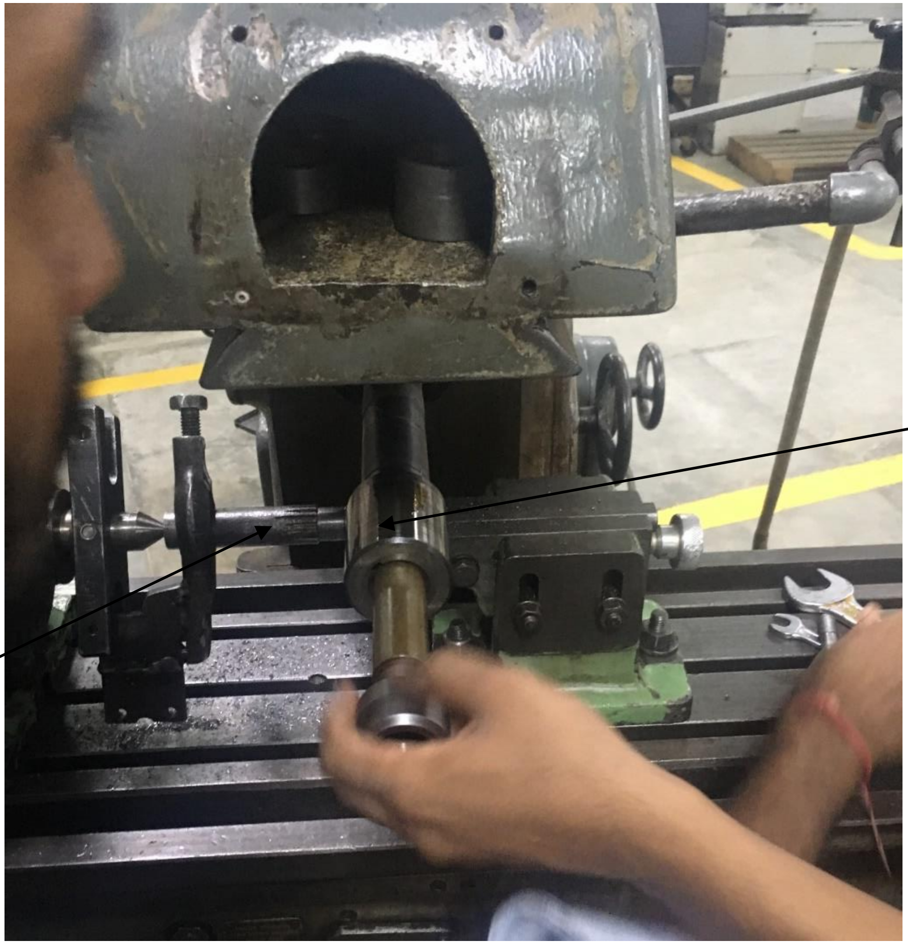


Step 7: Turn the gear blank down to size



Step 8: Move to the milling machine

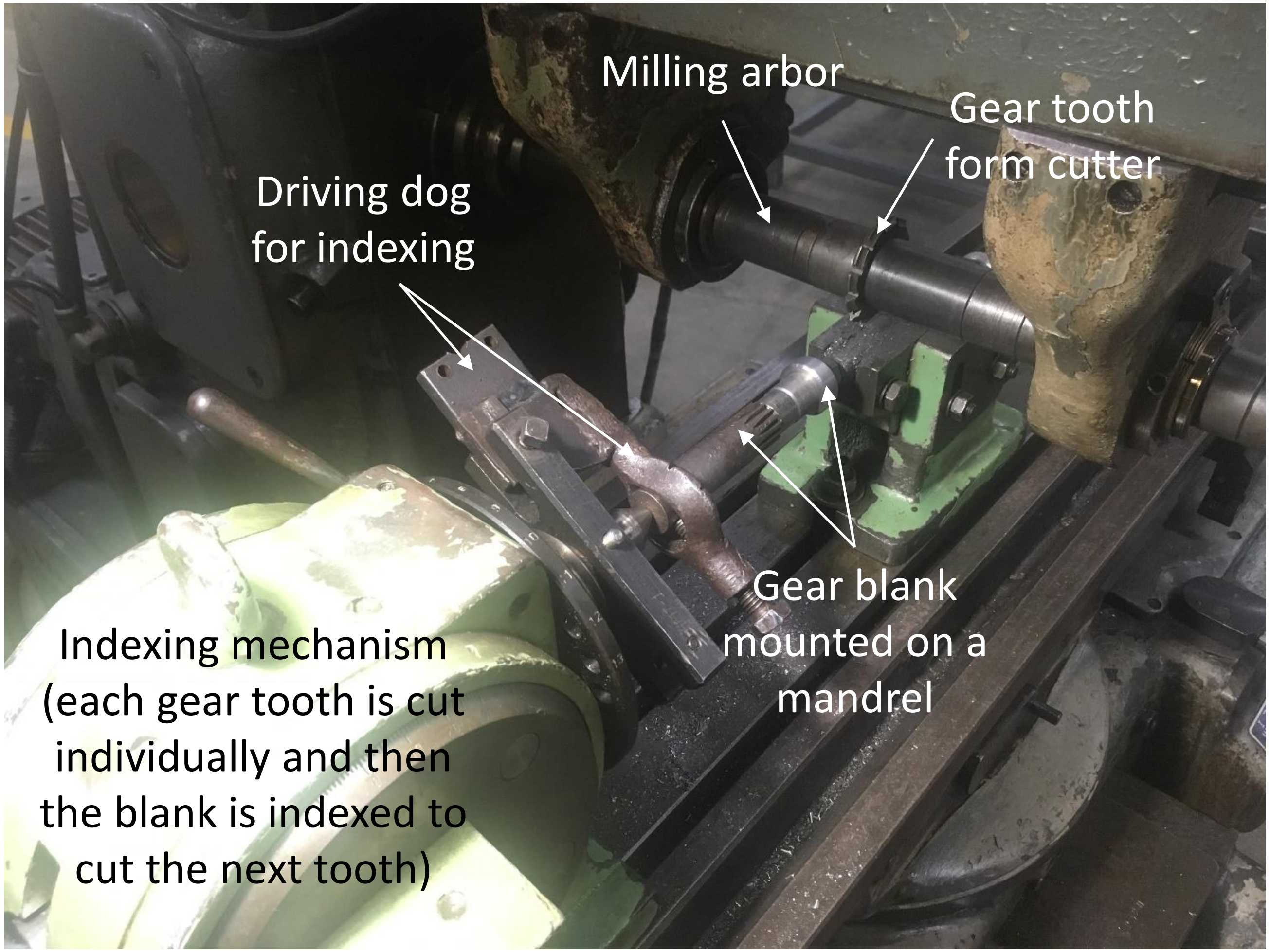
Step 9: Mount the mandrel in the milling machine between centres (like in the lathe) + mount the form cutter for cutting gear teeth



Mount the gear form cutter in the milling arbor (disassemble + assemble the arbor as necessary)

Gear blank mounted on the mandrel between centers

Step 10: Mount the driving dog for indexing the blank



Step 11: Cut the gear teeth one at a time

Index

Your spur gear should look like:

You will have used

Recounting all the steps in making a spur gear

Step 1: Identify the raw material

Step 2: Mount the cylinder in the lathe

Step 3: Measure the diameter, and turn it to size

Step 4: Drill a through hole

Step 5: Assemble the gear blank in a mandrel

Step 6: Mount the mandrel in the machine

Step 7: Turn the gear blank down to size

Step 8: Move to the milling machine

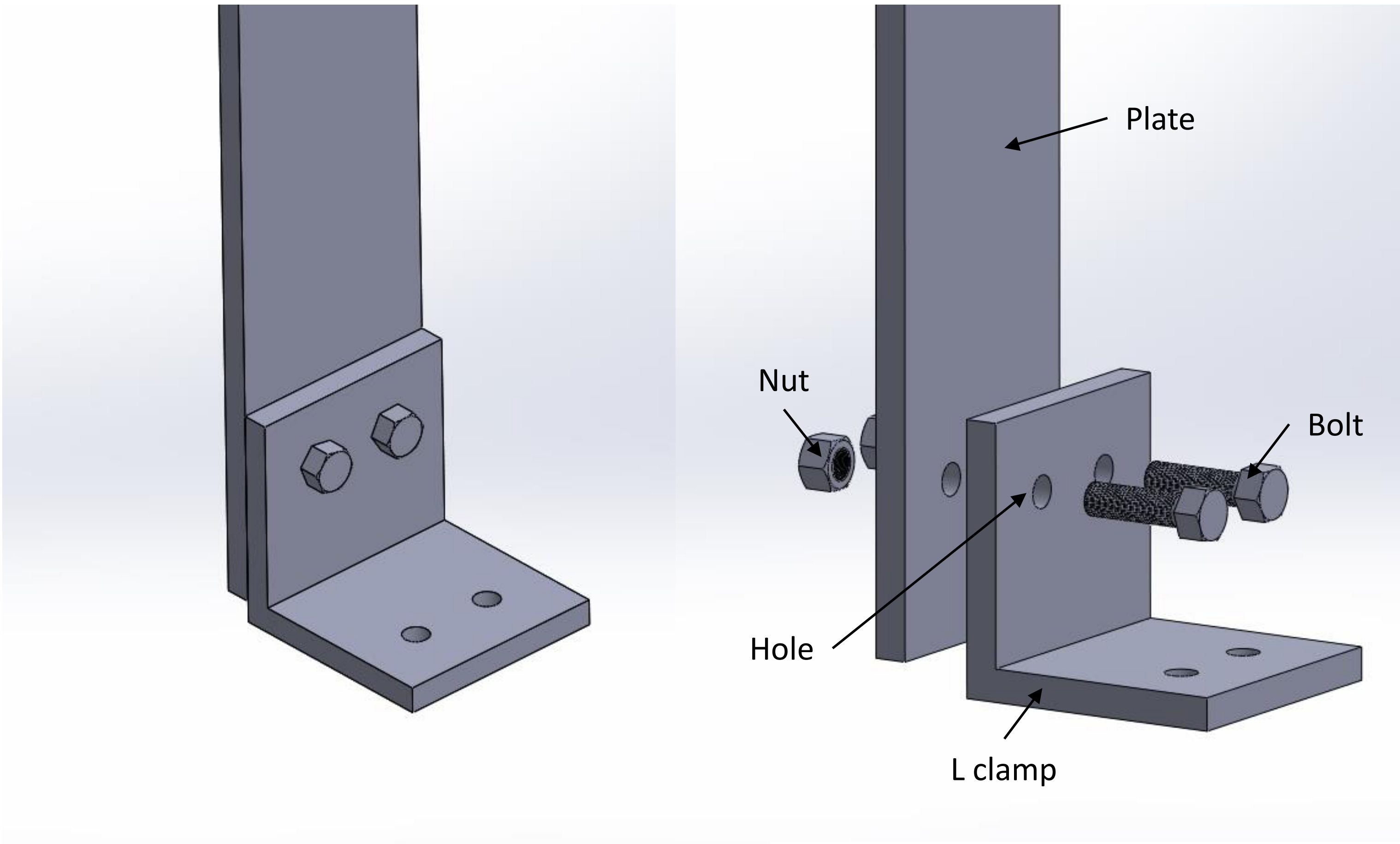
Step 9: Mount the mandrel and the cutter

Step 10: Mount the driving dog for indexing

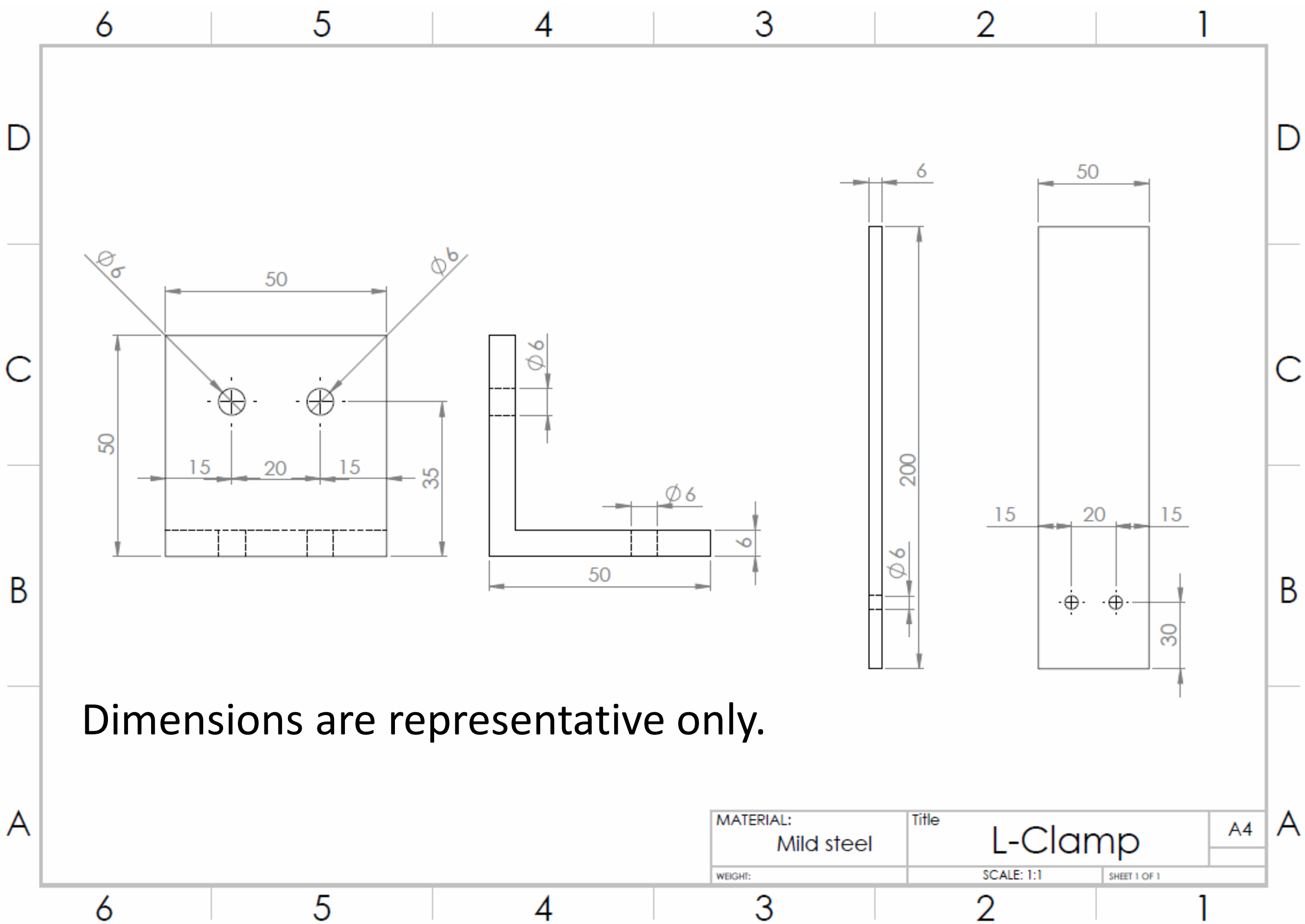
Step 11: Cut the gear teeth one at a time

… and, finally:

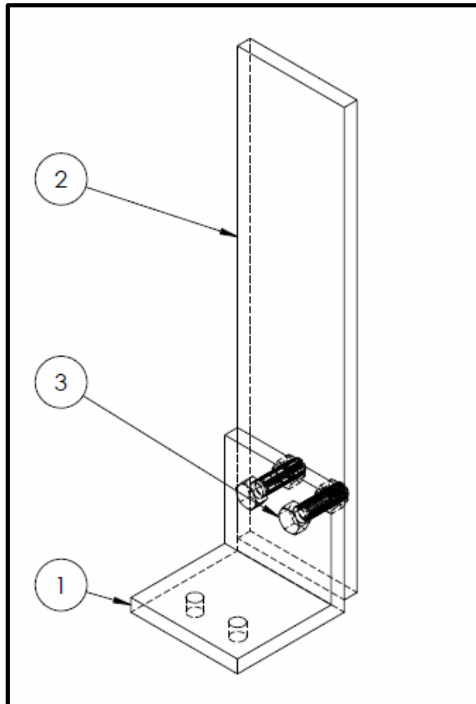
Part B: Making of a L-Clamp



Production drawing of L-Clamp



Dimensions are representative only.



ITEM NO.	DESCRIPTION	QTY.
1	L clamp	1
2	plate	1
3	Bolt (M6)	2
4	nut	2

Bill of material

Steps in making a L-Clamp

Step 1: Identify raw material

Step 2: Mark location for the hole

Step 3: Punch the marked location

Step 4: Move to drilling machine

Step 5: Mount drill chuck and drill

Step 6: Clamp the vice on the work table, and work-piece in the vice

Step 6: Adjust the table position

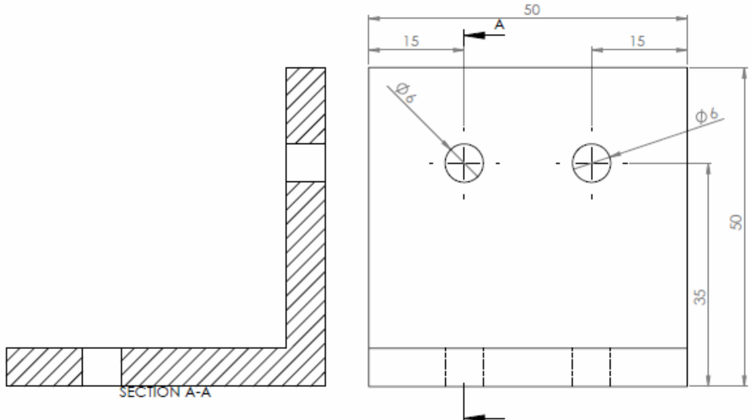
Step 7: Drill holes one at a time

Step 8: Assemble parts

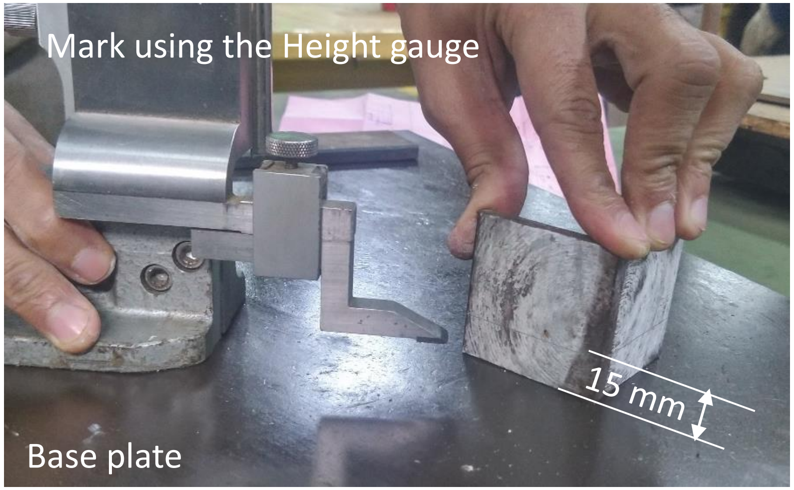
… and, finally:

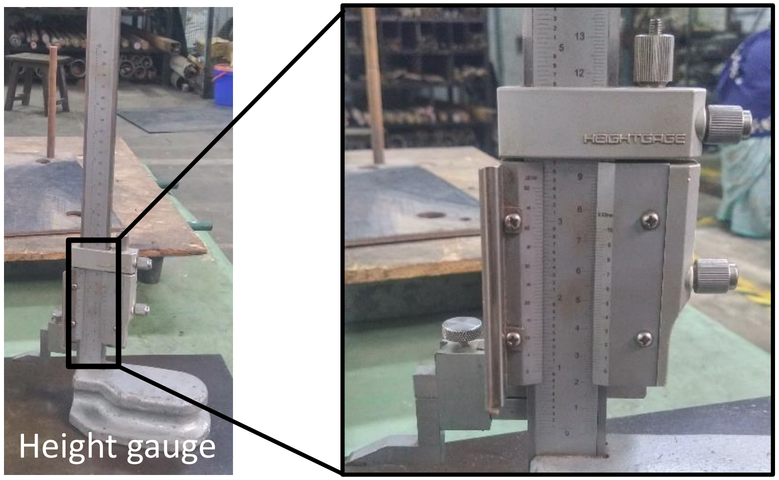
Step 1: Identify raw material

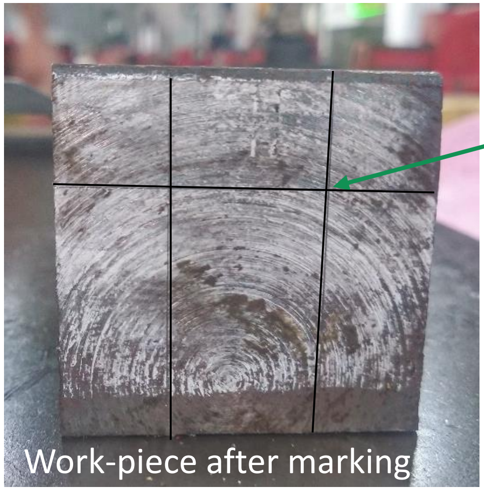
Step 2: Mark location for the hole



Drawing of L Clamp



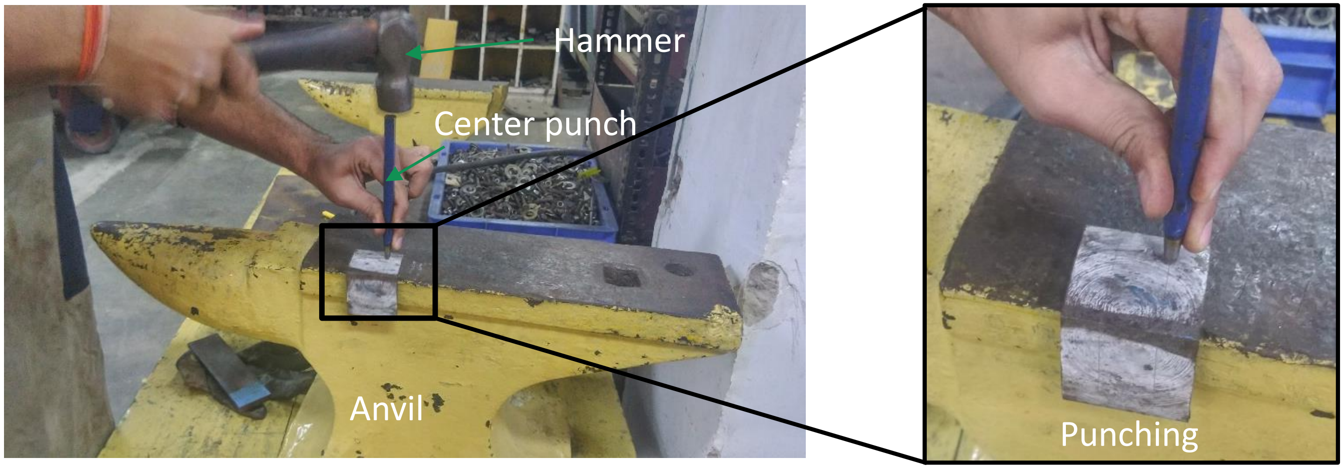


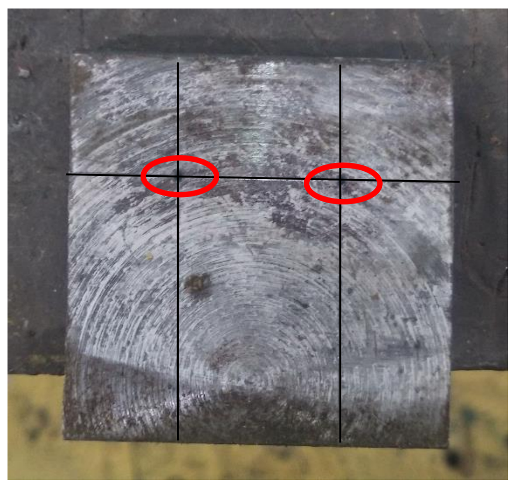


Location of the hole

Note: Similar procedure should be followed for plate

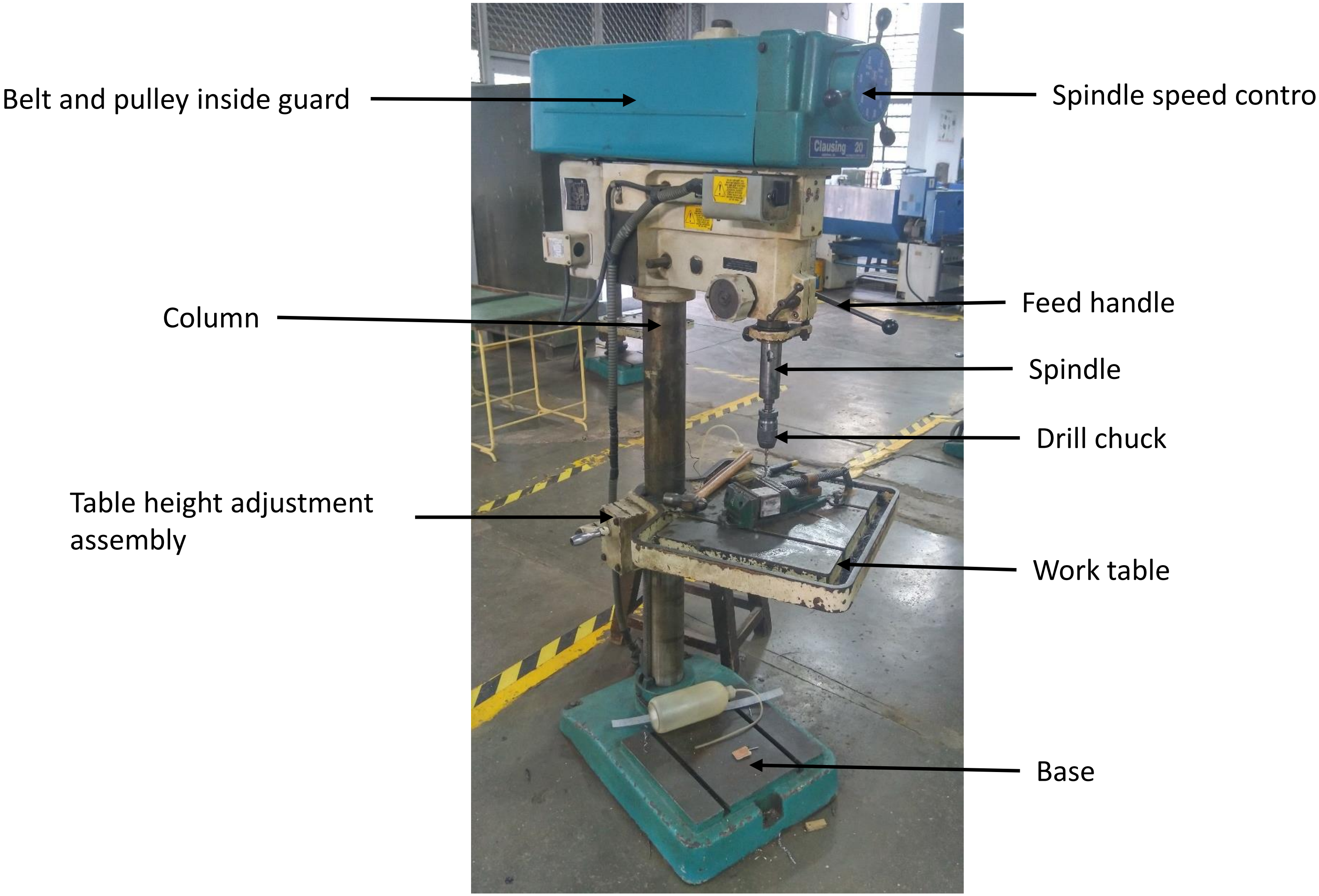
Step 3: Punching marked location





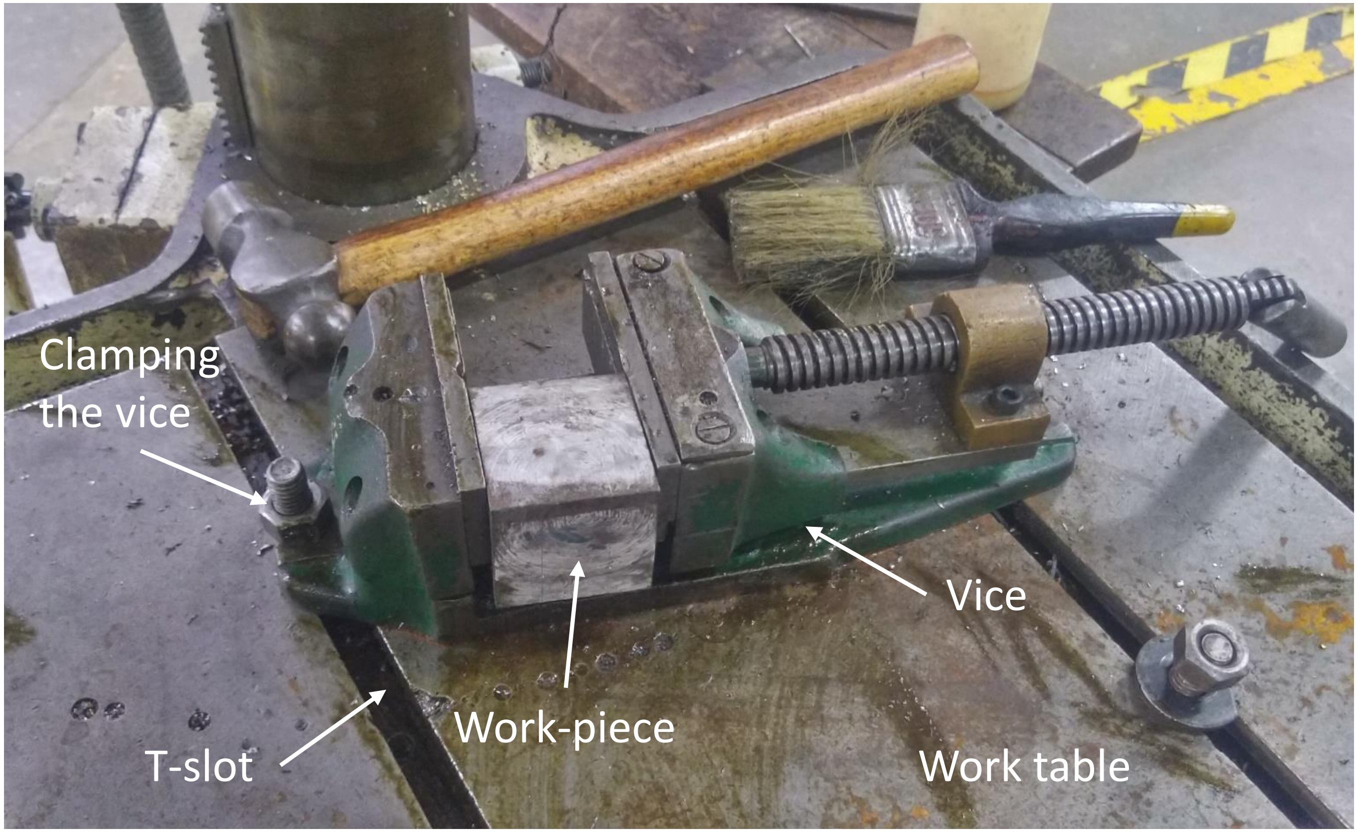
Work-piece after punching

Step 4: Move to a drilling machine

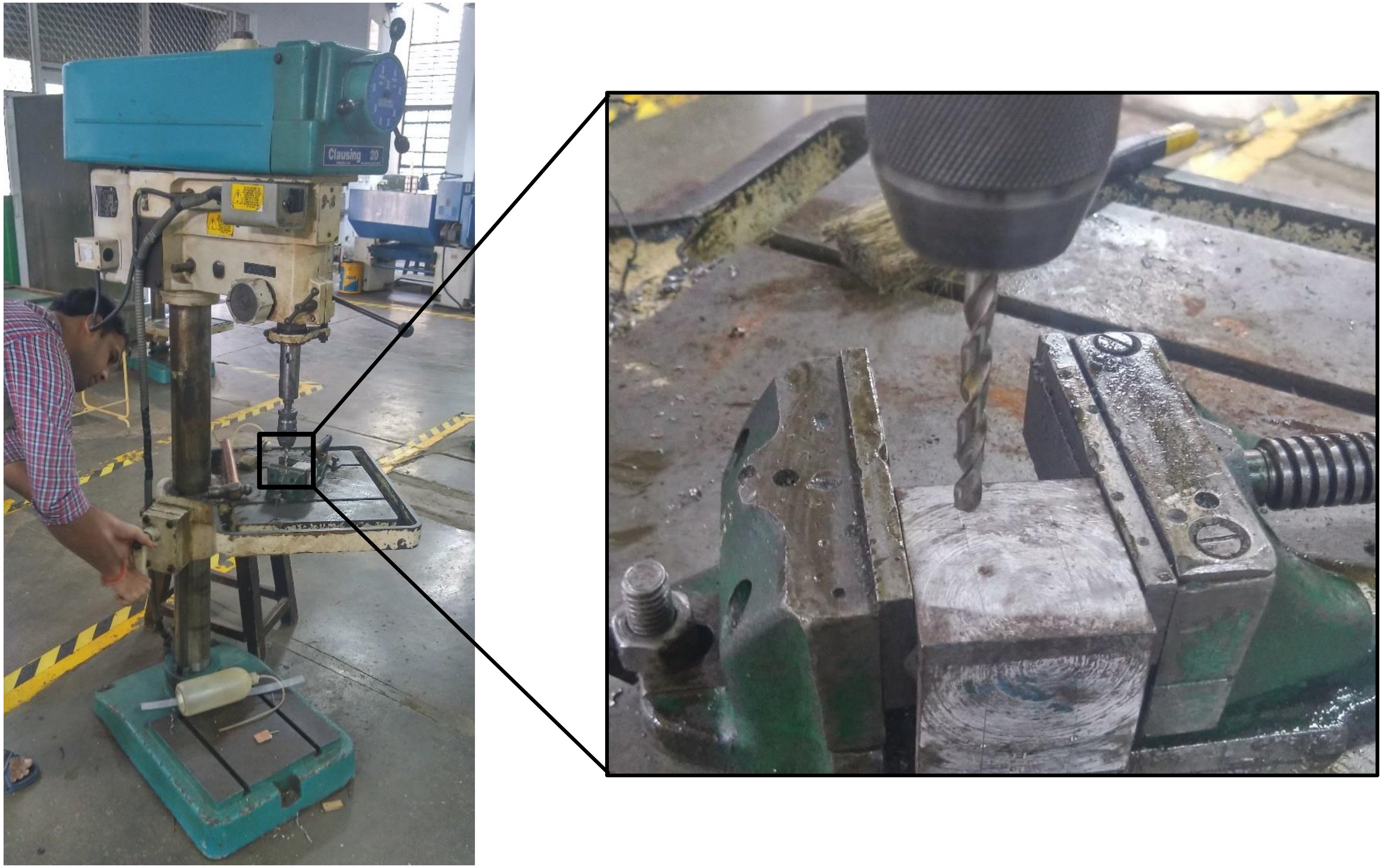


Step 6: Mount drill chuck and drill

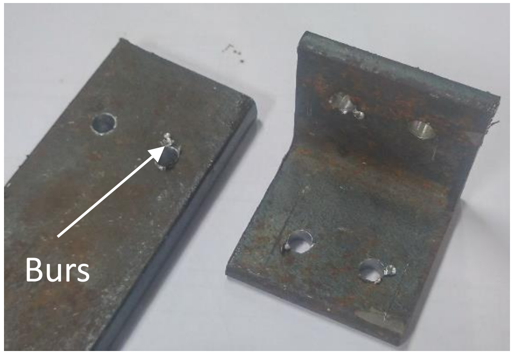
Step 5: Clamp the vice on the work table, and the work-piece in the vice

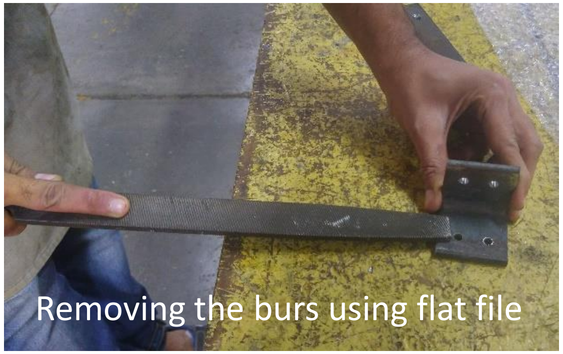


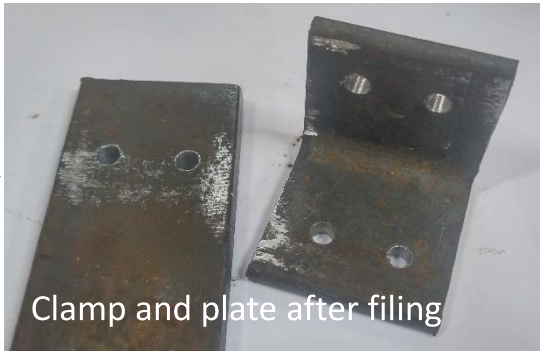
Step 6: Adjust the table position



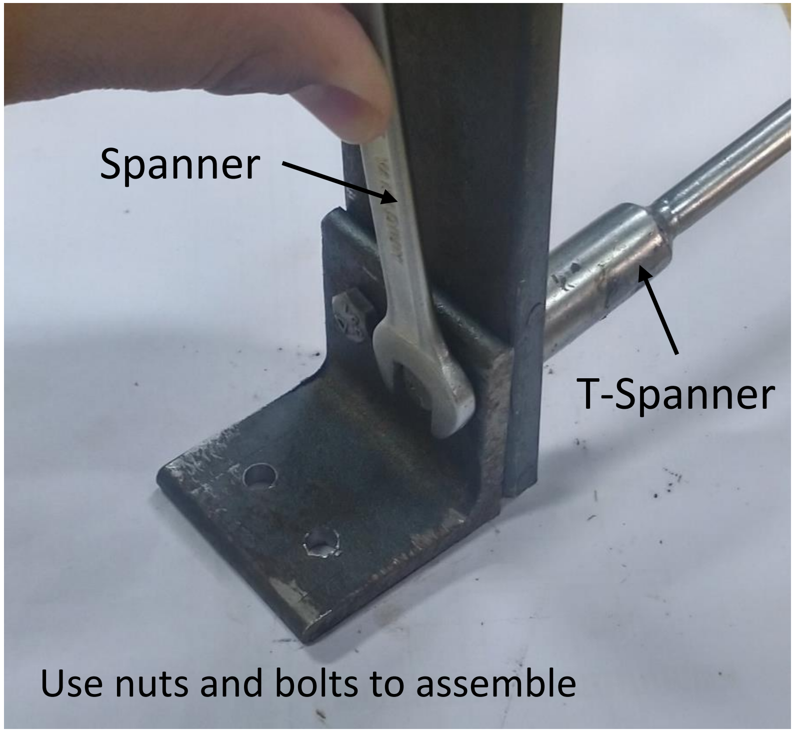
Step 7: Drill holes one at a time



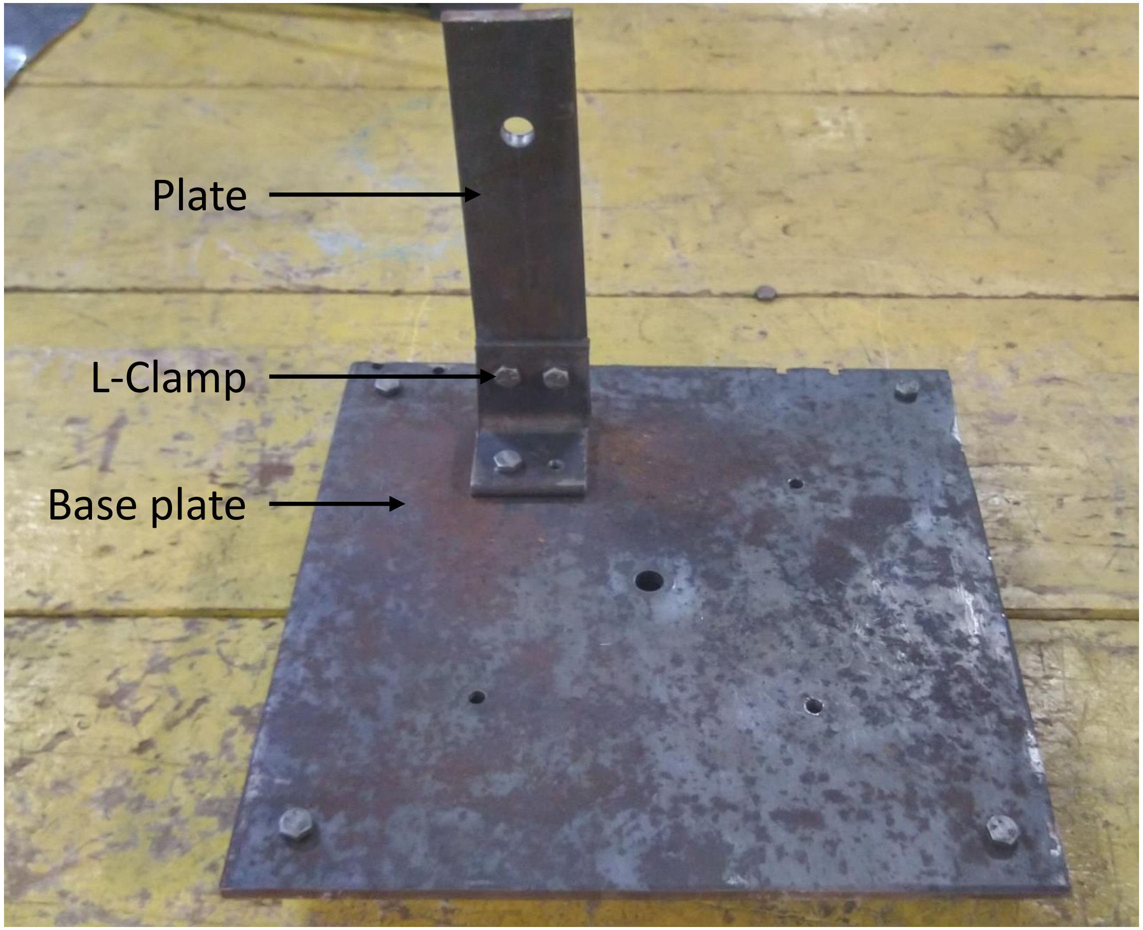




Step 8: Assemble parts



Finally, your L-Clamp should look like this:



Recounting all the steps in making L-Clamp

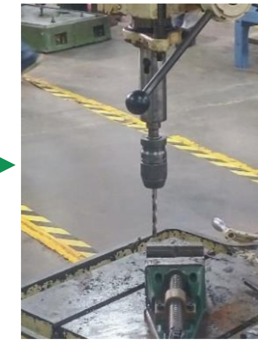
Step 1: Identify raw material

Step 2: Mark location for the hole

Step 3: Punch marked location

Step 4: Move to drilling machine

Step 5: Mount drill chuck and drill



Step 6: Clamp the vice on the work table, and work-piece on the vice

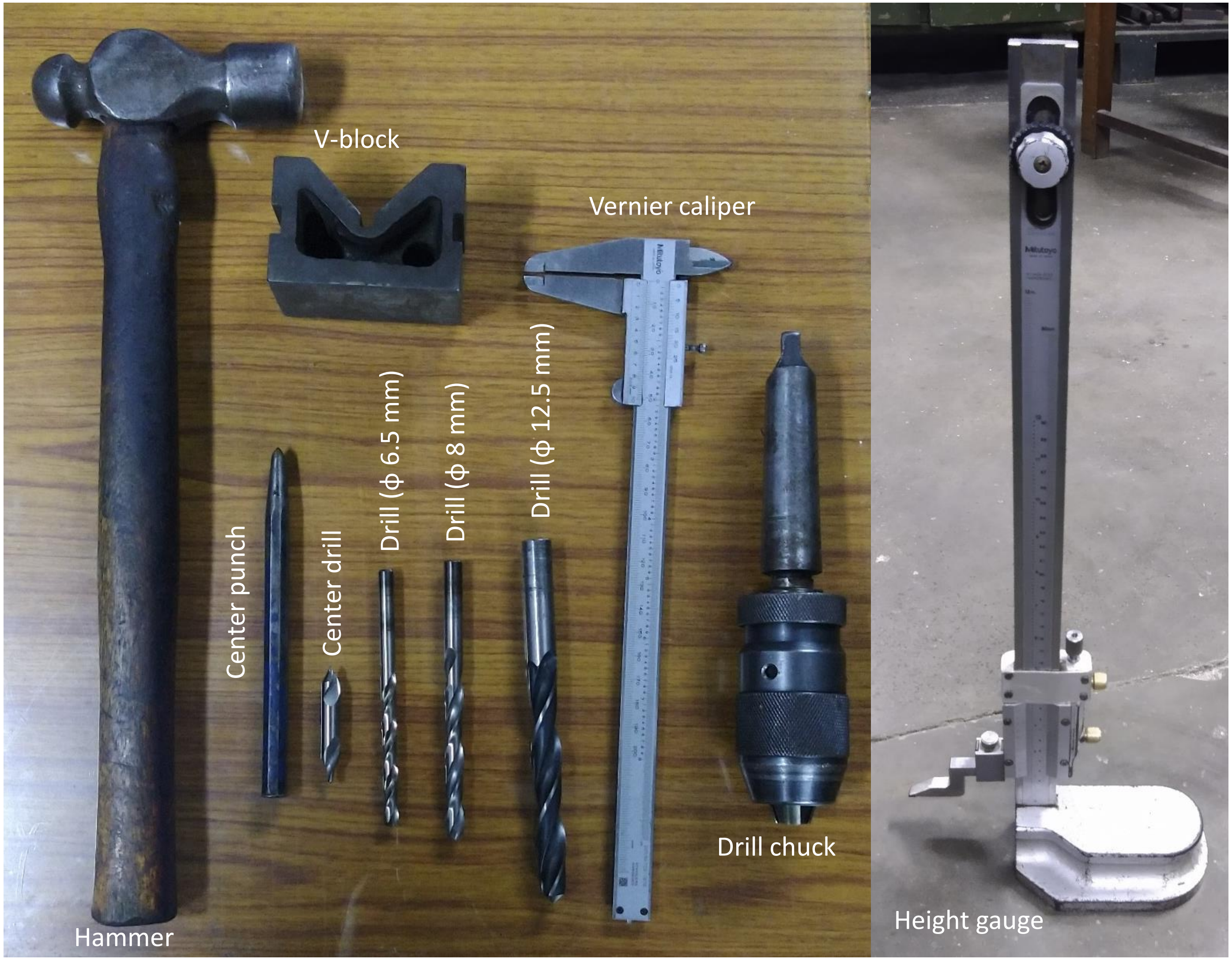
Step 6: Adjust the table position

Step 7: Drill holes one at a time

Step 8: Assemble parts

… and, finally:

You will have used

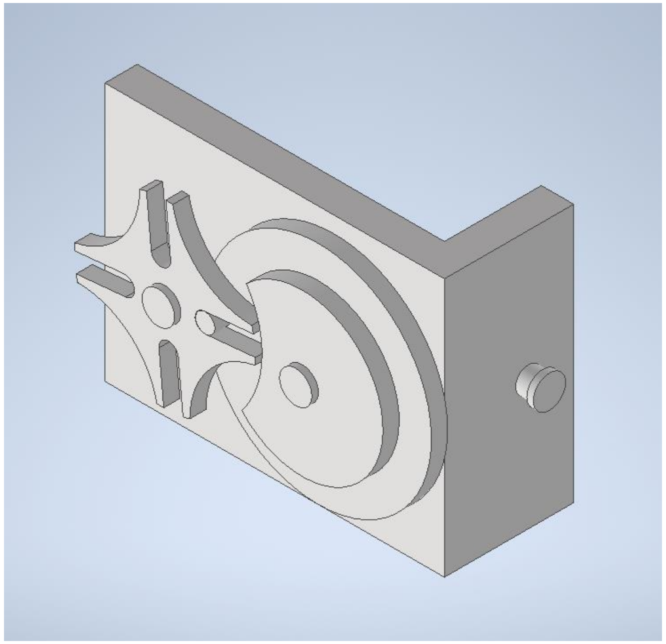


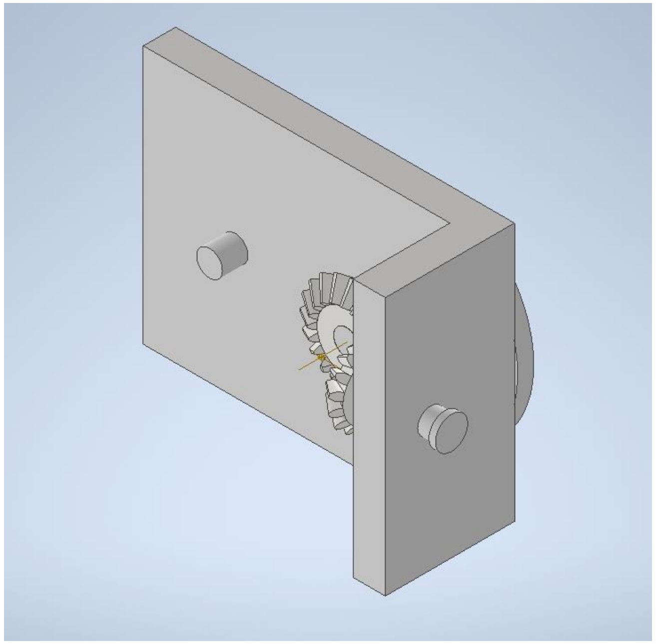
CAD Exercise

Contributed by:
Karthik R,
Senior Project Associate, MadLab, IITK

CAD Exercise

Objective: To familiarize you with CAD by requiring you to make parts and assemblies similar to what we expect you will need to make for your projects. In this CAD exercise you will create parts required for a Geneva Mechanism driven by a set of bevel gears and assemble all the parts. You will also learn to make a 2D manufacturing drawing of a part.



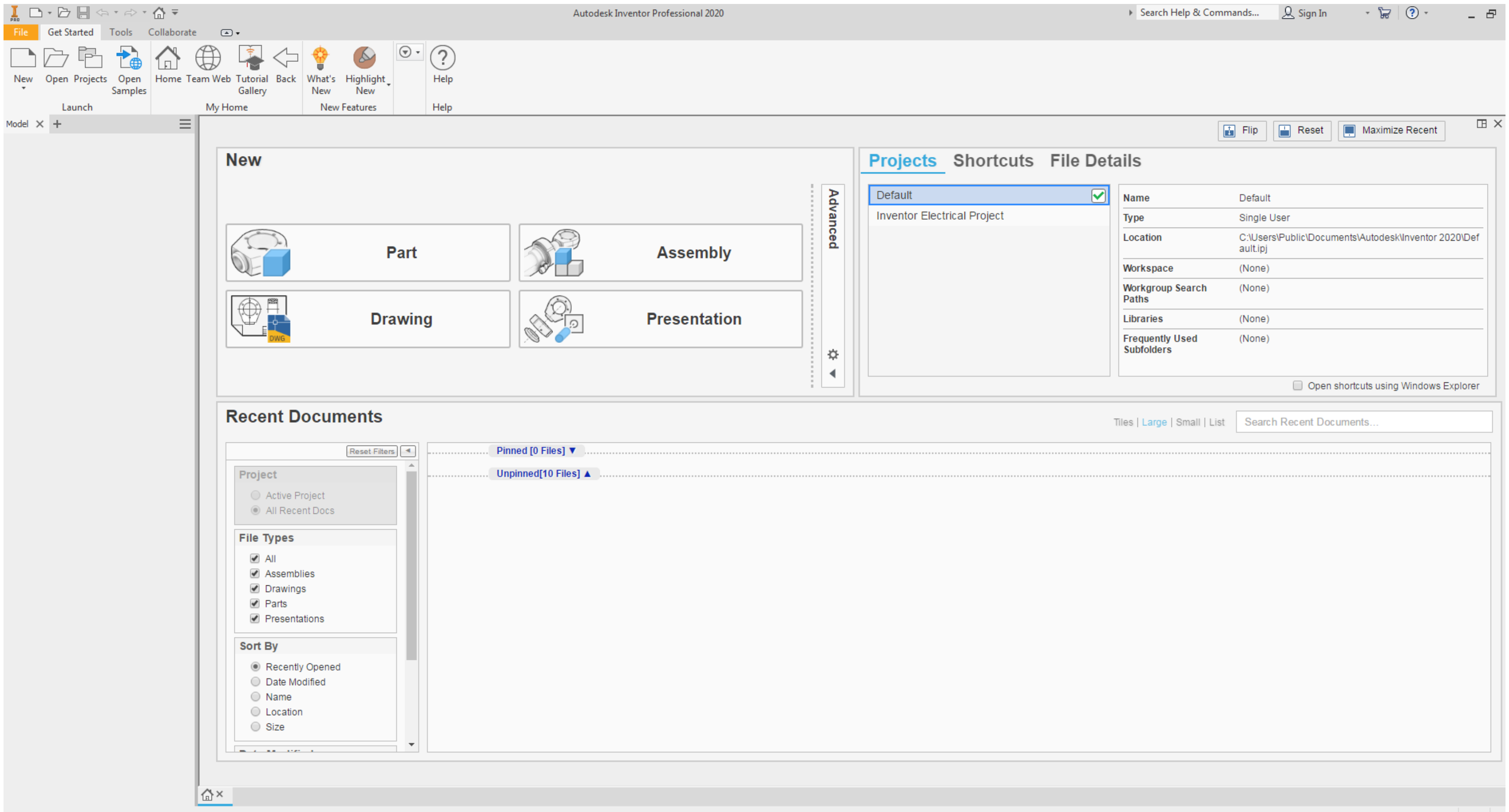


Preliminaries

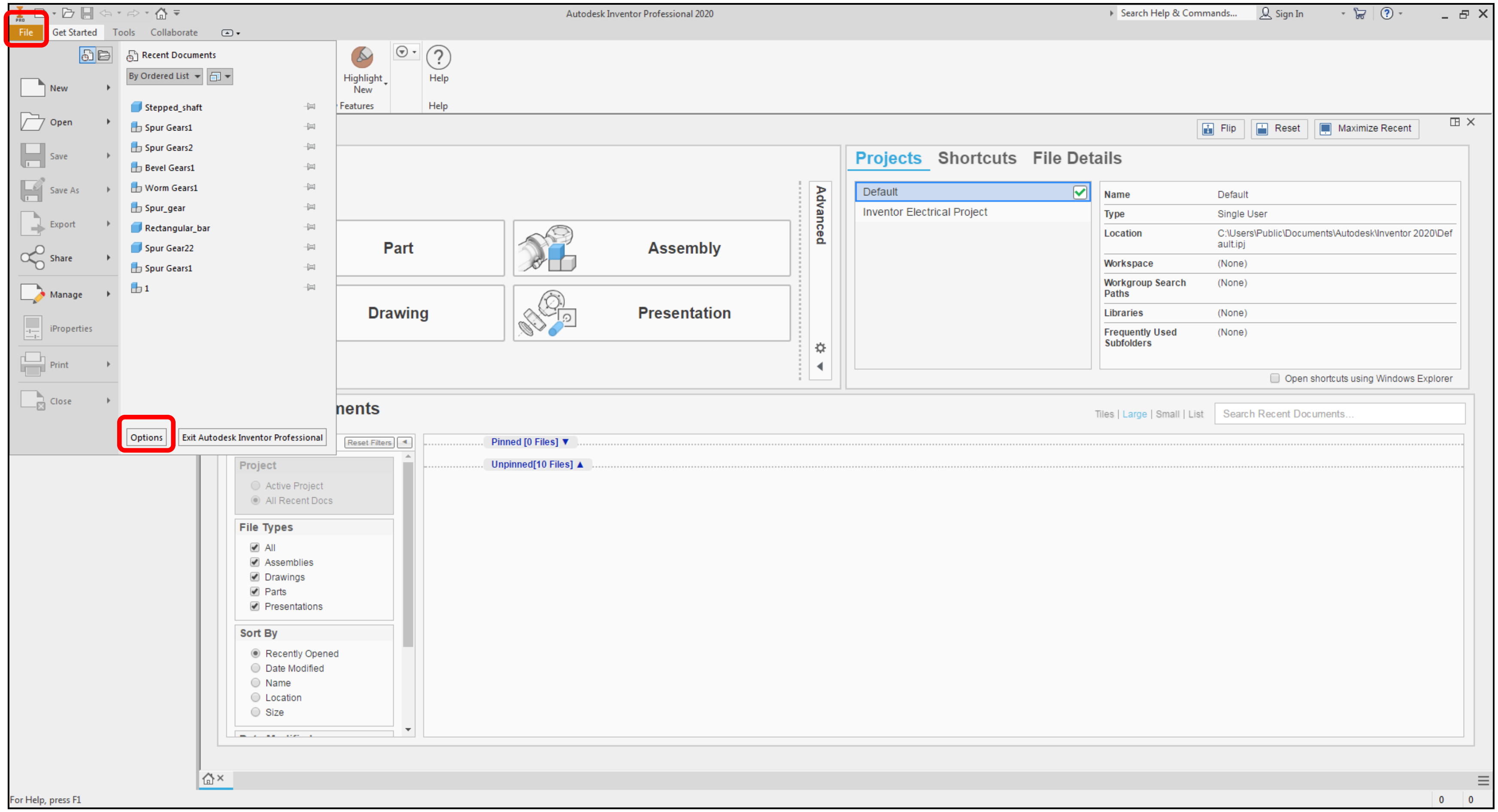
Load the software

You might see an interface like this after loading the software

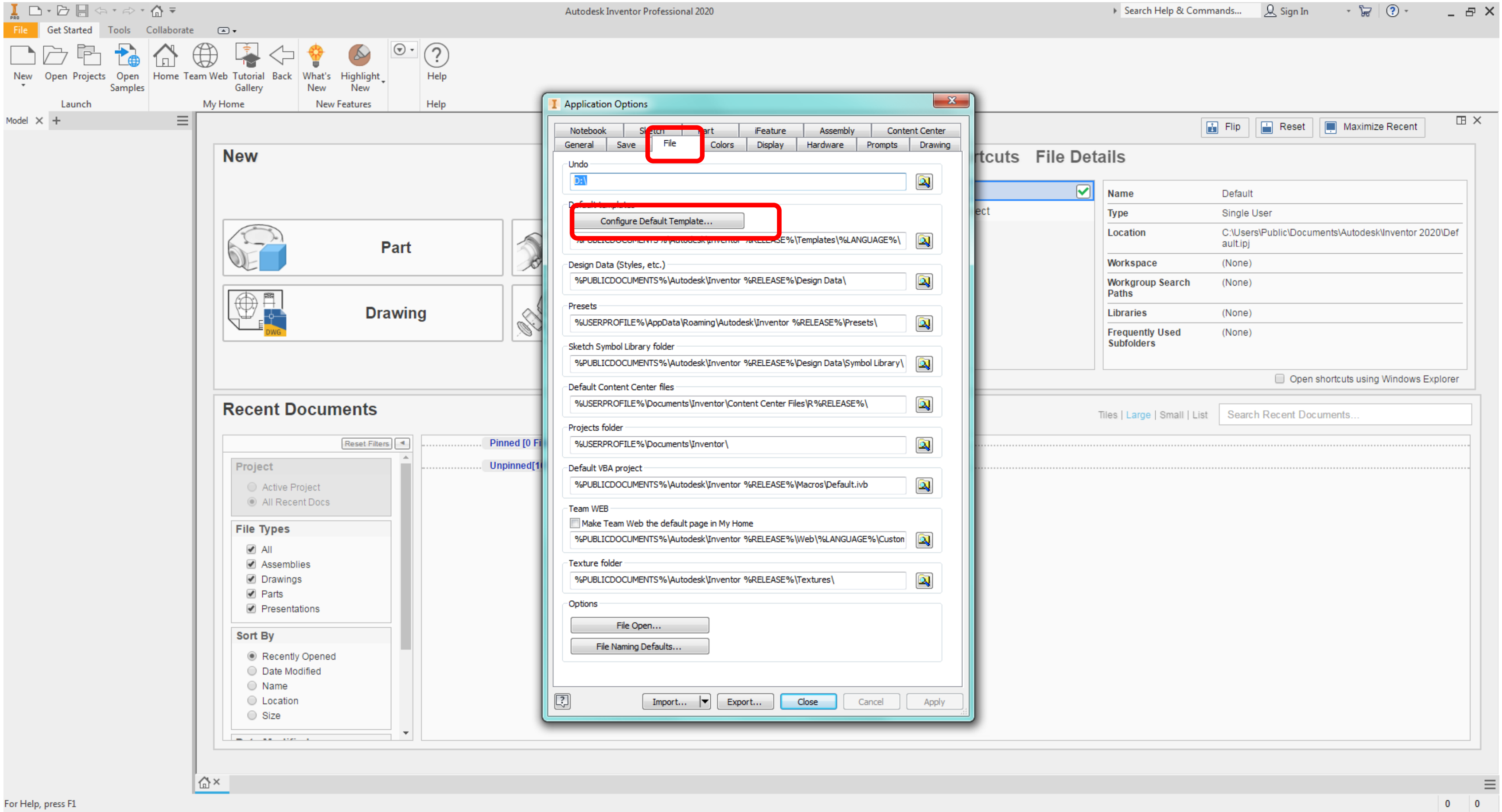
First, configure default units to SI



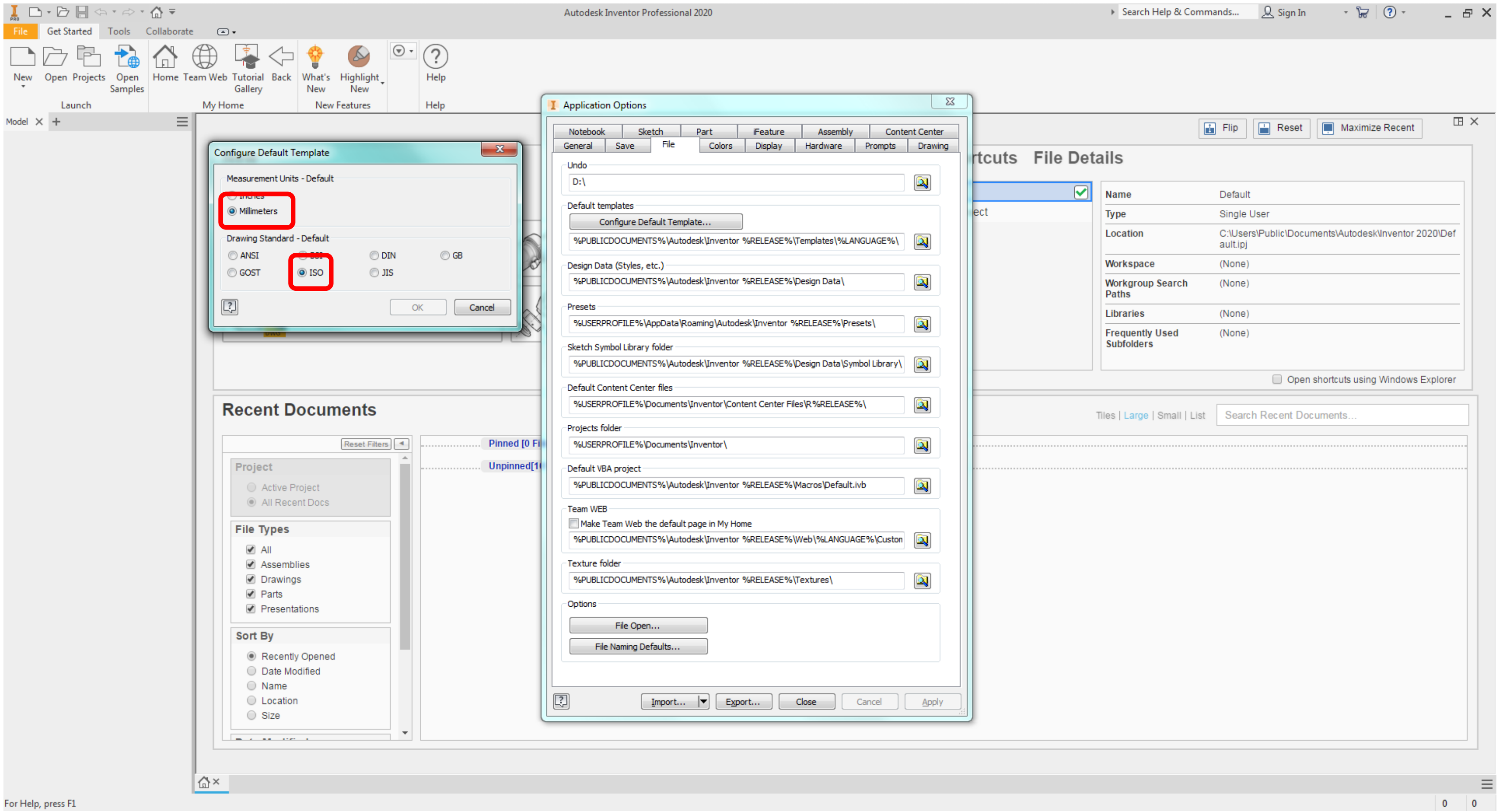
Click on File and then click on options



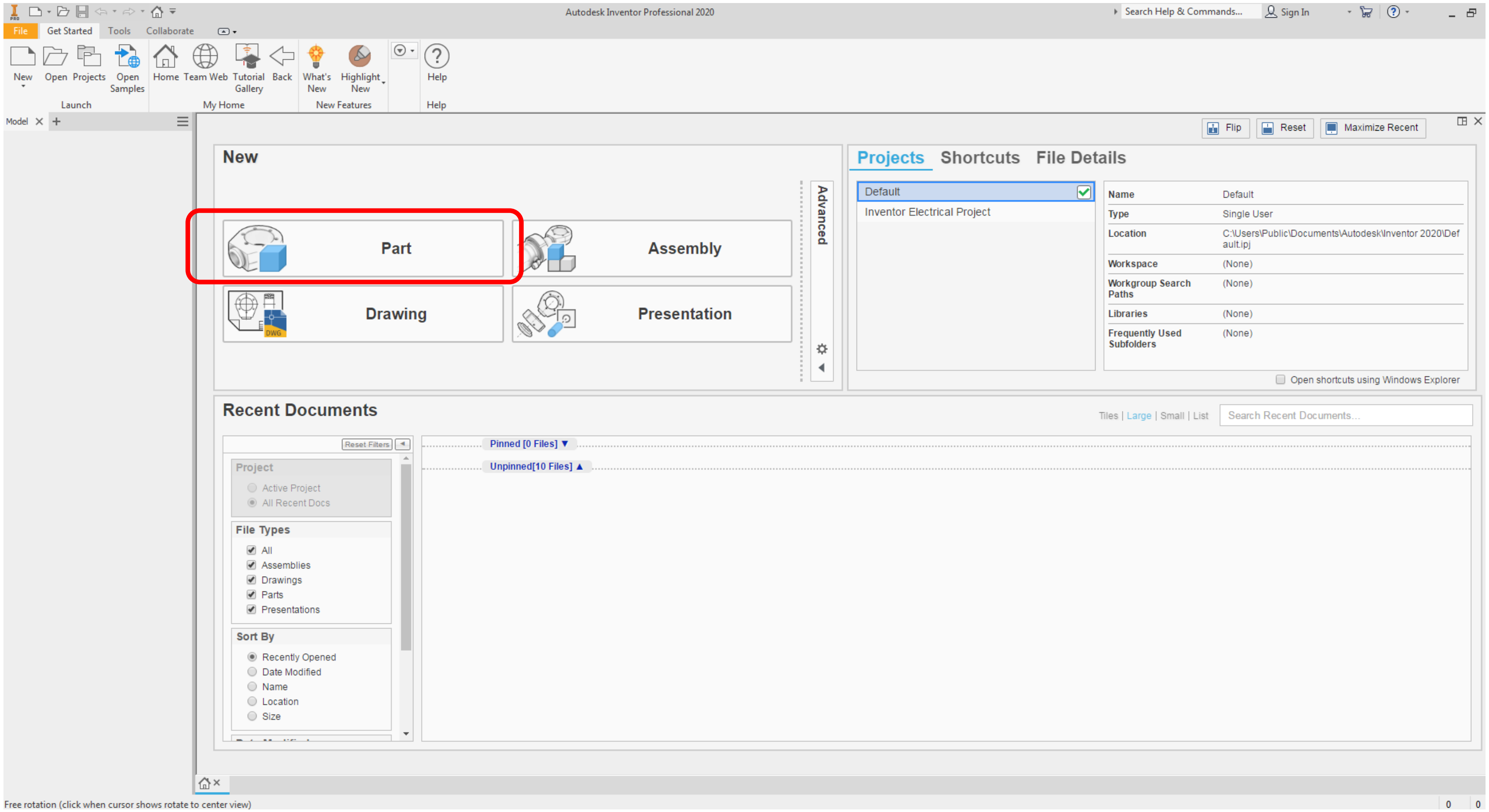
Click on “File” and then click on “Configure Default Template”



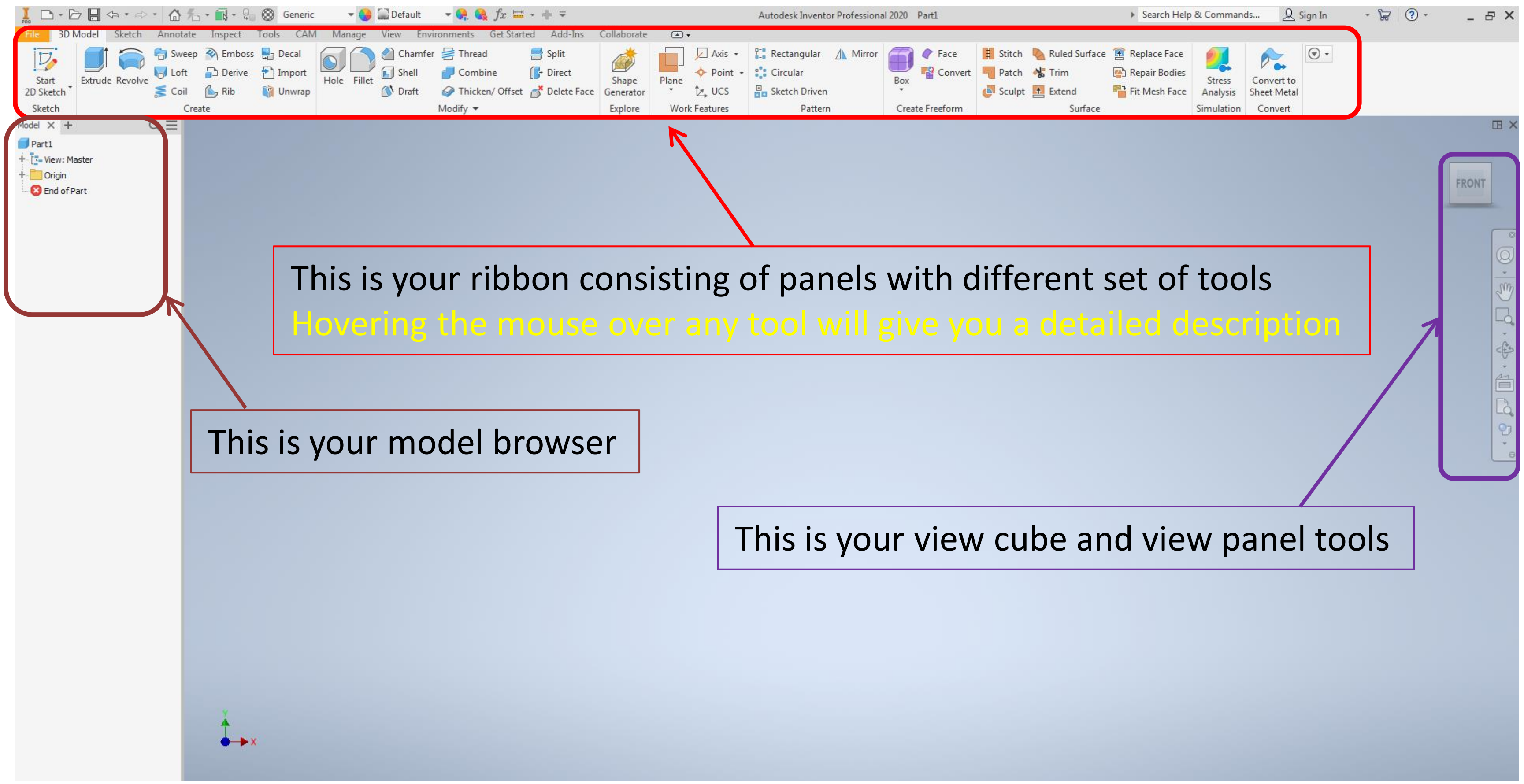
Change Measurement units to Millimeters and Drawing Standard to ISO, and ‘Apply’ the changes. Your default units should now be SI.



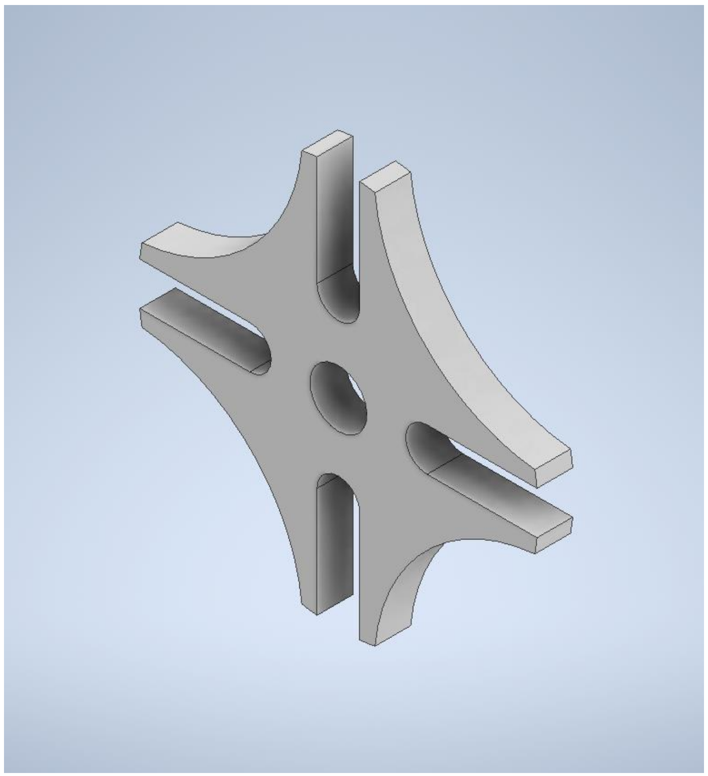
Click on “Part”, to begin Part Modeling

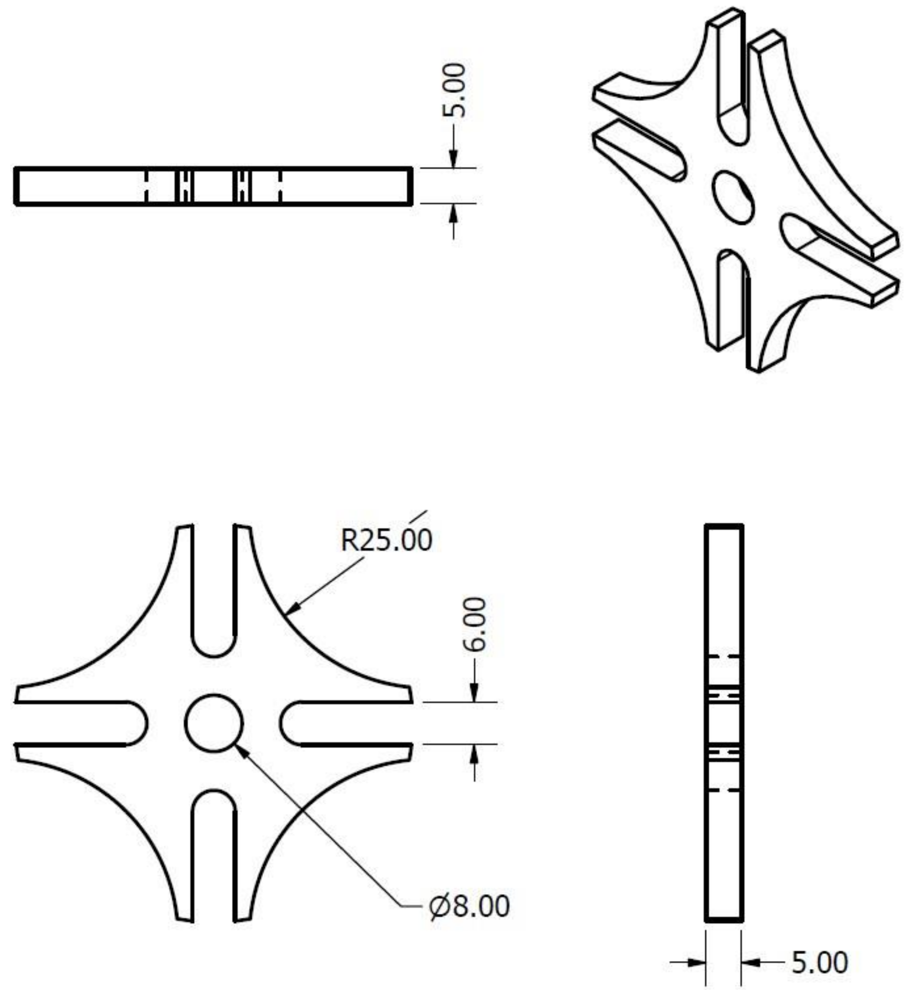


Your interface should look like this
Most of the common tools like Undo(Ctrl+Z), Redo(Ctrl+Y), Copy(Ctrl+C) and Paste(Ctrl+P) work the same here

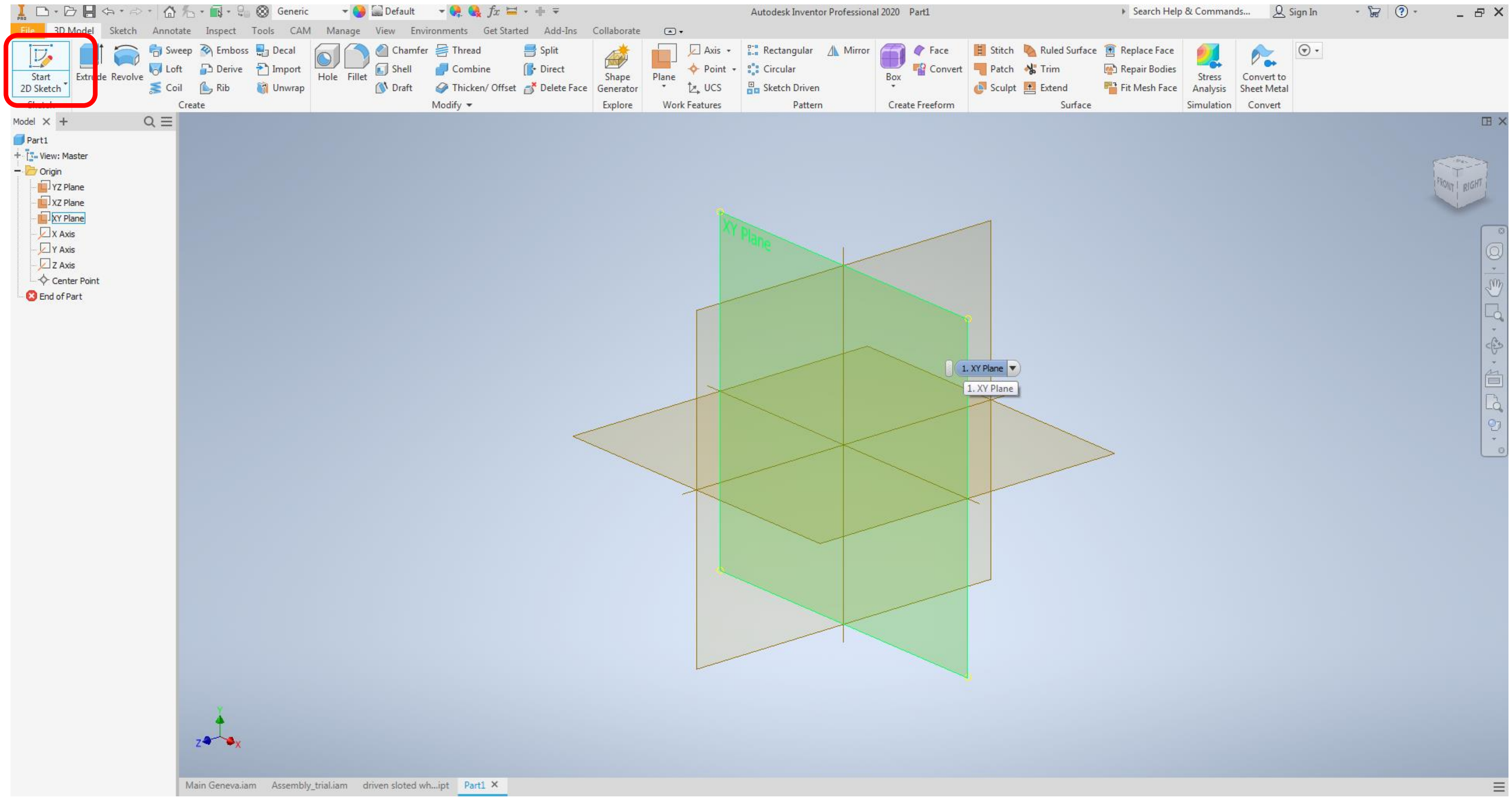


Task 1
To make a Geneva Driven Wheel

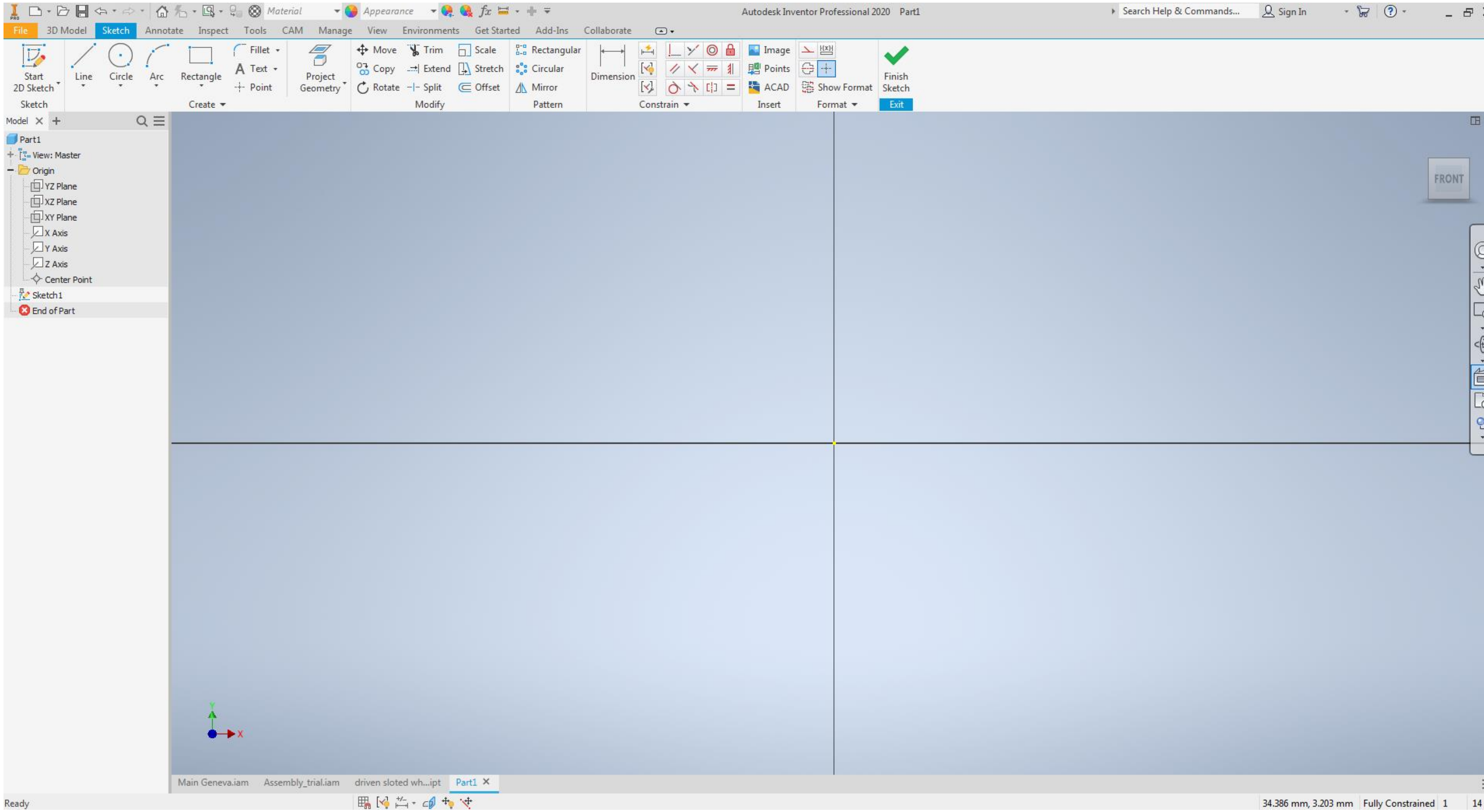




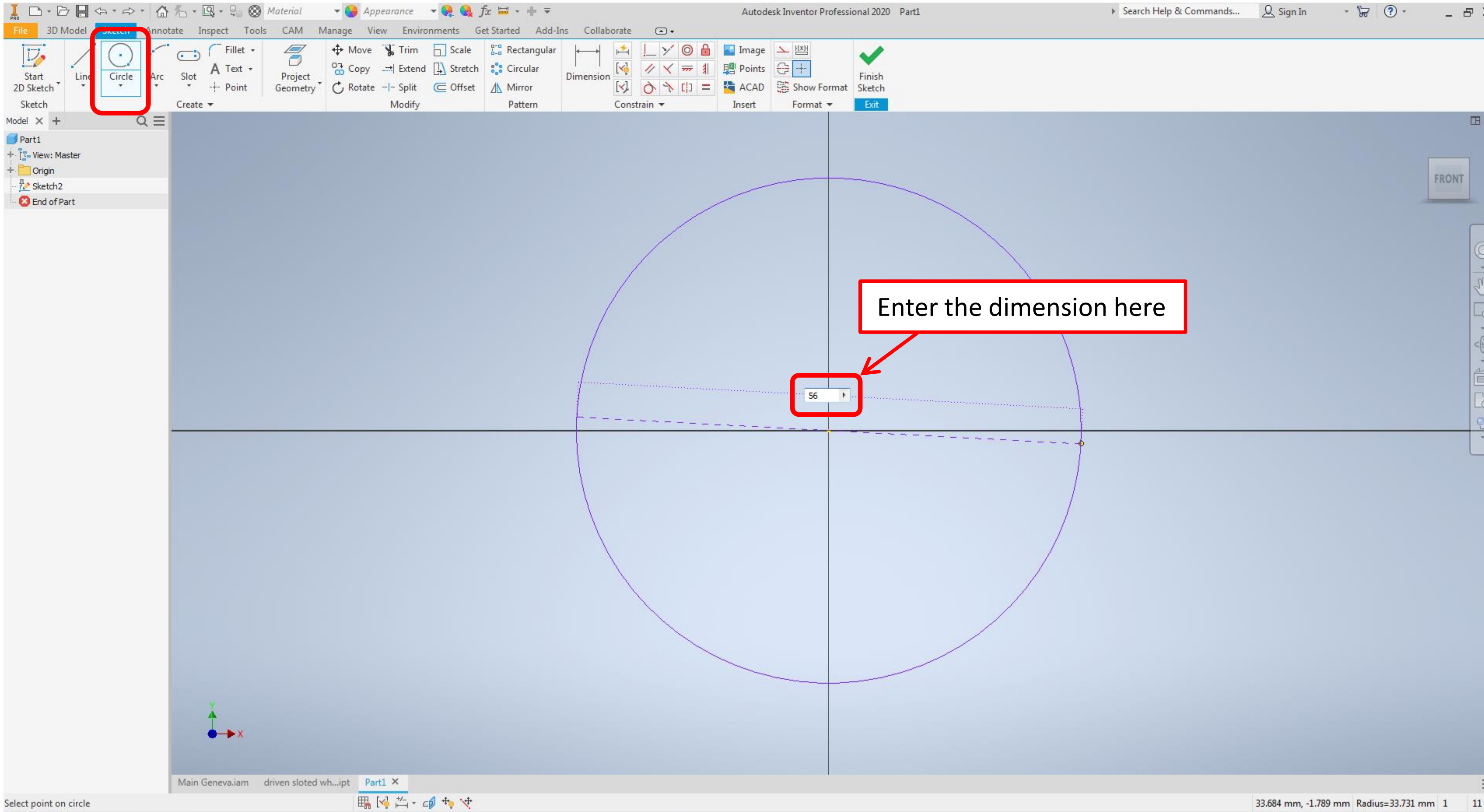
Click on Start 2D sketch and select the XY plane



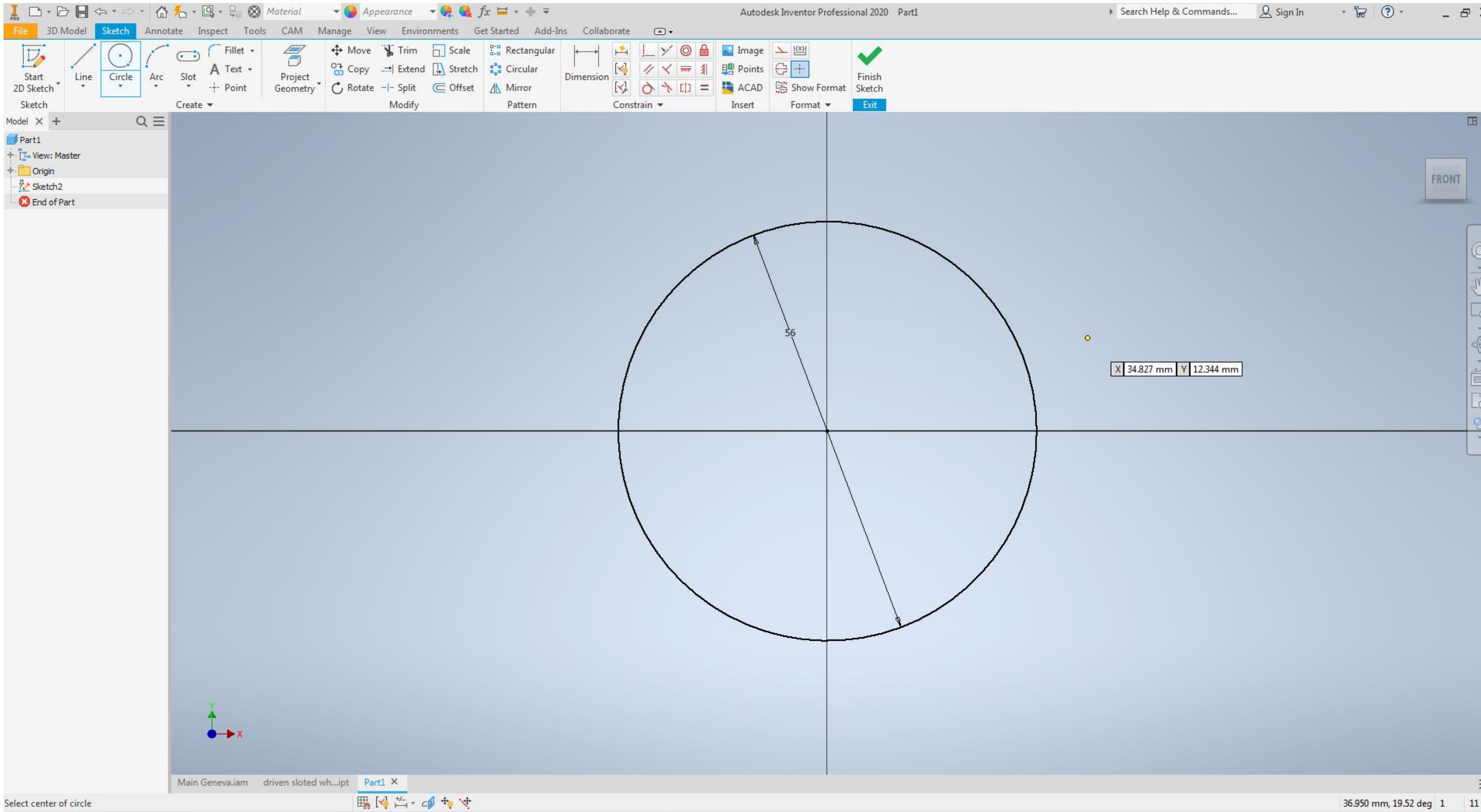
You will see the cross-hair



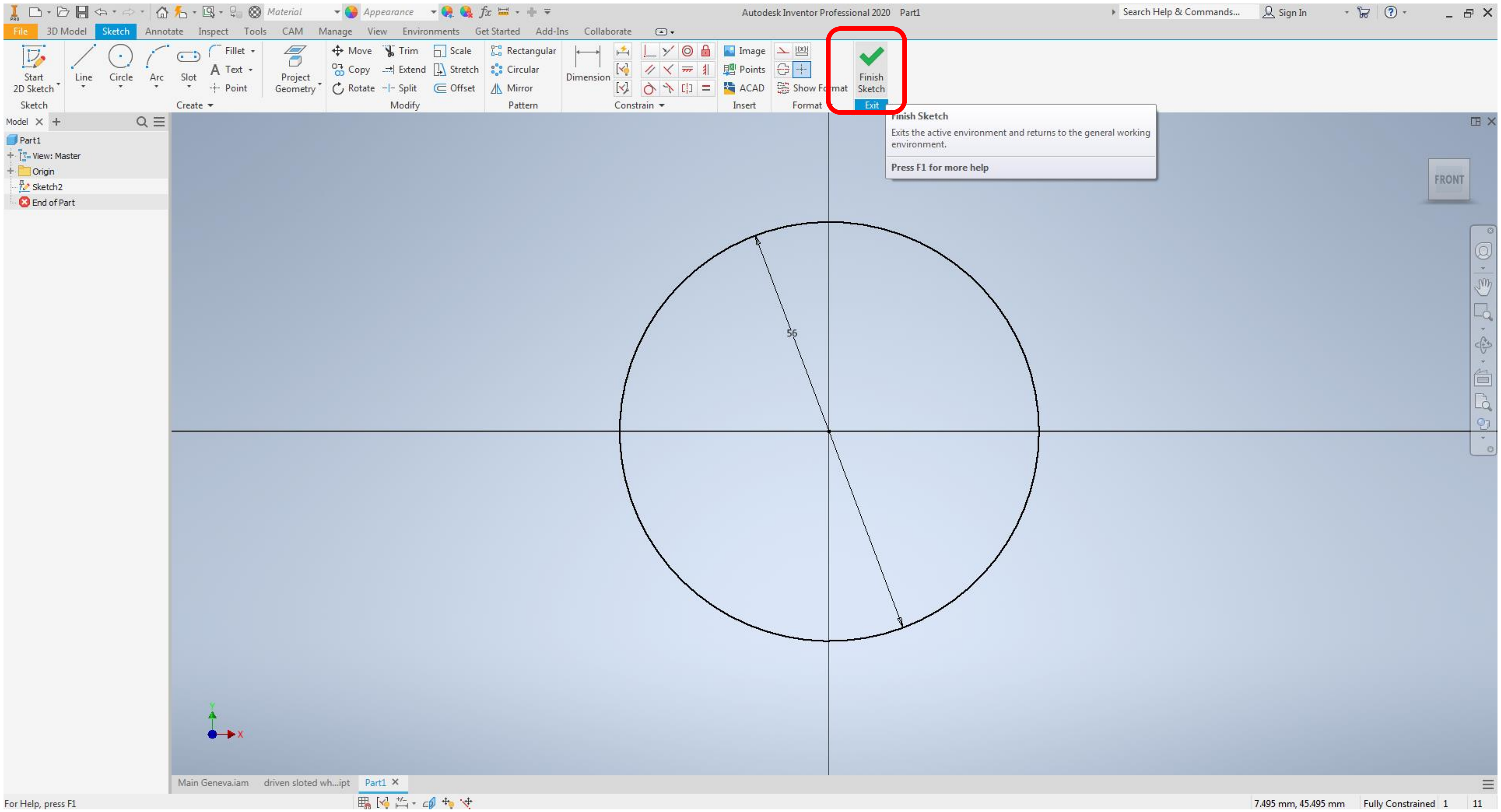
Click on “Circle” and click on the origin to set the center
You can enter the dimension directly.
Set the diameter to 56mm and press enter.



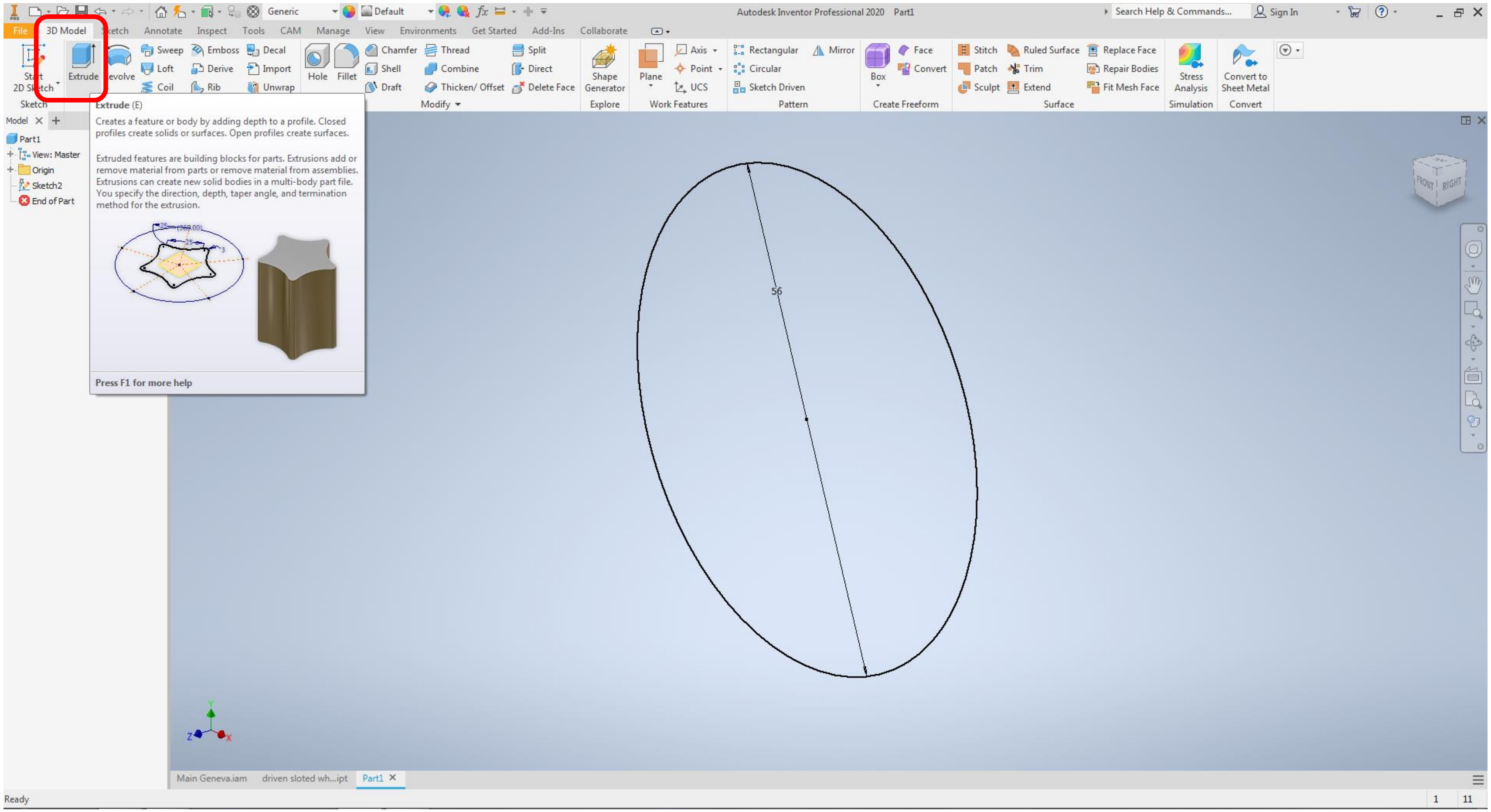
Press “Esc” to exit circle mode.



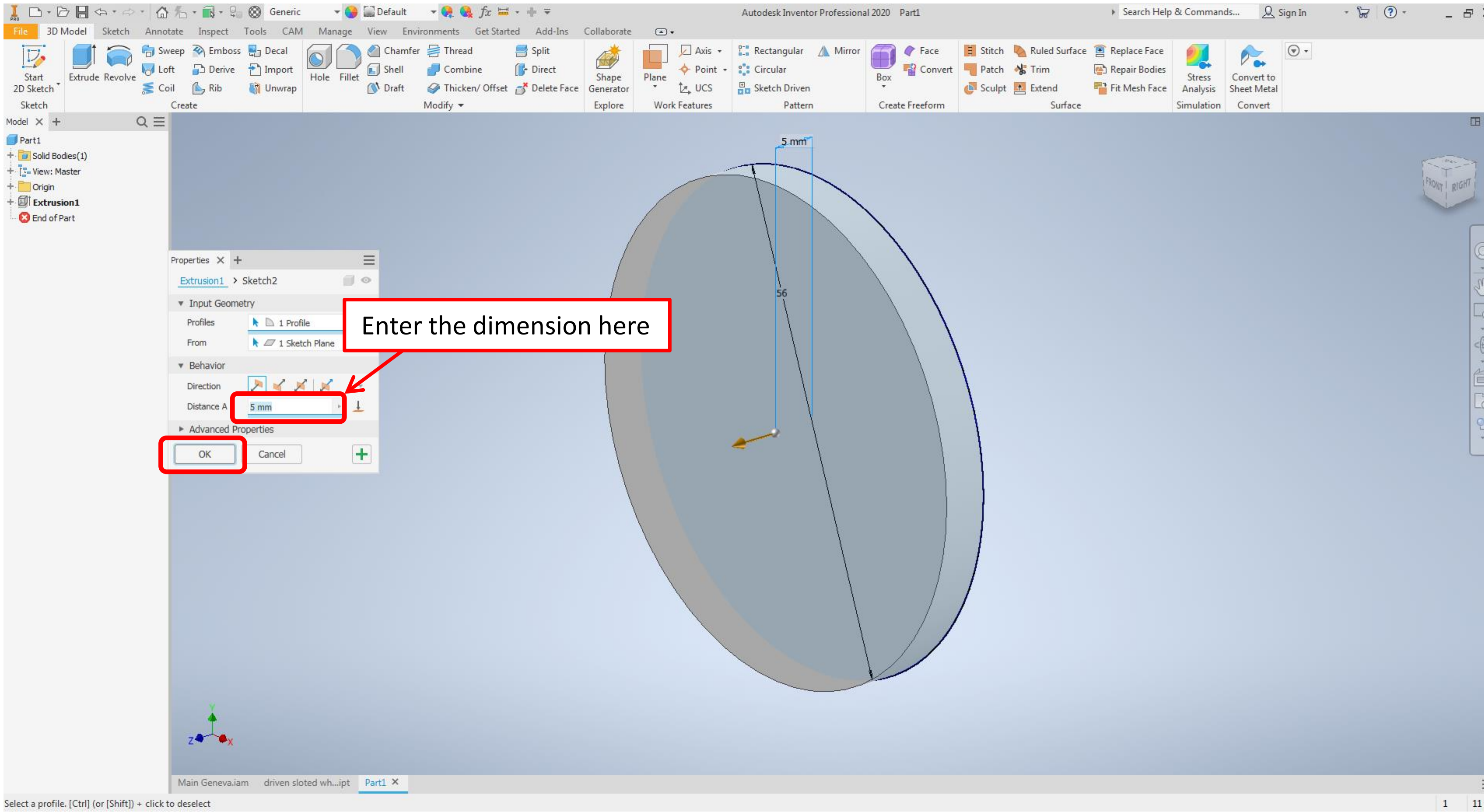
Click on “Finish Sketch” to exit the sketch mode.



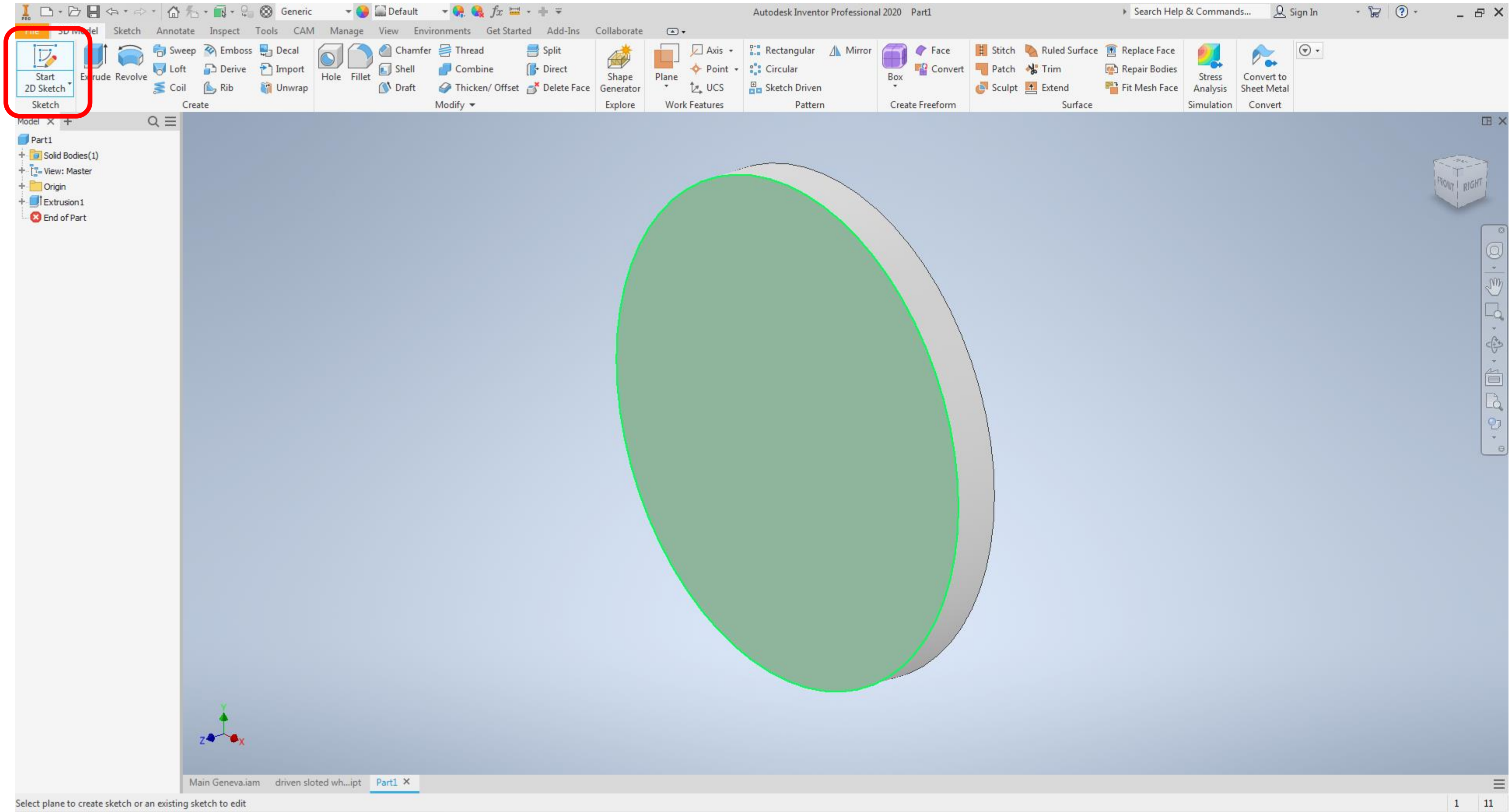
In 3D Modeling mode click on “Extrude”



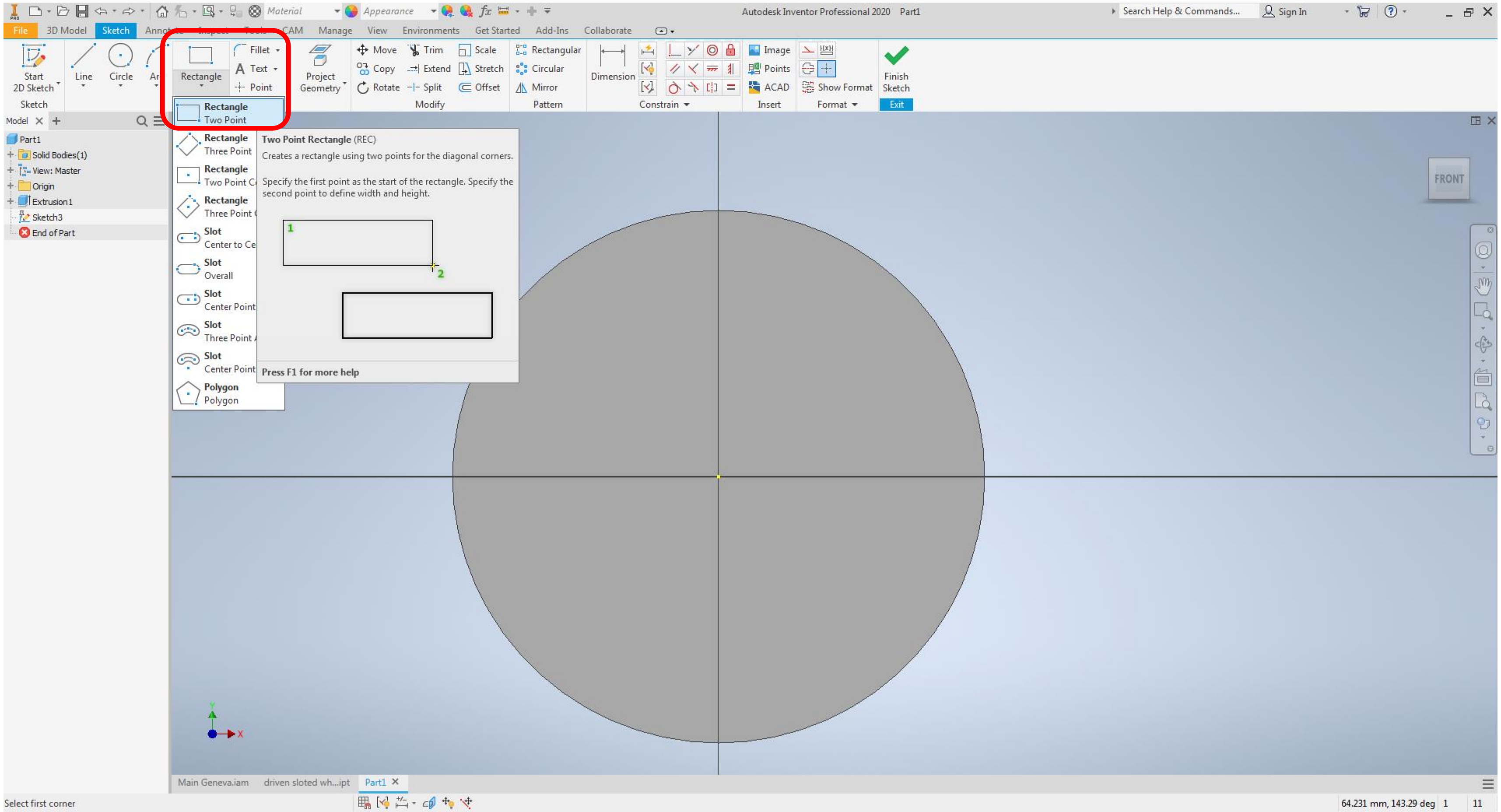
Extrude the circle into a cylinder of 5mm in positive Z direction and click “OK”



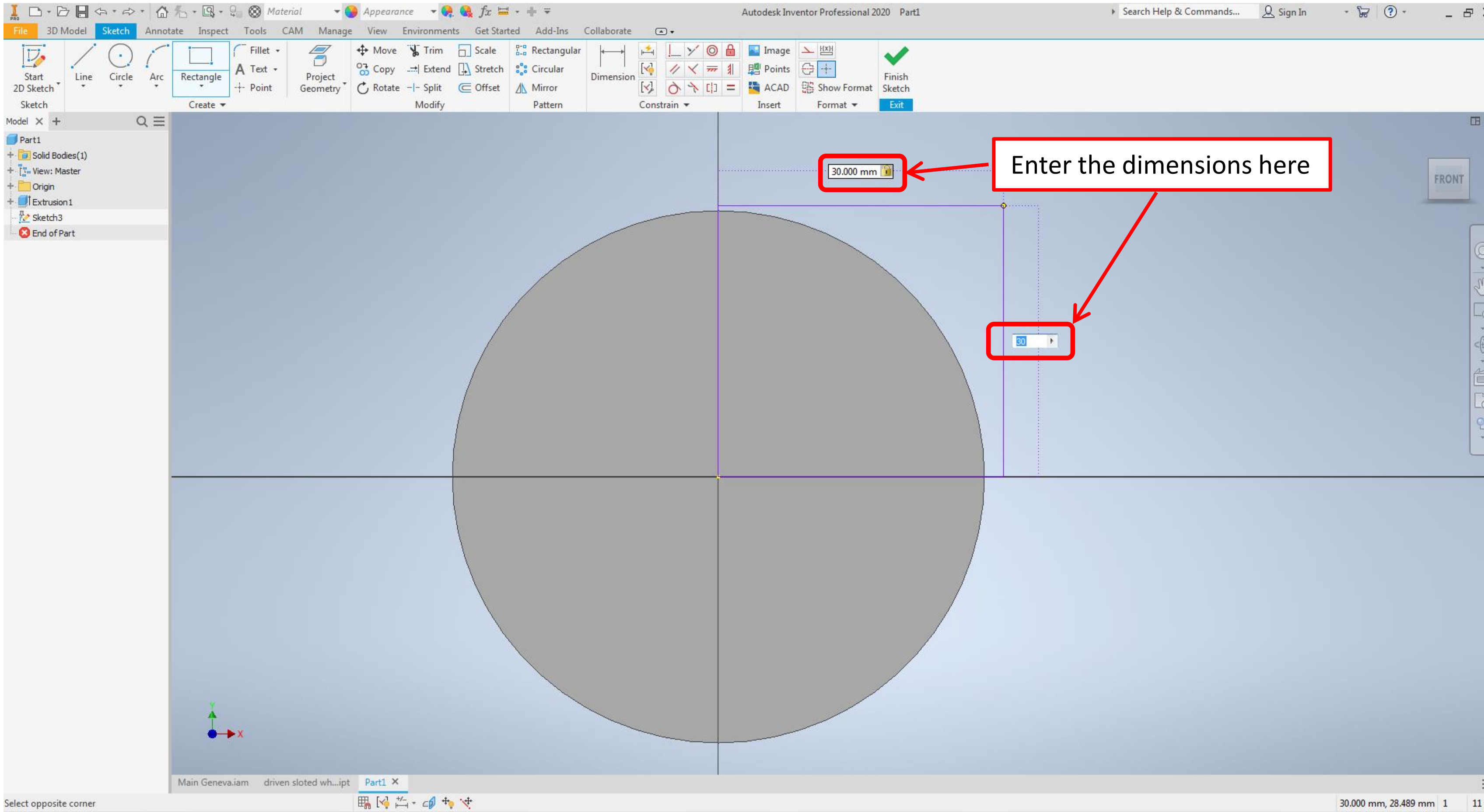
Click on “Start 2D Sketch” and select the top face of the cylinder



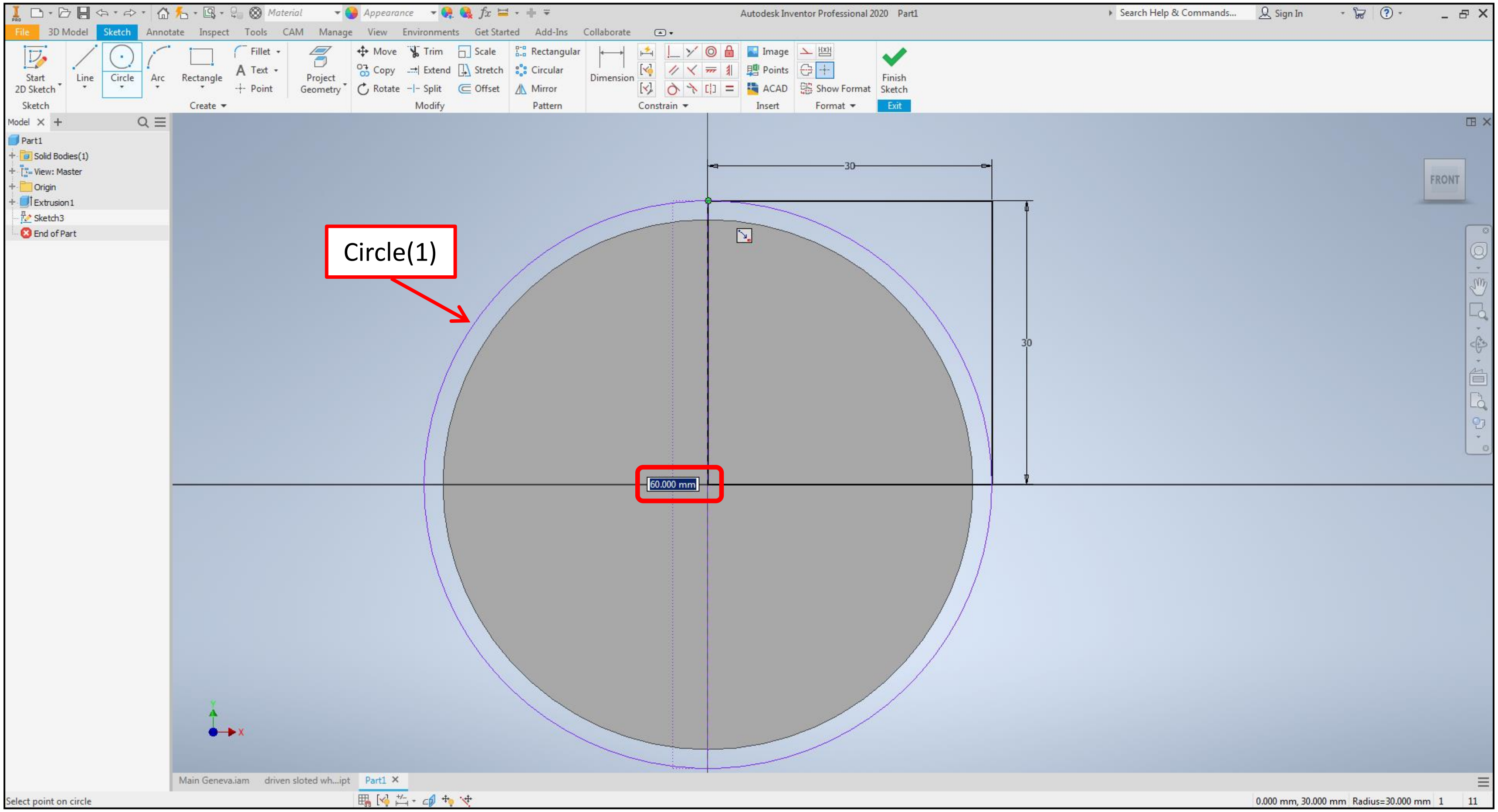
Click on "Rectangle (Two Point)" and click on the origin to begin



Enter the dimension 30mm and press Tab to switch to second dimension
Enter 30mm again and press Enter to create a square

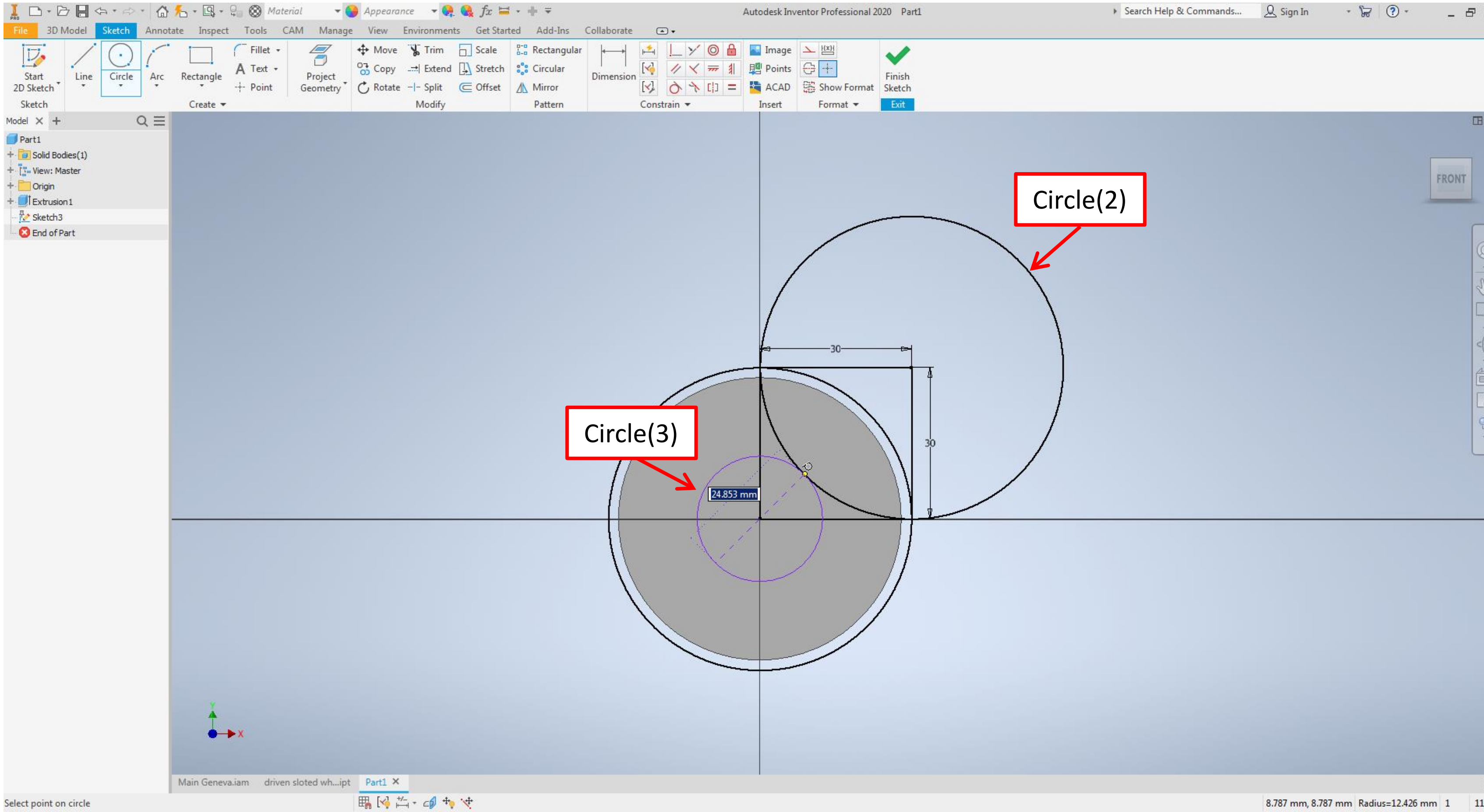


Create a Circle (1) of diameter 60mm with the origin as the center

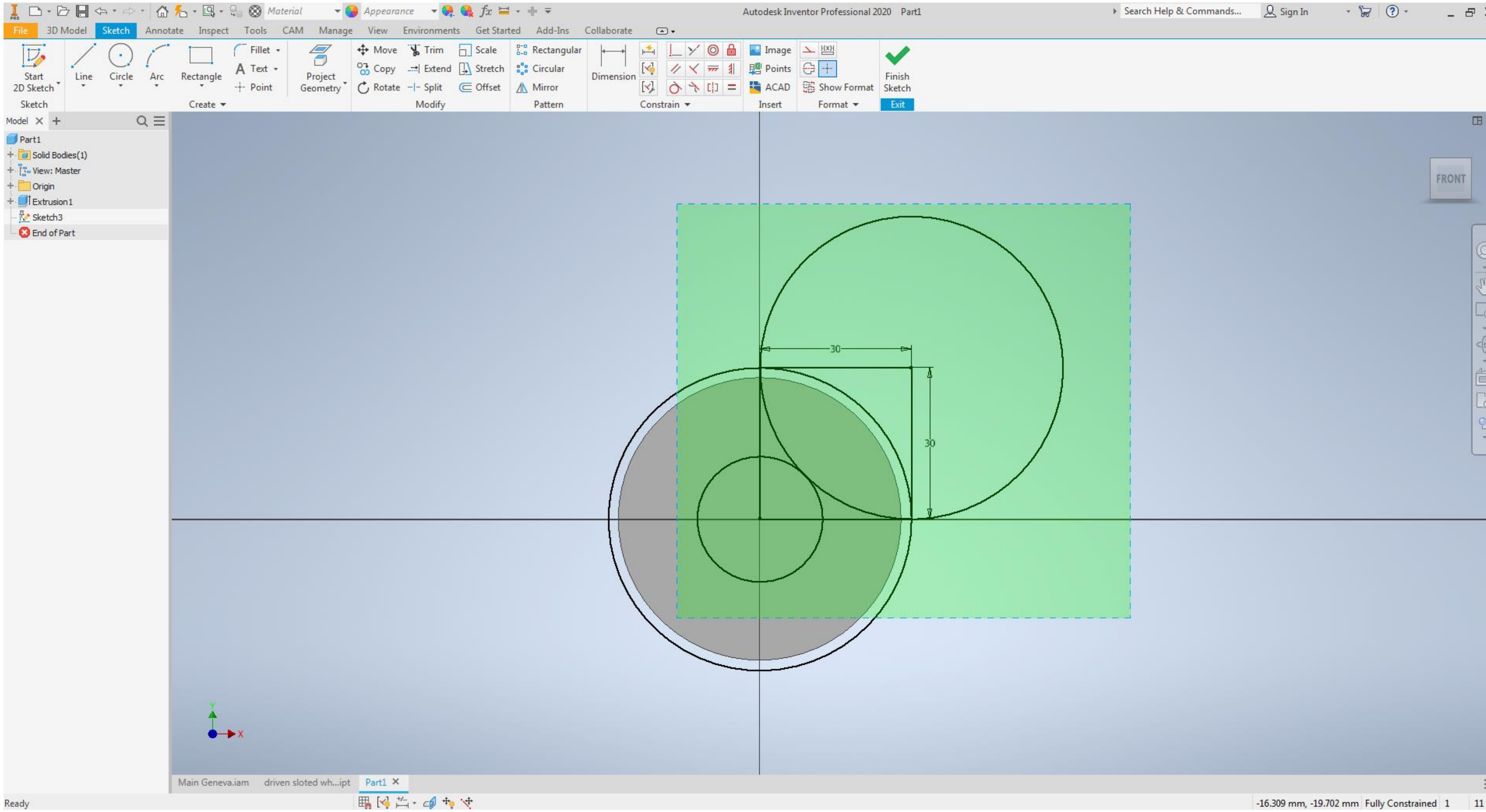


Create Circle (2) with the top right corner of the square as the center and a diameter of 60mm

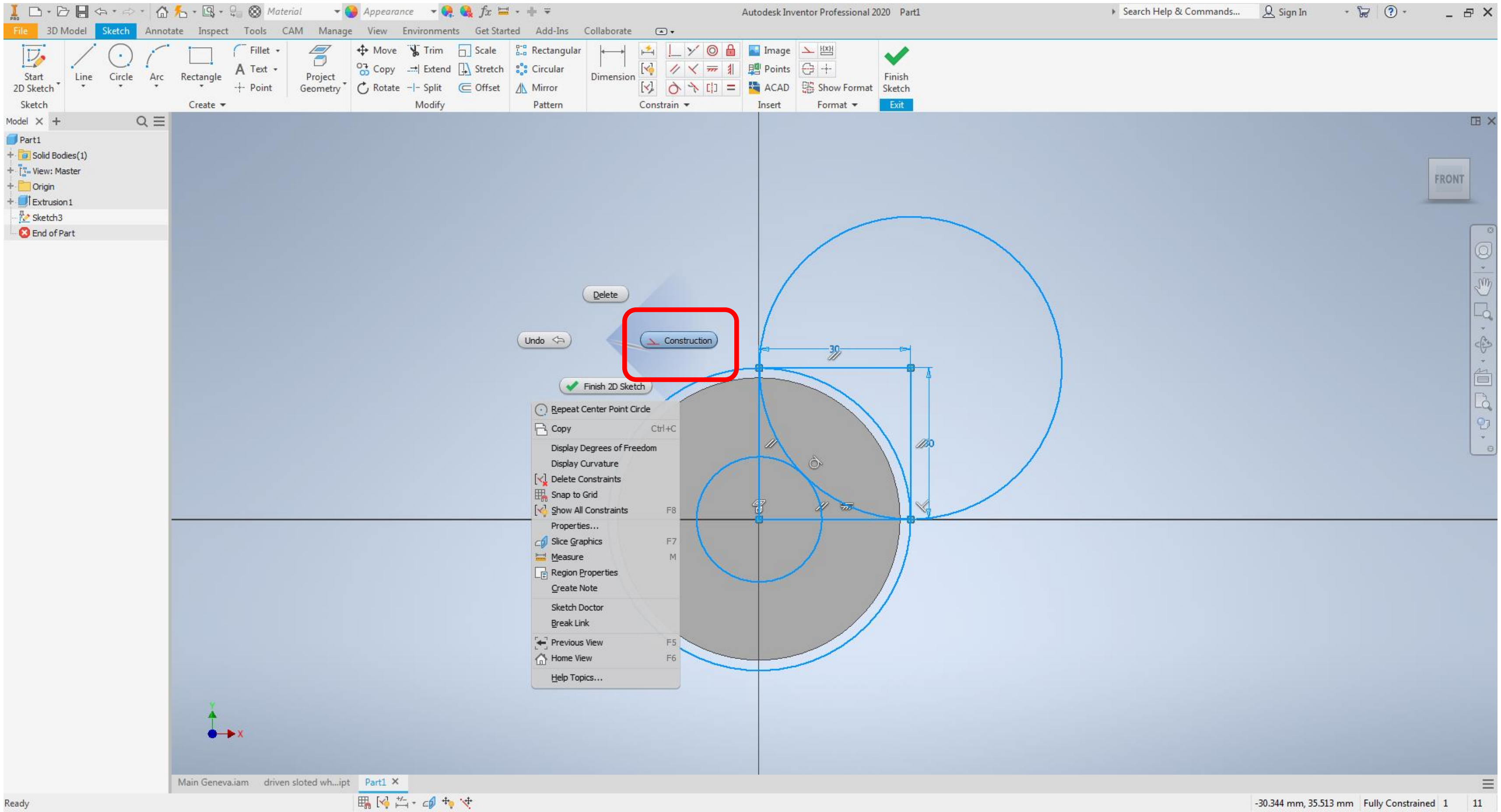
Create Circle (3) with origin as center and click on the circumference of the circle (2) to make them touching circles



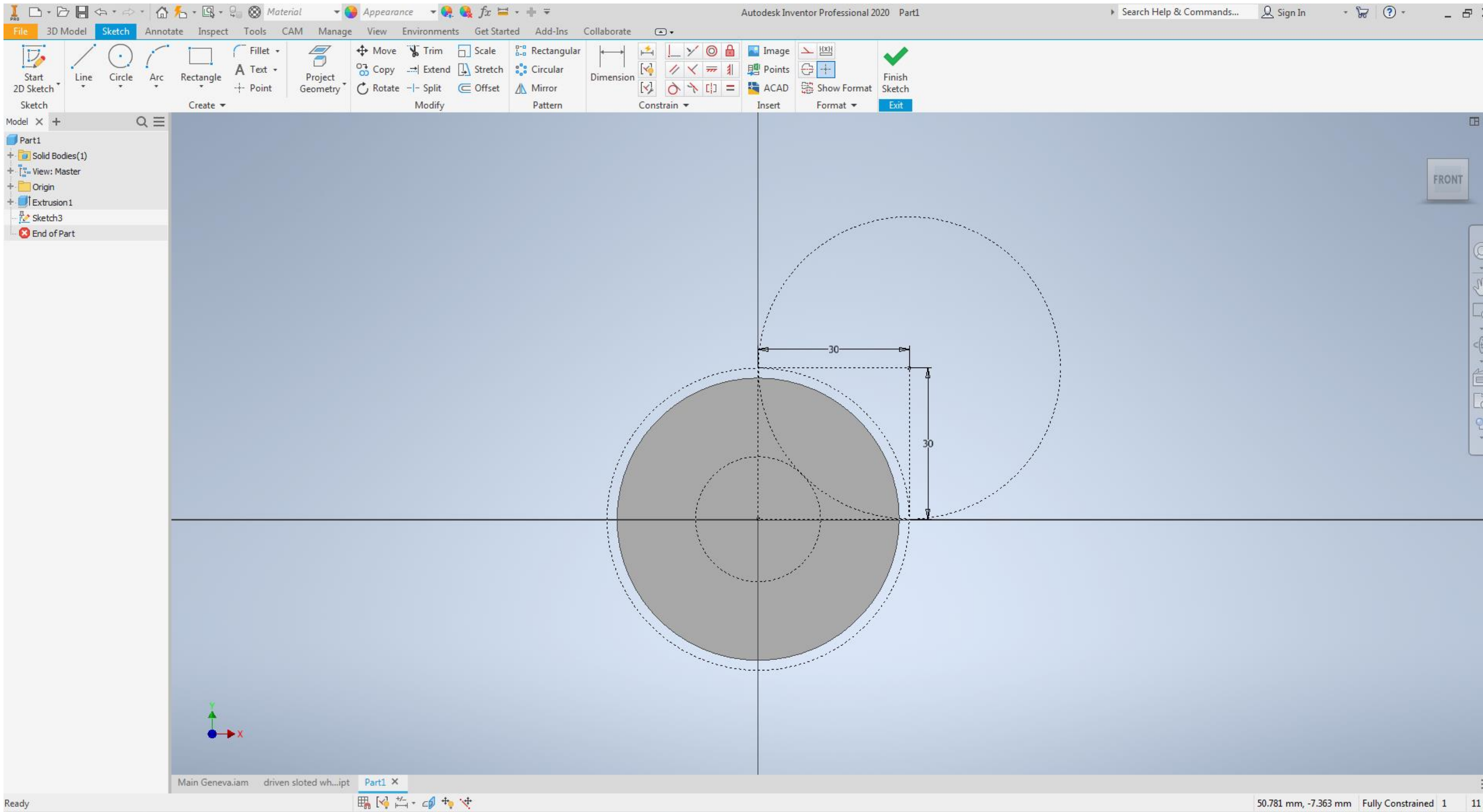
Left click on your mouse and drag towards left to select all the shapes
Left drag selects every geometry that the selection box touches



Right click and select “Construction” to convert the selected geometries into construction lines

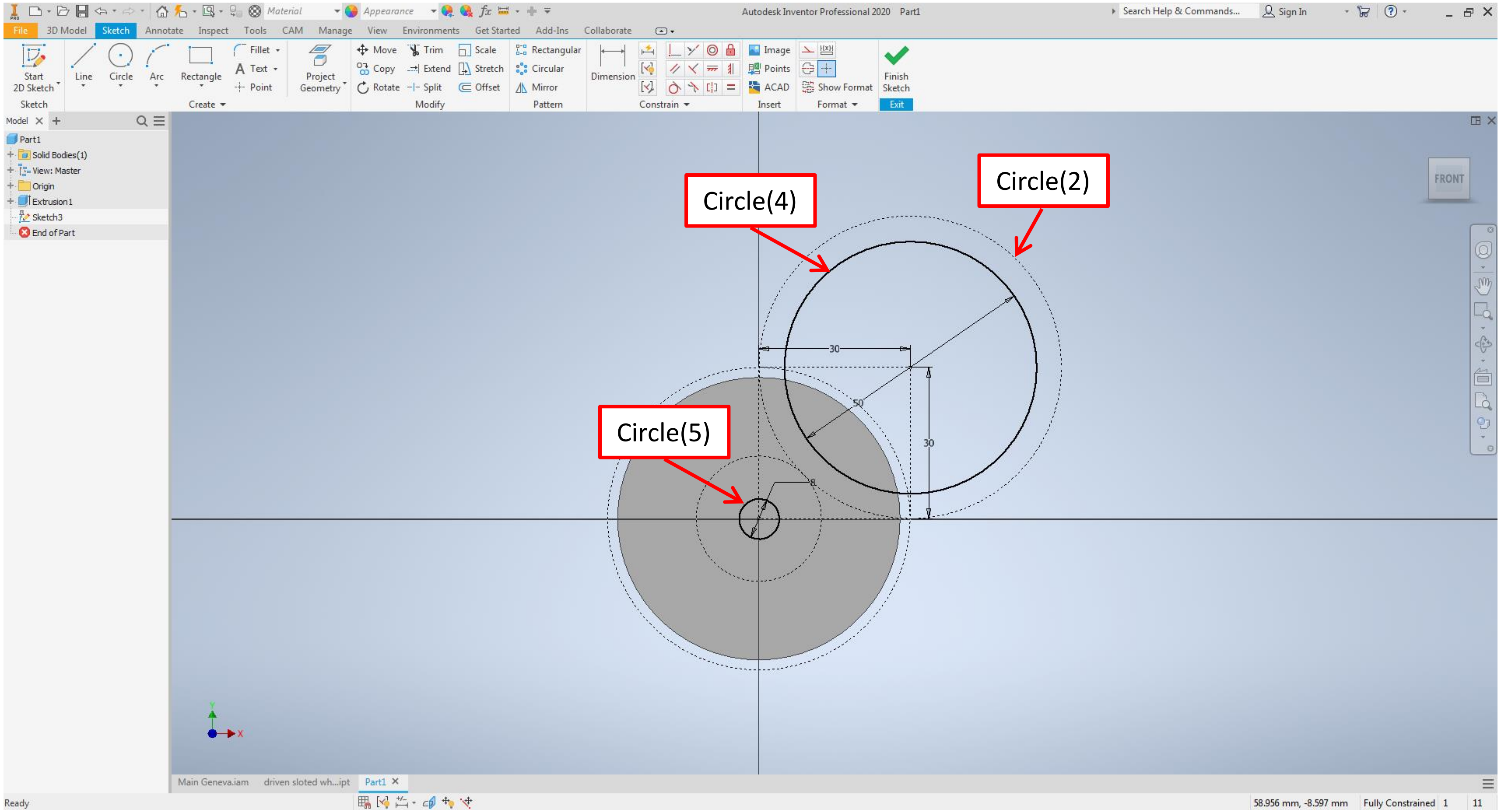


Your drawing should look like this

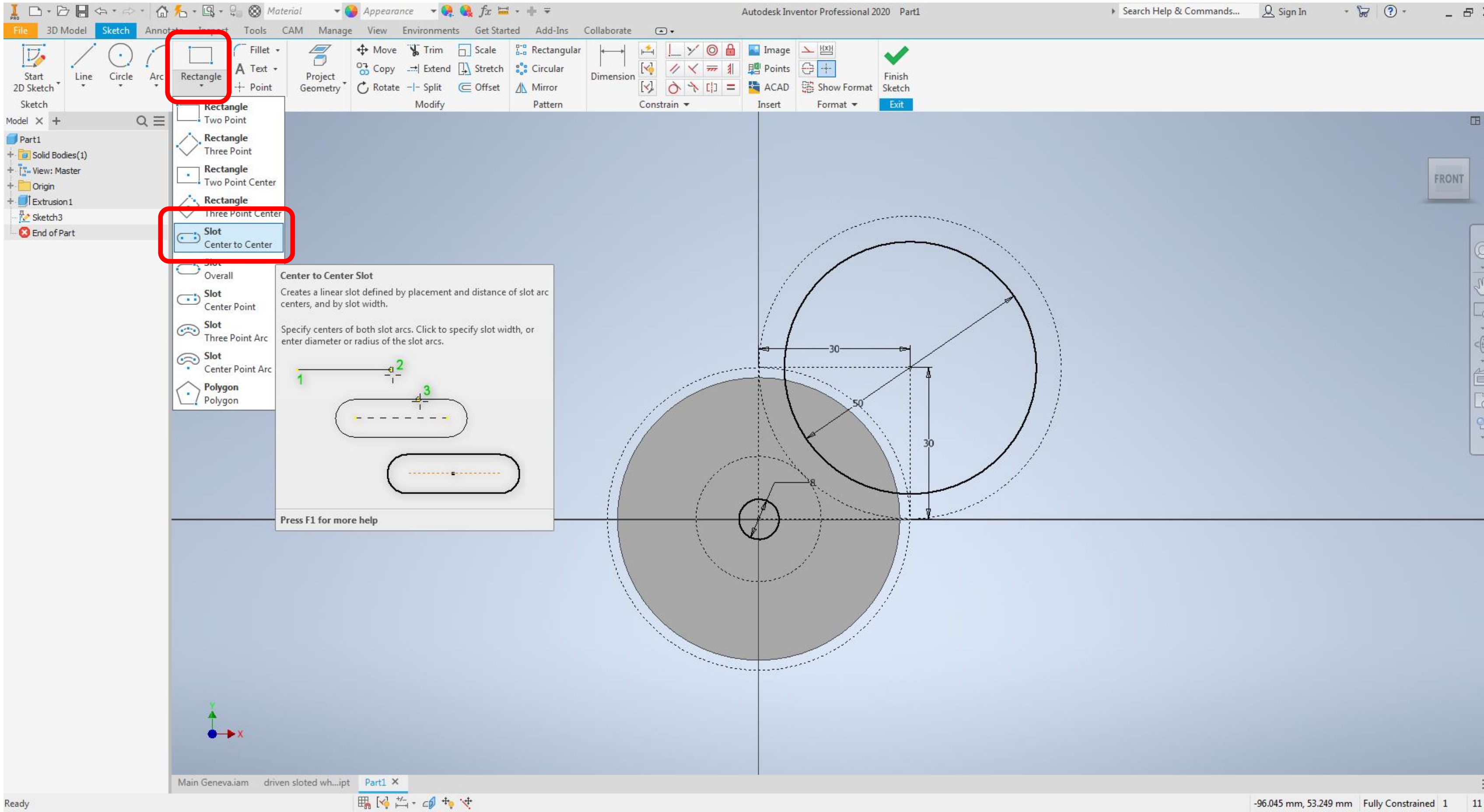


Draw circle(4) concentric to circle(2) with a diameter of 50mm

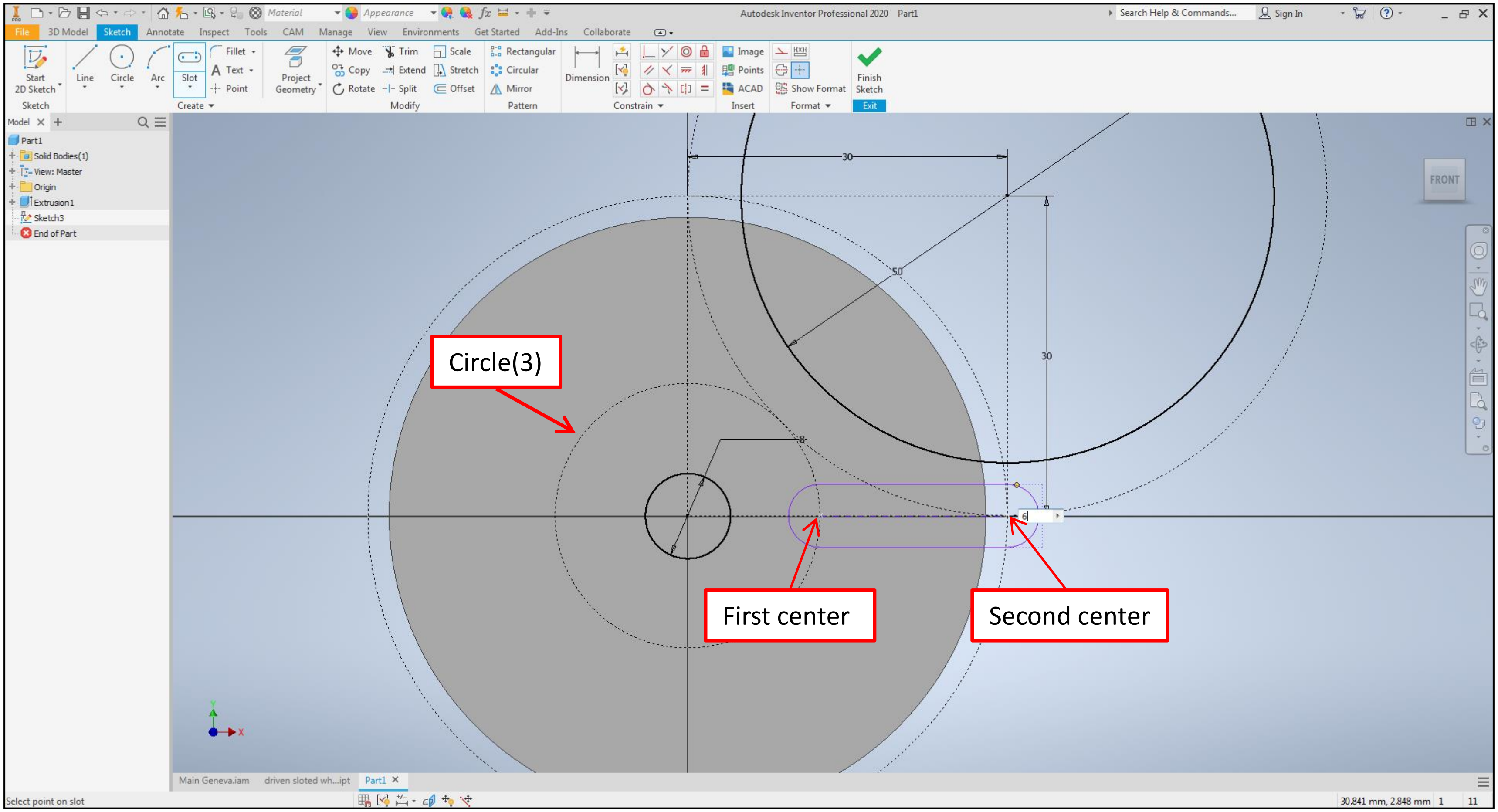
Draw circle(5) concentric to the origin with a diameter of 8mm



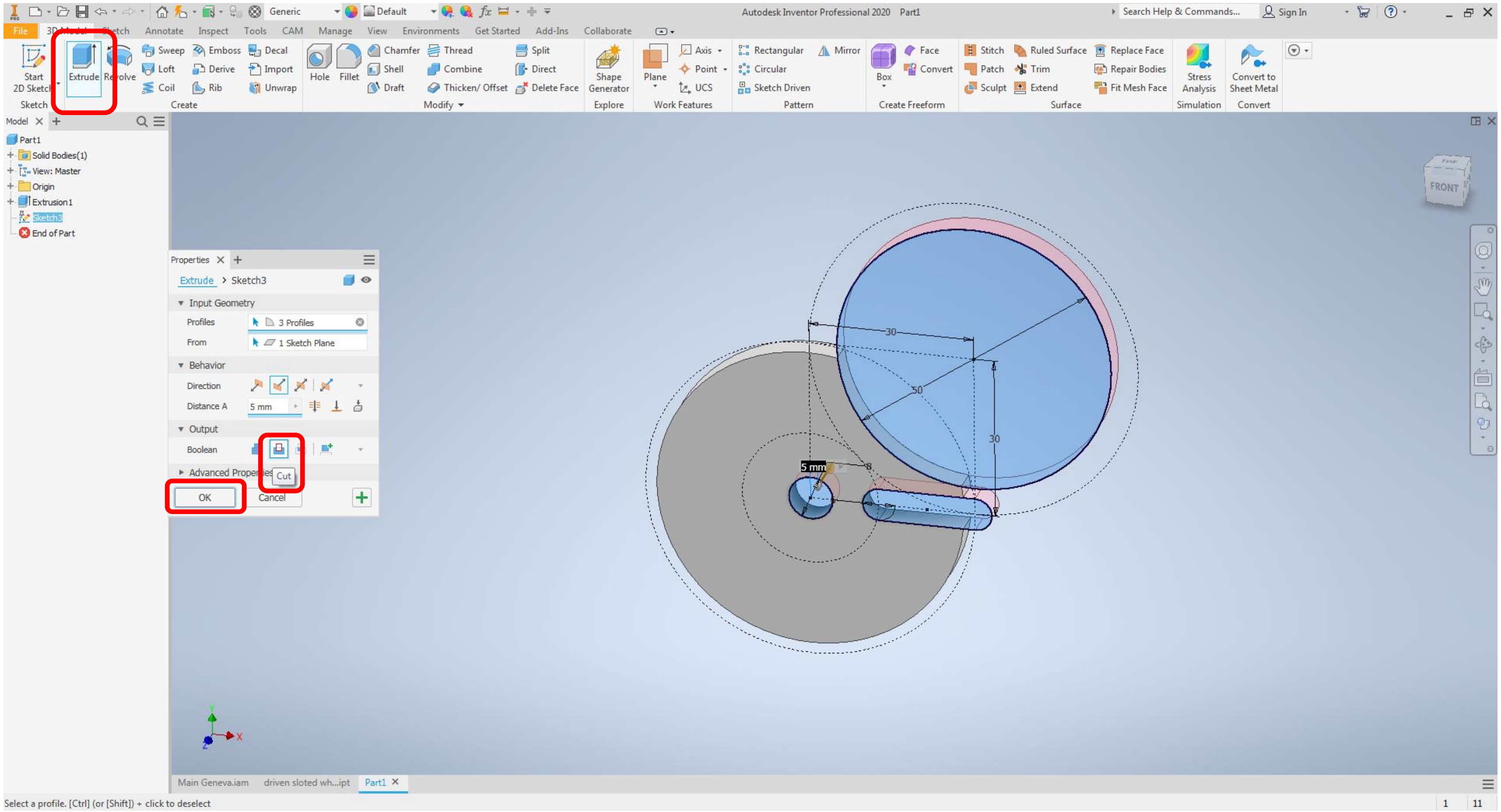
Click on "Rectangle (drop down)" and select "Slot (center to center)"



Click on the intersection of Circle(3) and X axis to set first center of the slot
Click on the right bottom vertex of the square to set the second center
Enter the slot width 6mm and press “Enter” then click “Finish Sketch”

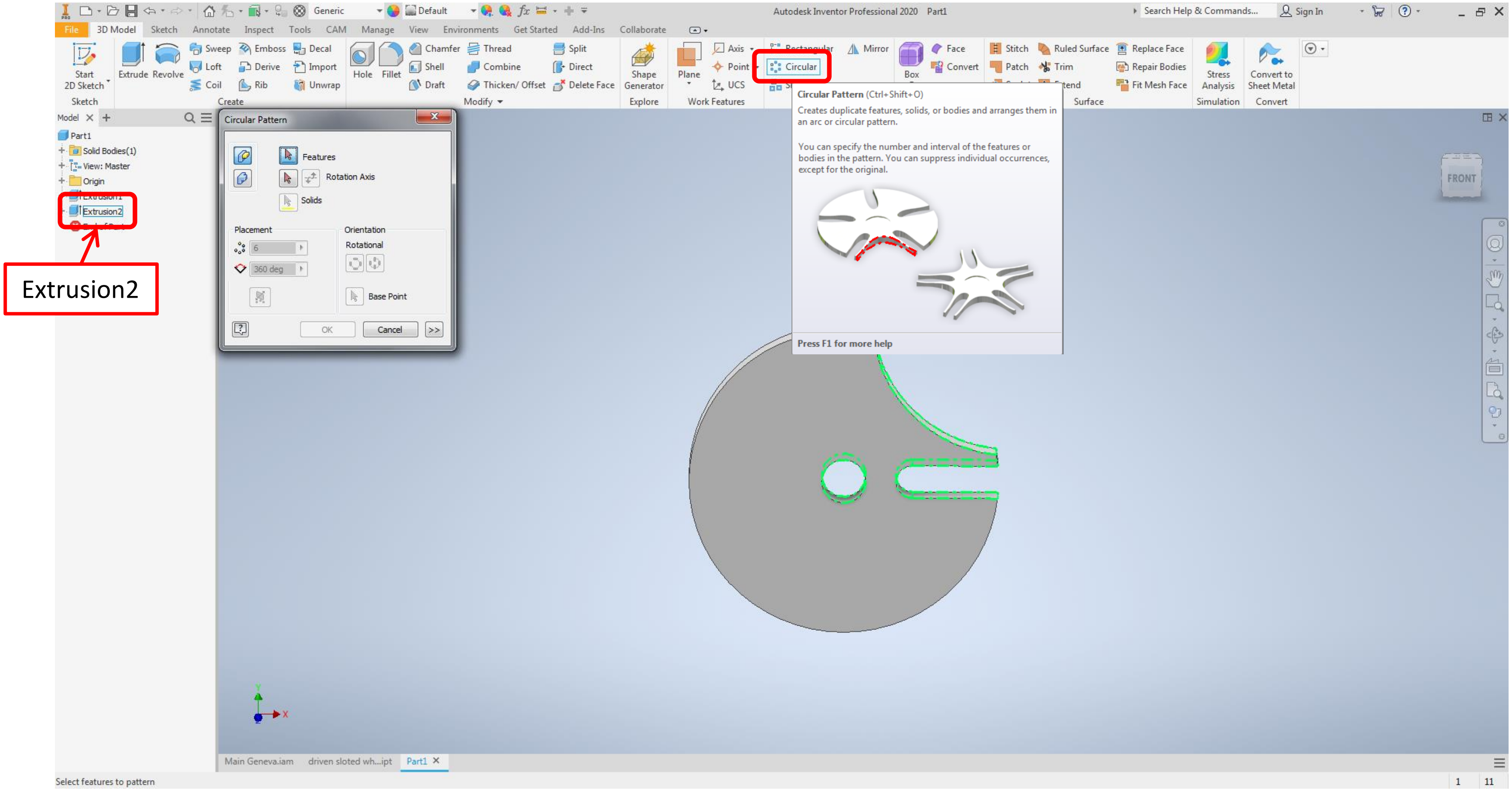


Click on “Extrude” and select “cut” to remove the selected profiles from the existing model and click “OK”

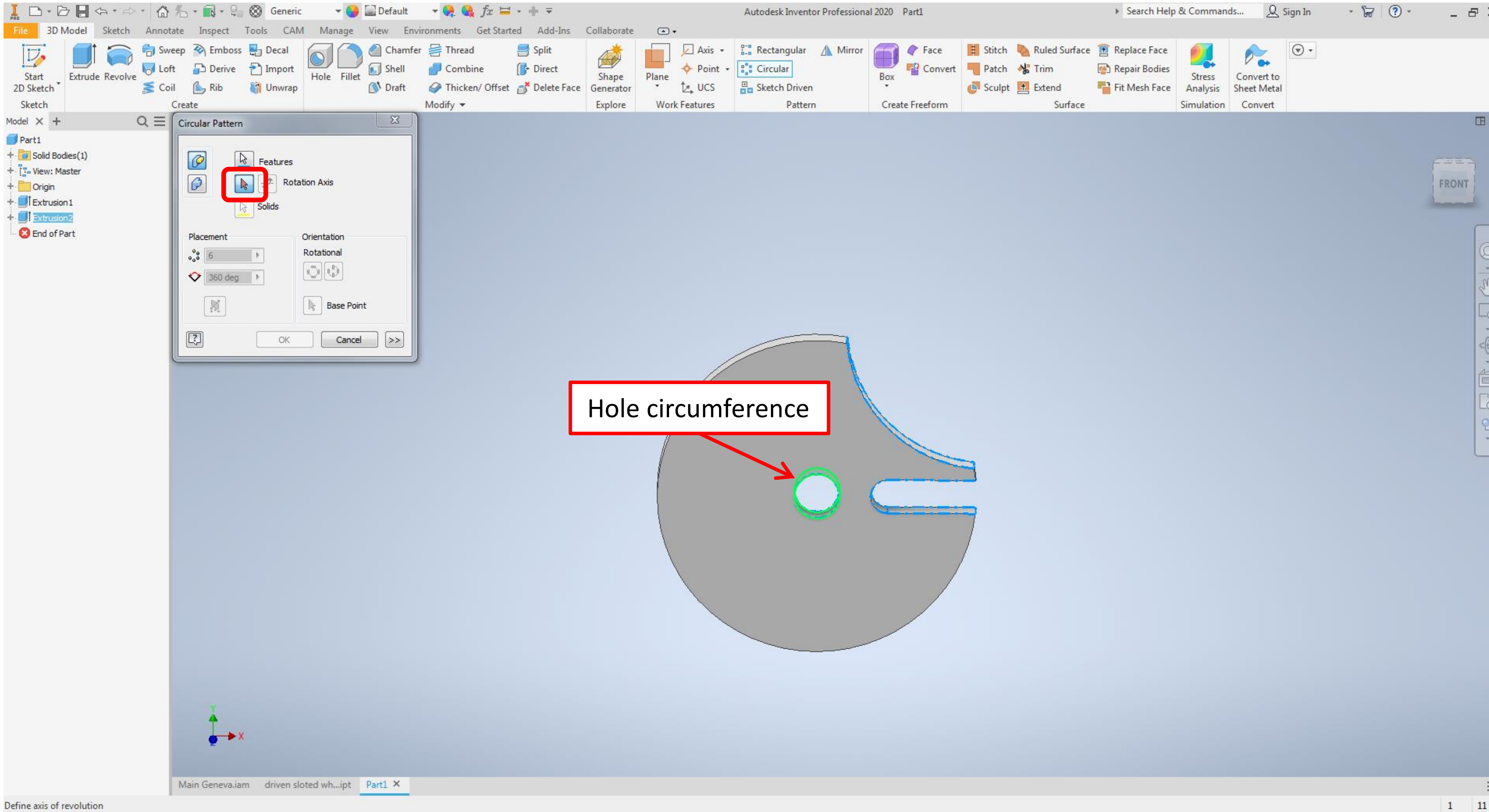


Click on “Circular (Circular pattern)”

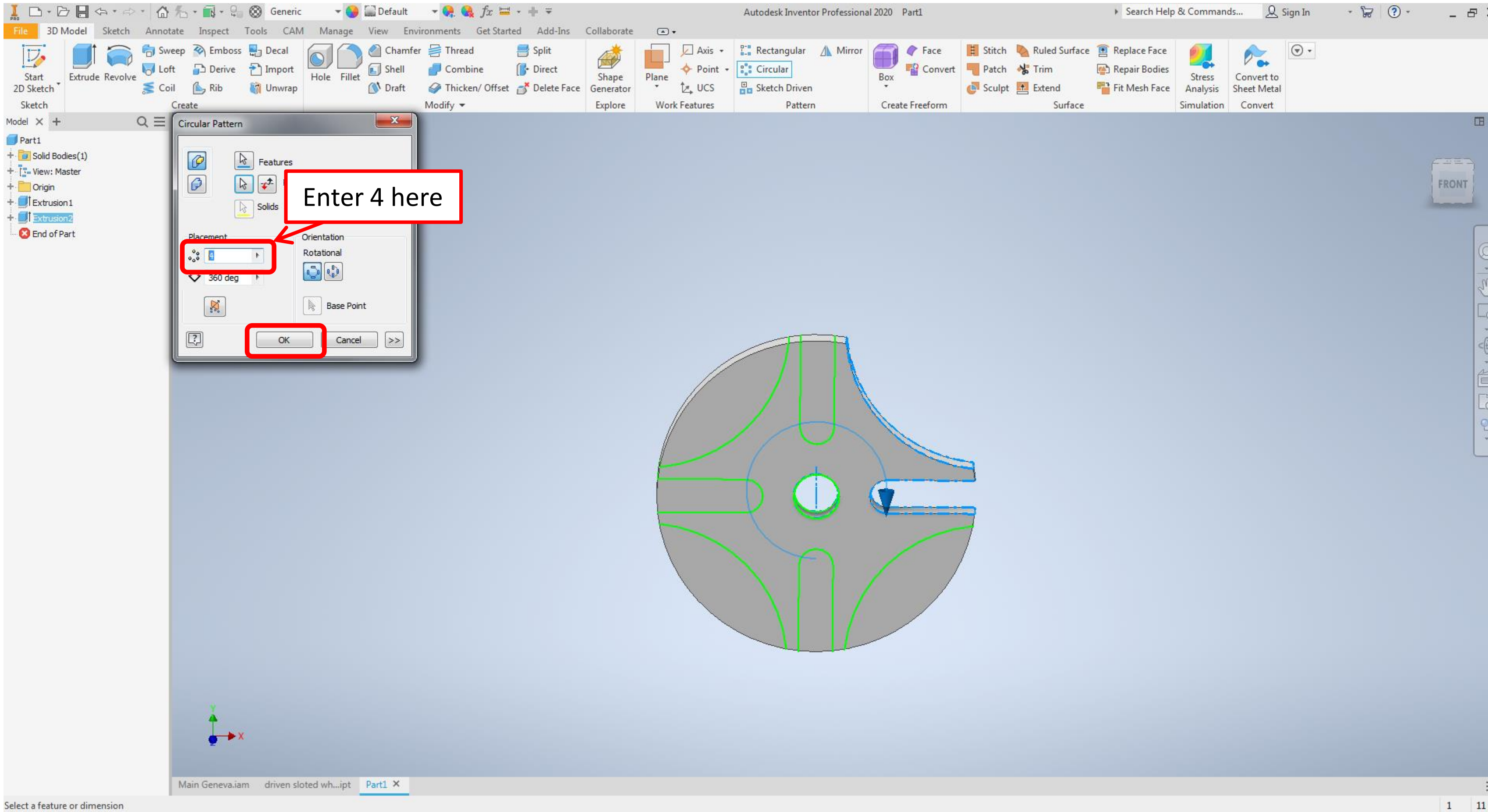
Click on the “Extrusion2” from the model browser and then click “OK”



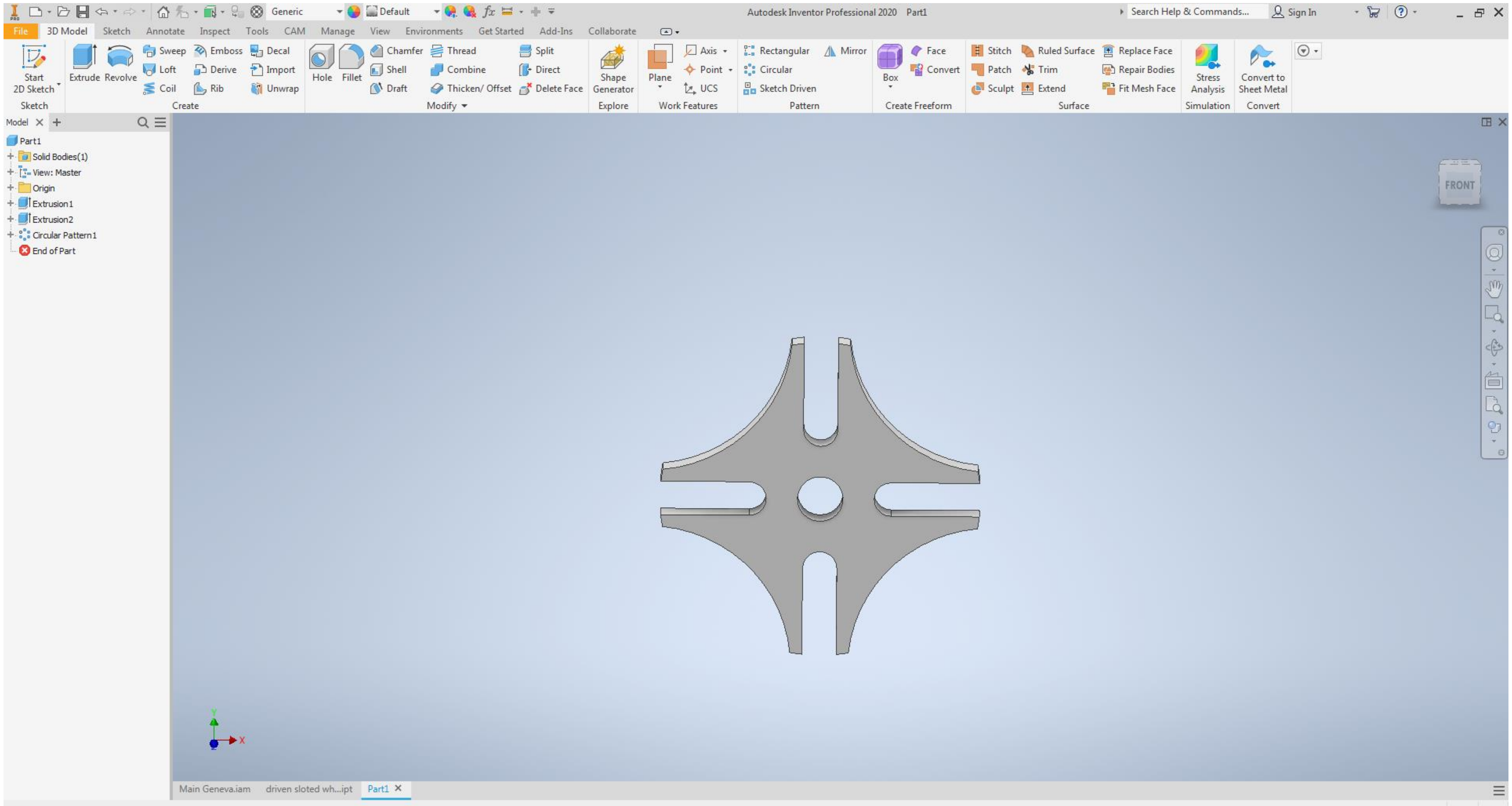
Click on “Rotation Axis” and click on the hole circumference to define rotation axis



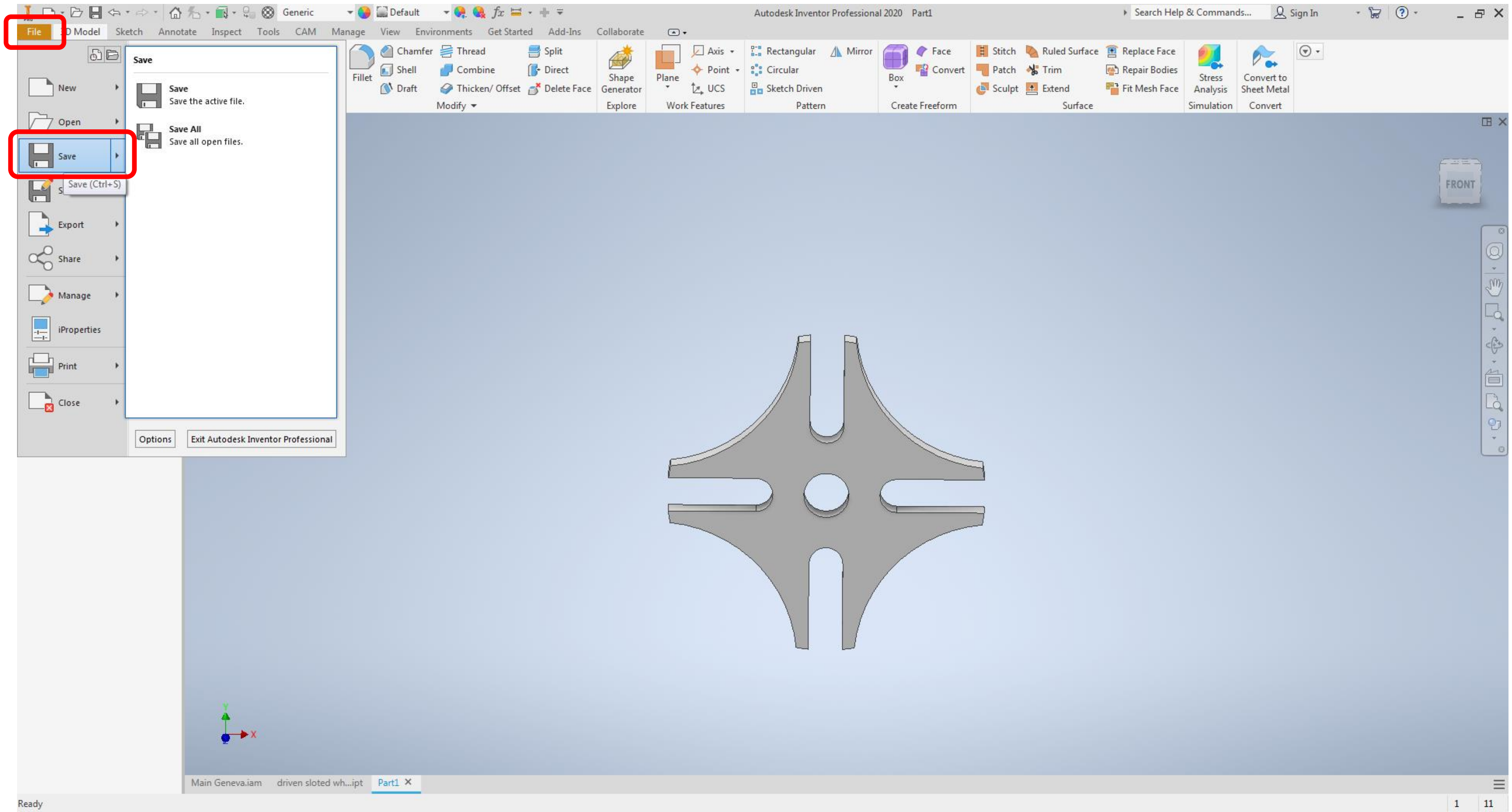
Enter the Occurrence count 4 in the Placement box and click “OK”



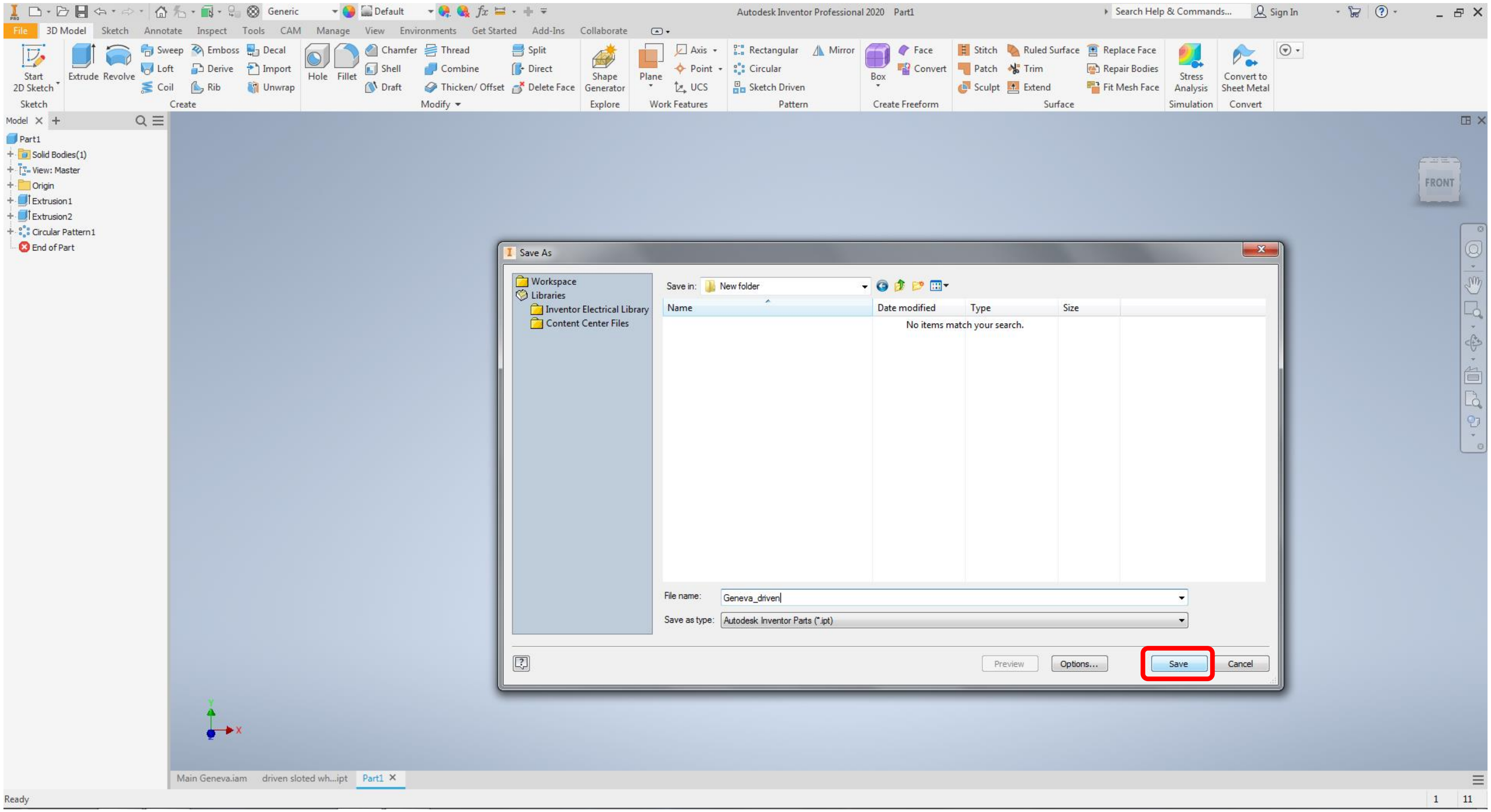
Your model should look like this



Click on “File” and select “Save”

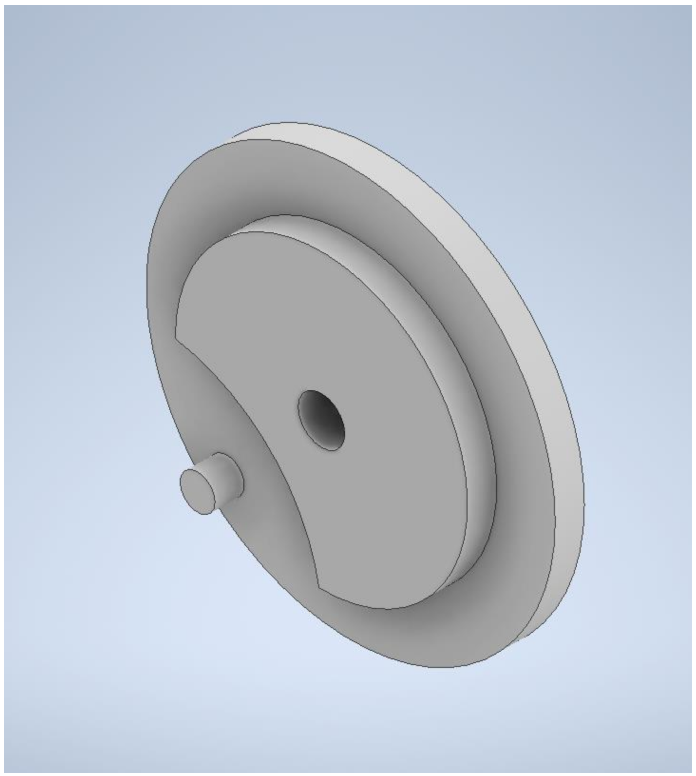


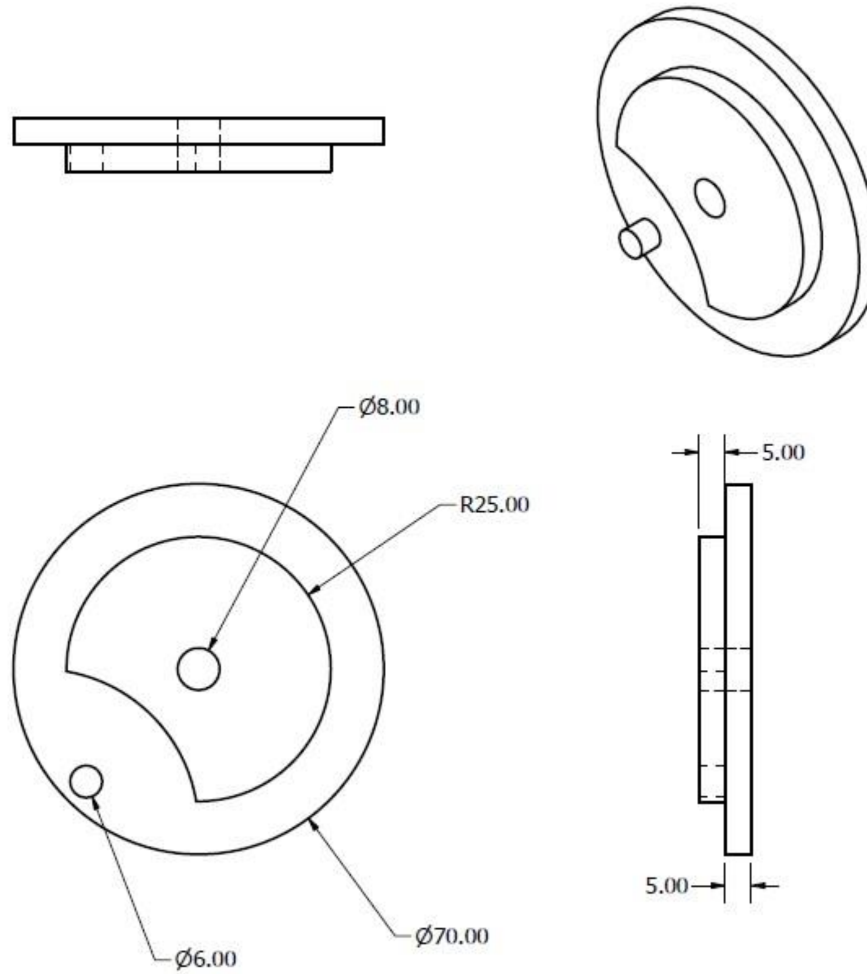
Enter the part name and click “Save”



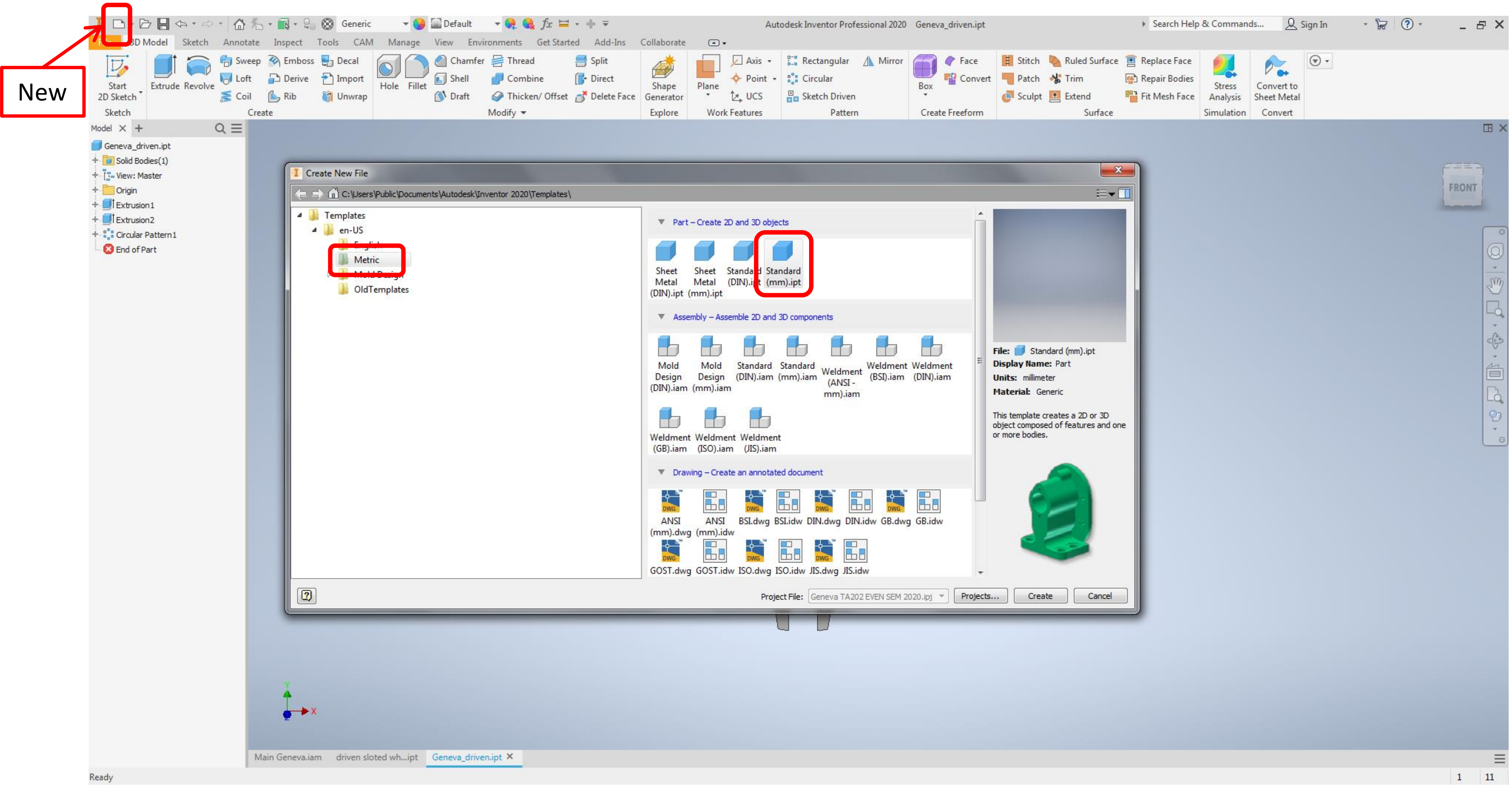
This ends your task 1. Please show your progress to your guide/TA/Tutor. Proceed after.

Task 2
Make a Geneva Driver Wheel

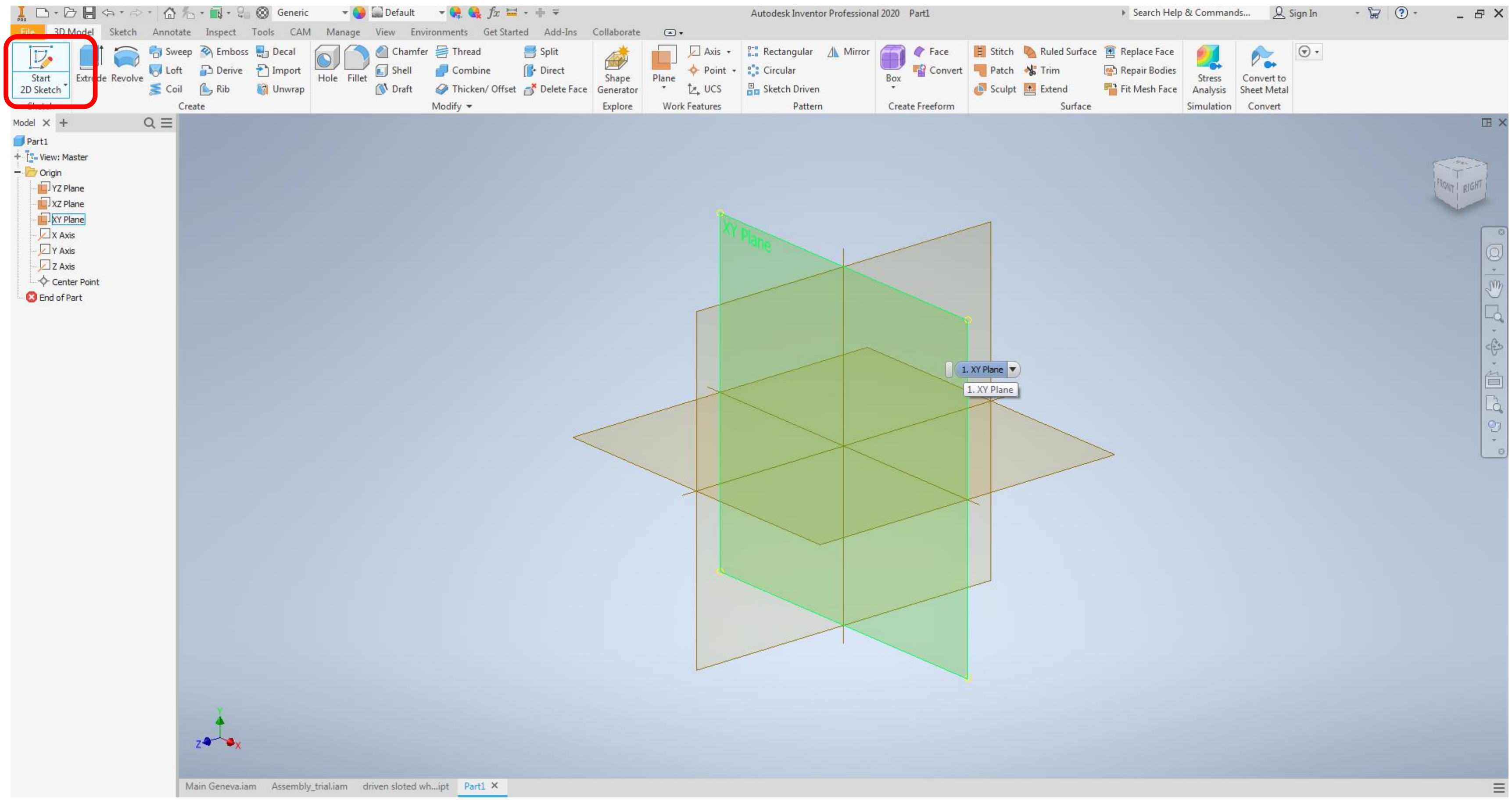




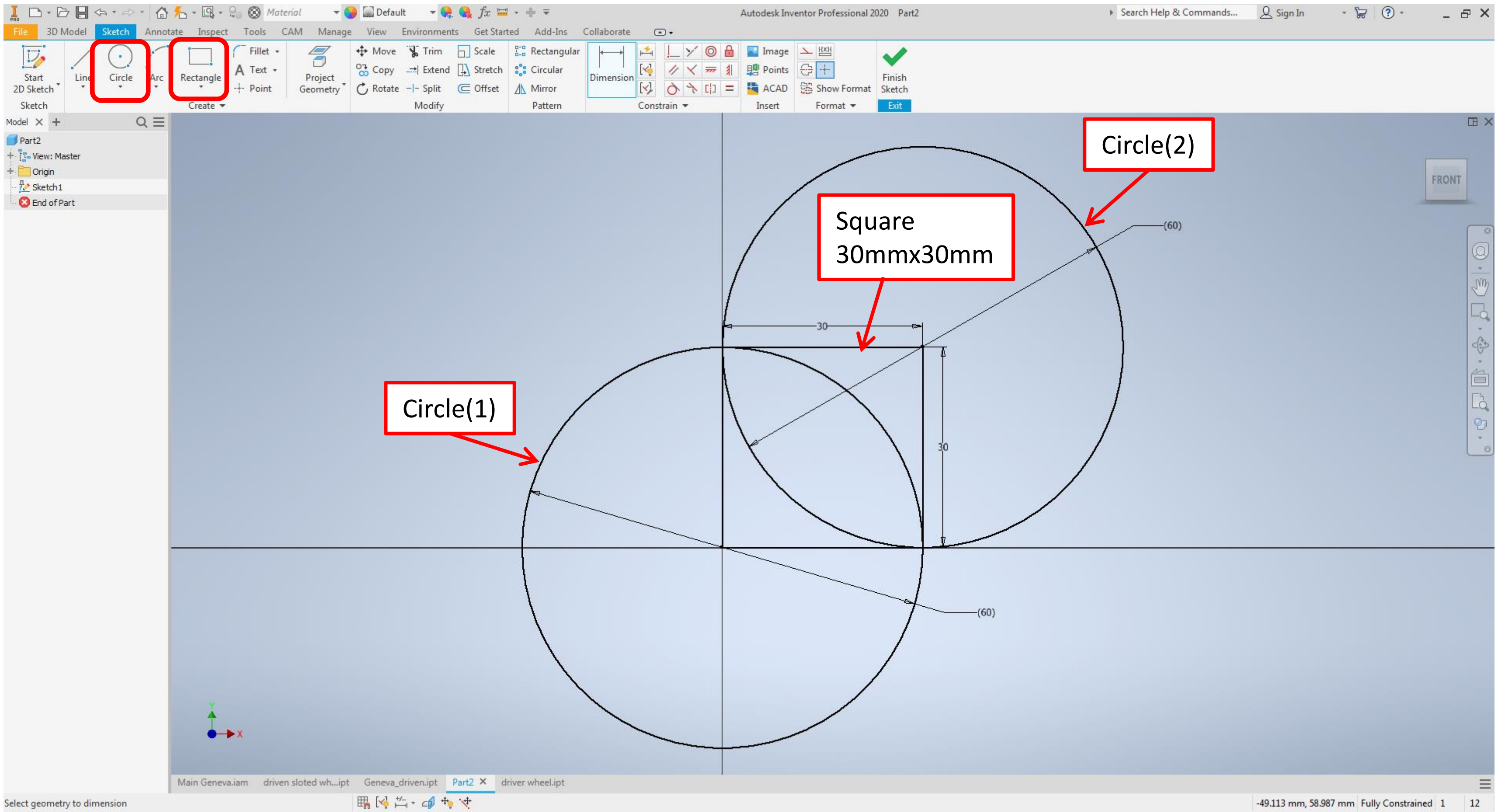
Click on “New” icon, select “Metric” and then select “Standard(mm).ipt”



Click on Start 2D sketch and select the XY plane

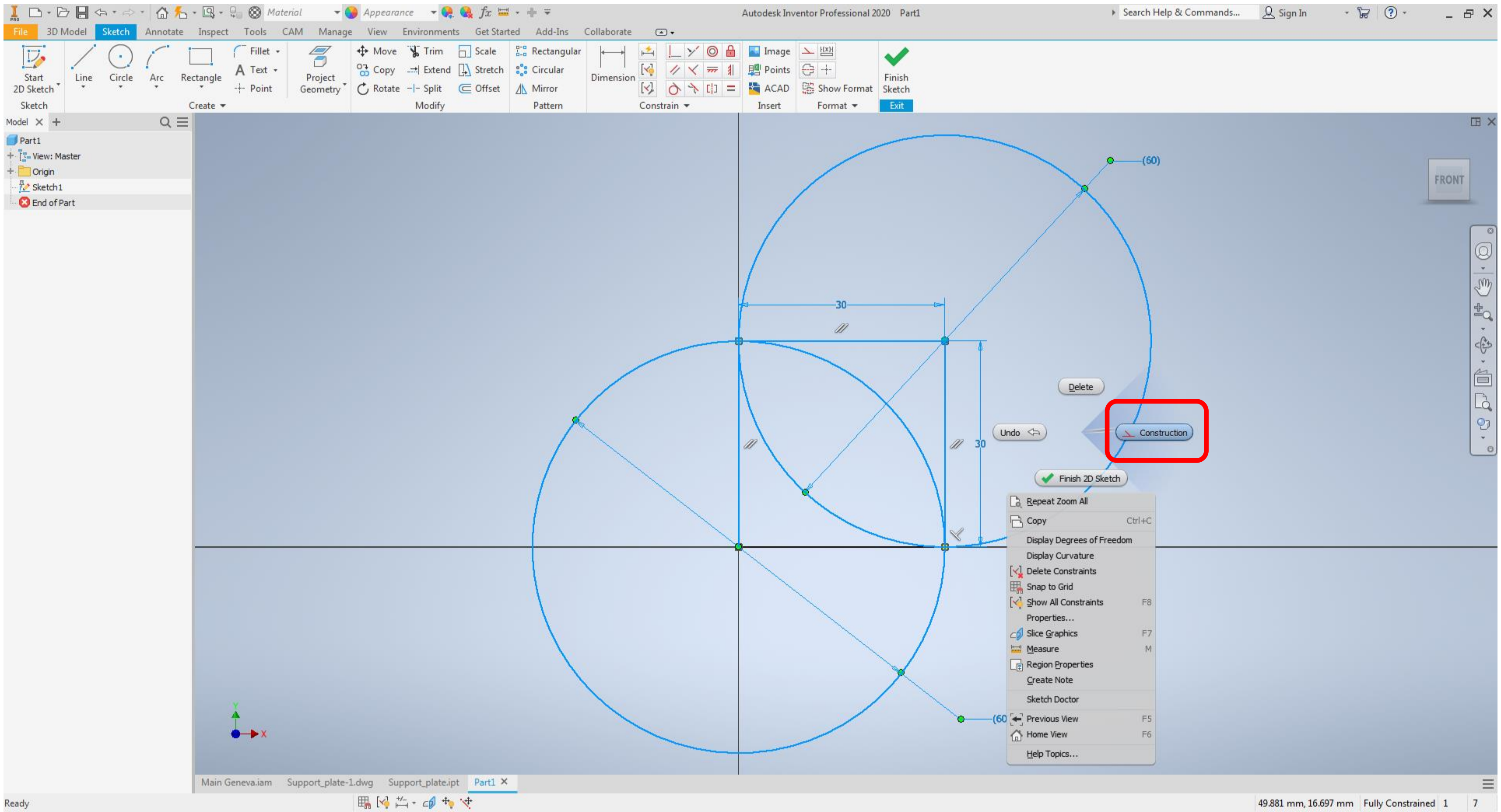


Click on “Rectangle”. With origin as the first point, draw a 30mmx30mm square as shown
Click on “Circle” and draw Circle(1) of diameter 60mm with origin as the center
Draw Circle(2) of diameter 60mm with top right corner of the square as the center

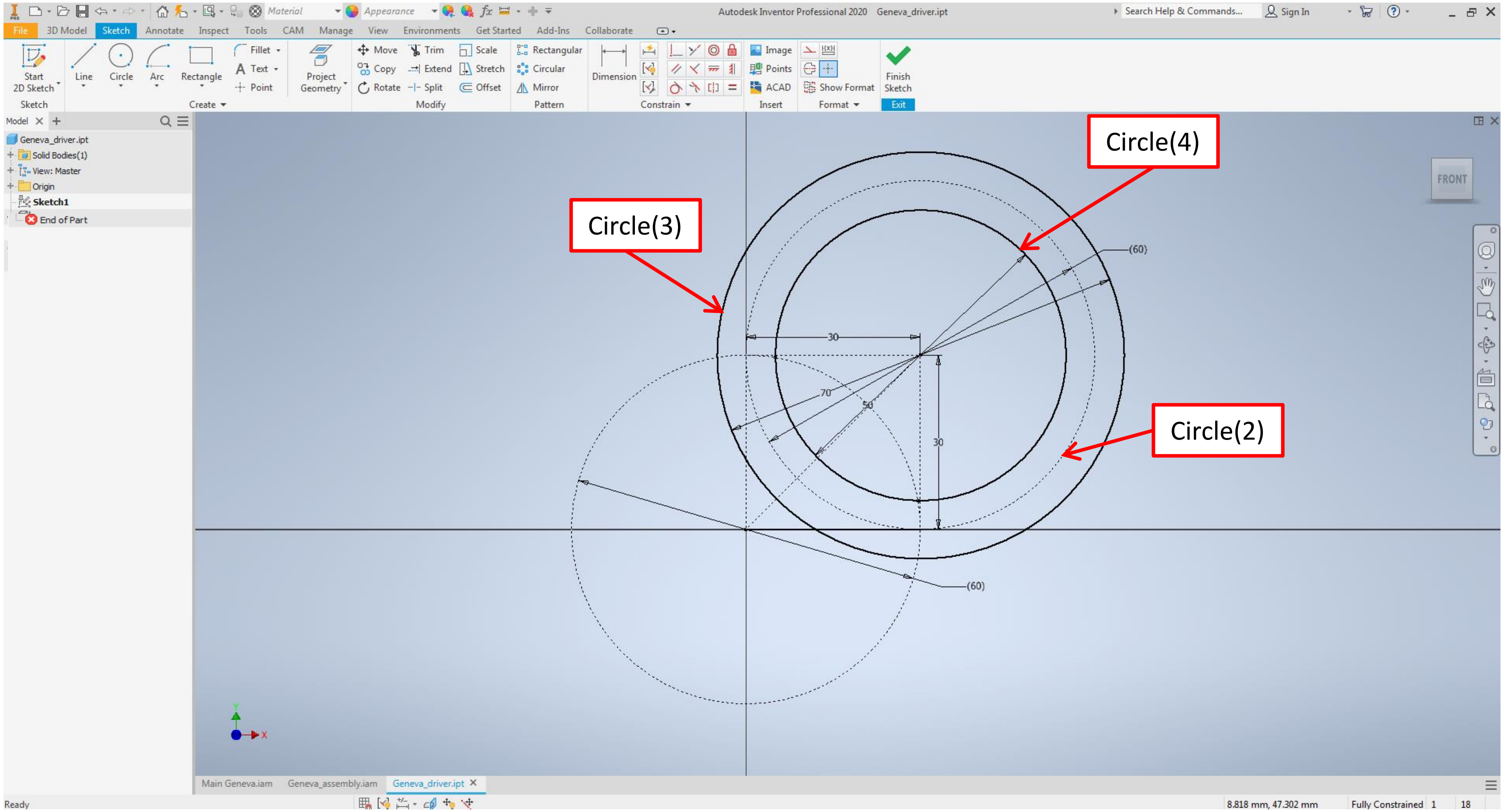


Left click on your mouse and drag towards left/right to select all the geometries(Square, Circle(1) and Circle(2))

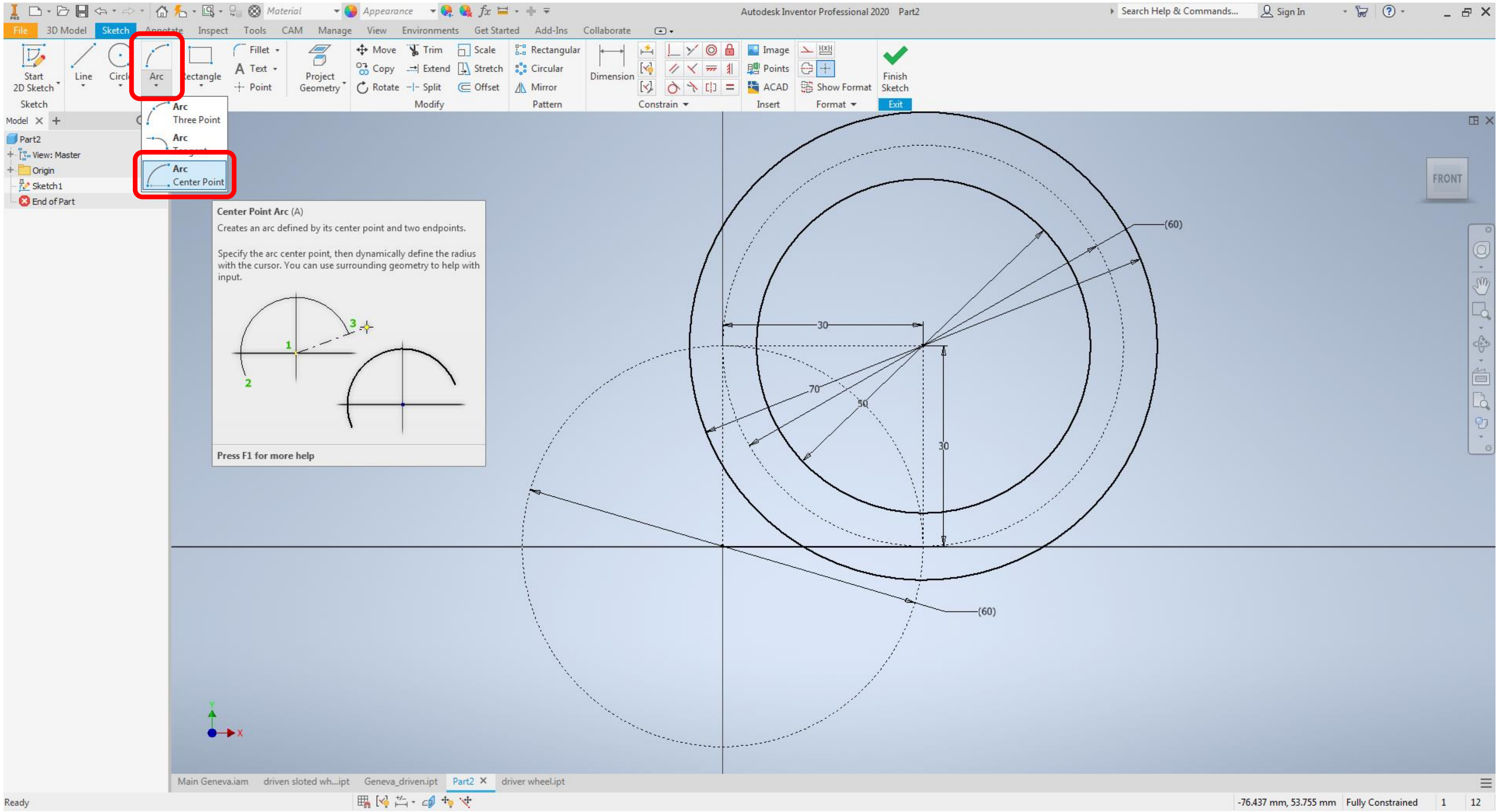
Right click and select “Construction” to convert the selected geometries into construction lines



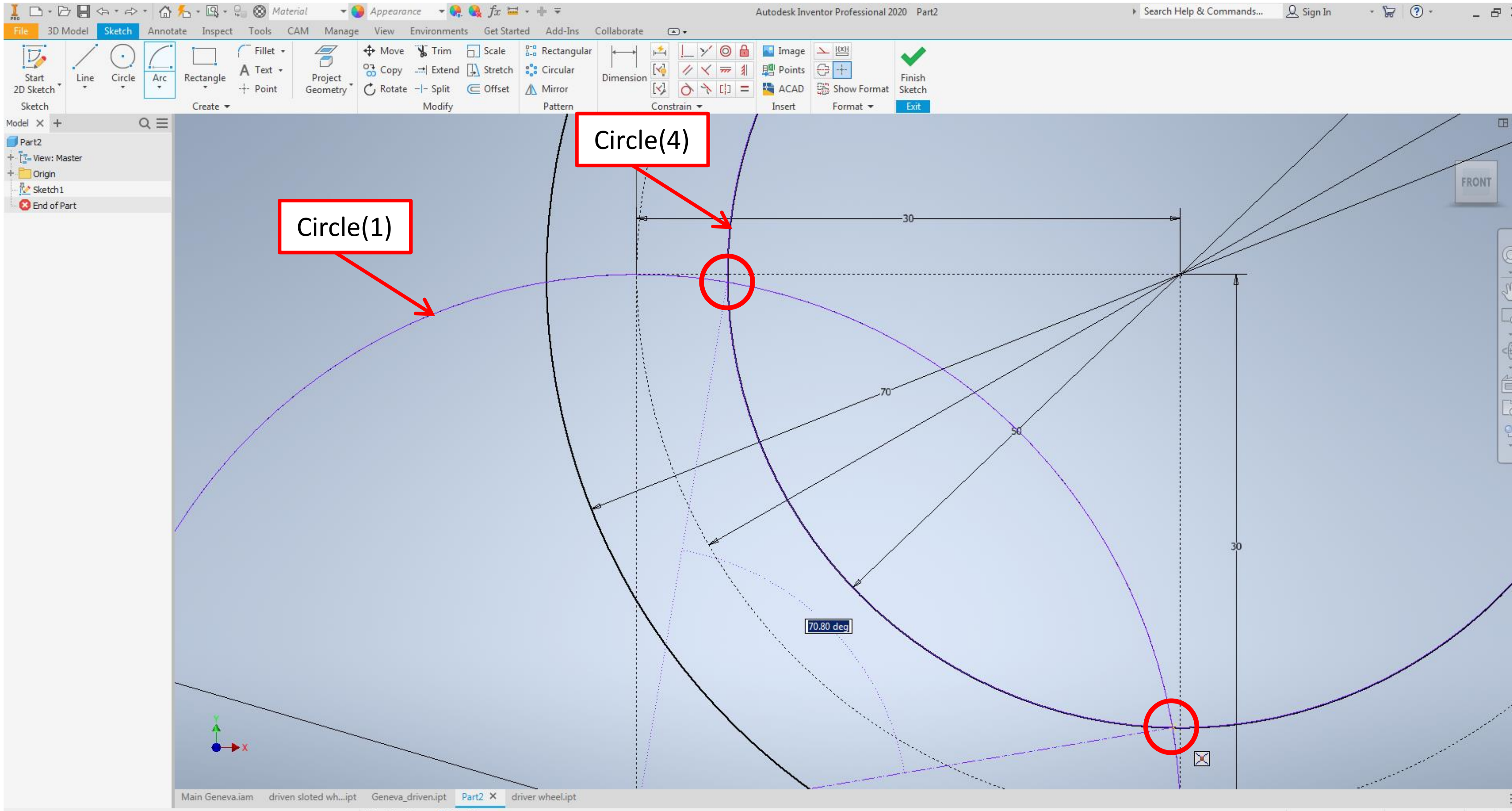
Create Circle(3) of diameter 70mm and Circle(4) of diameter 50mm, both concentric to Circle(2)



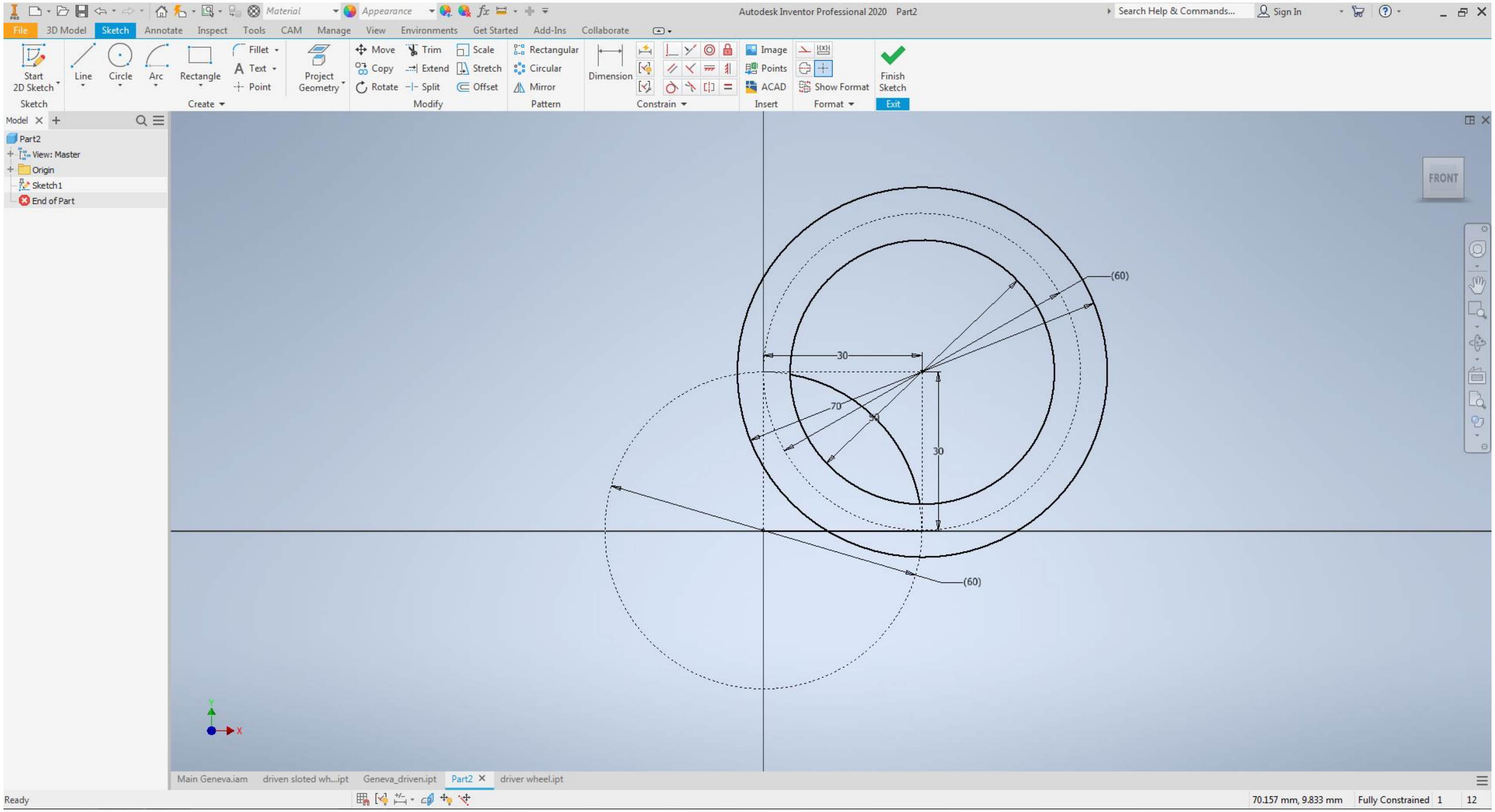
Click on “Arc (drop down)” and select “Arc - Center Point”



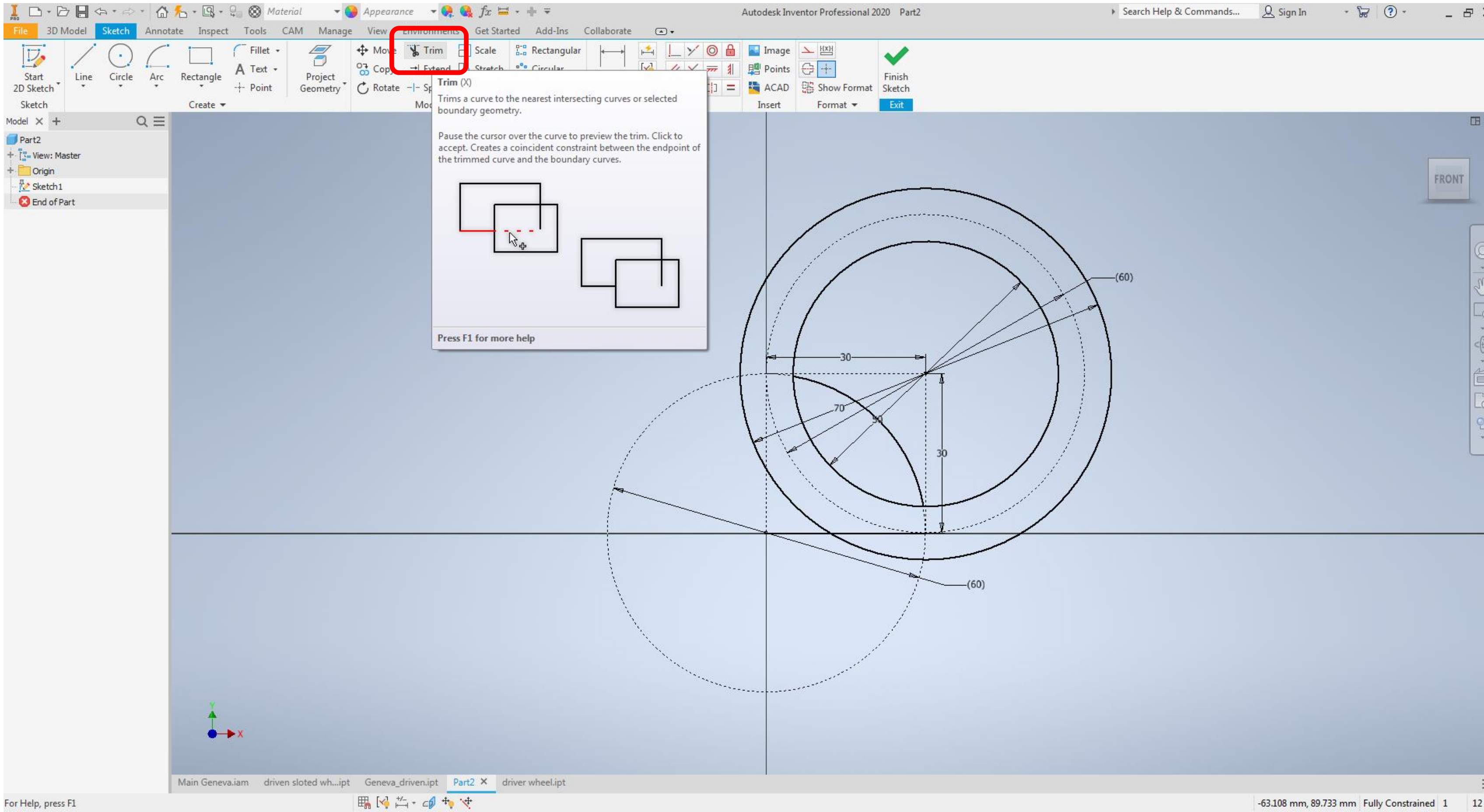
Select the origin as the center and draw an arc by clicking on the intersection points between Circle(1) and Circle(4)



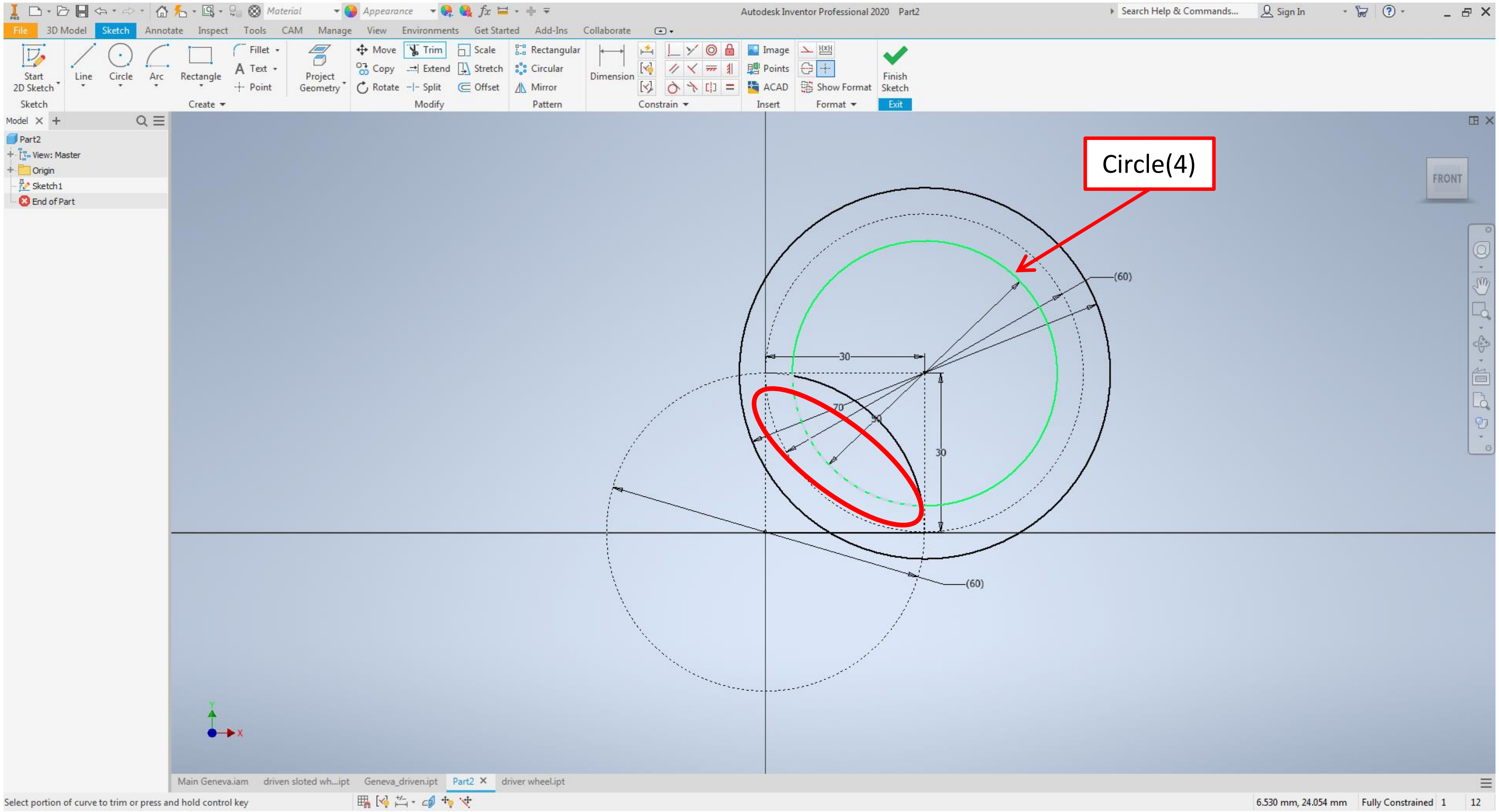
Your drawing should look like this now



Click on “Trim”

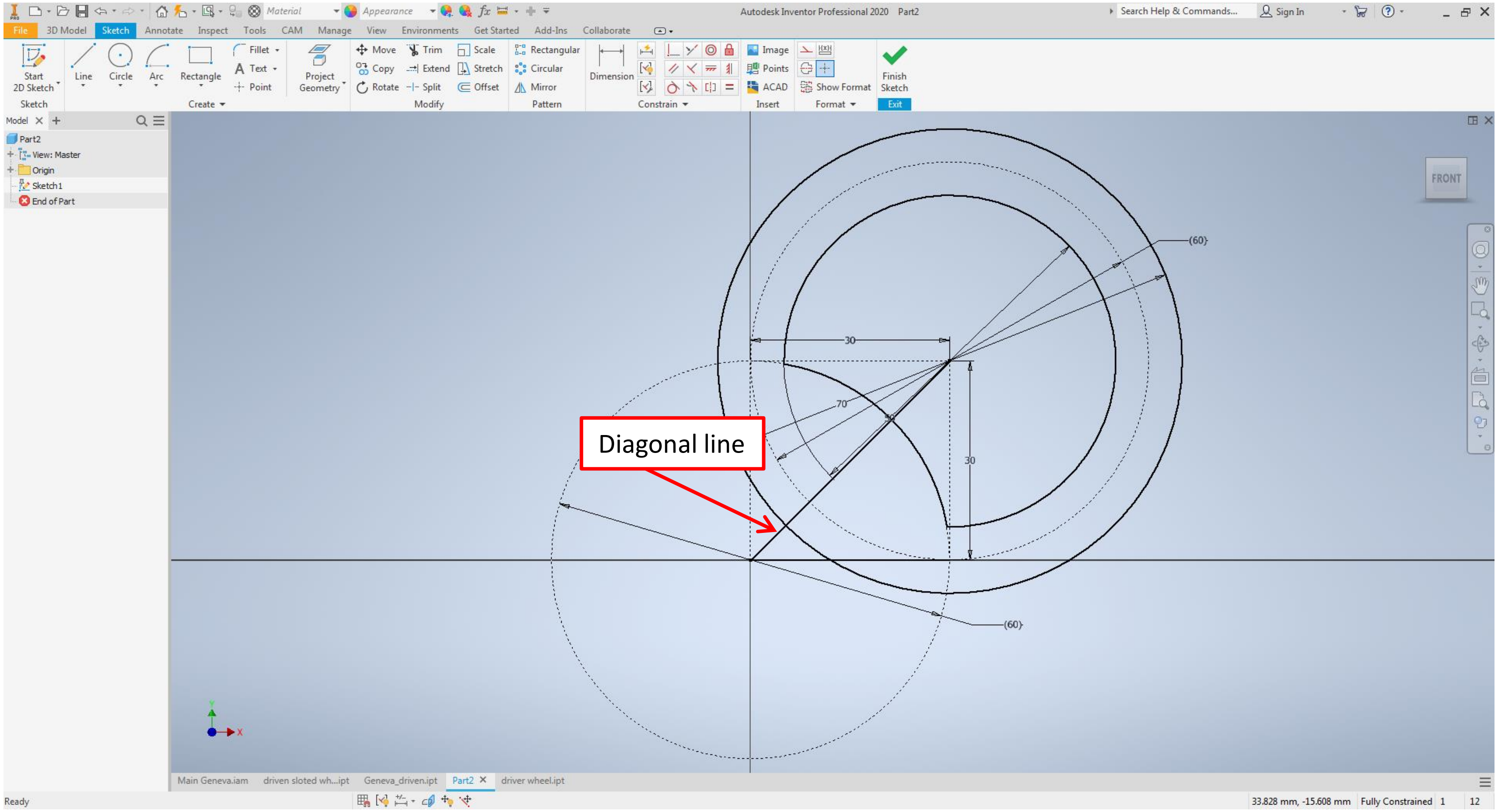


Click on the smaller circumference of circle(4) that is intersected by the Arc

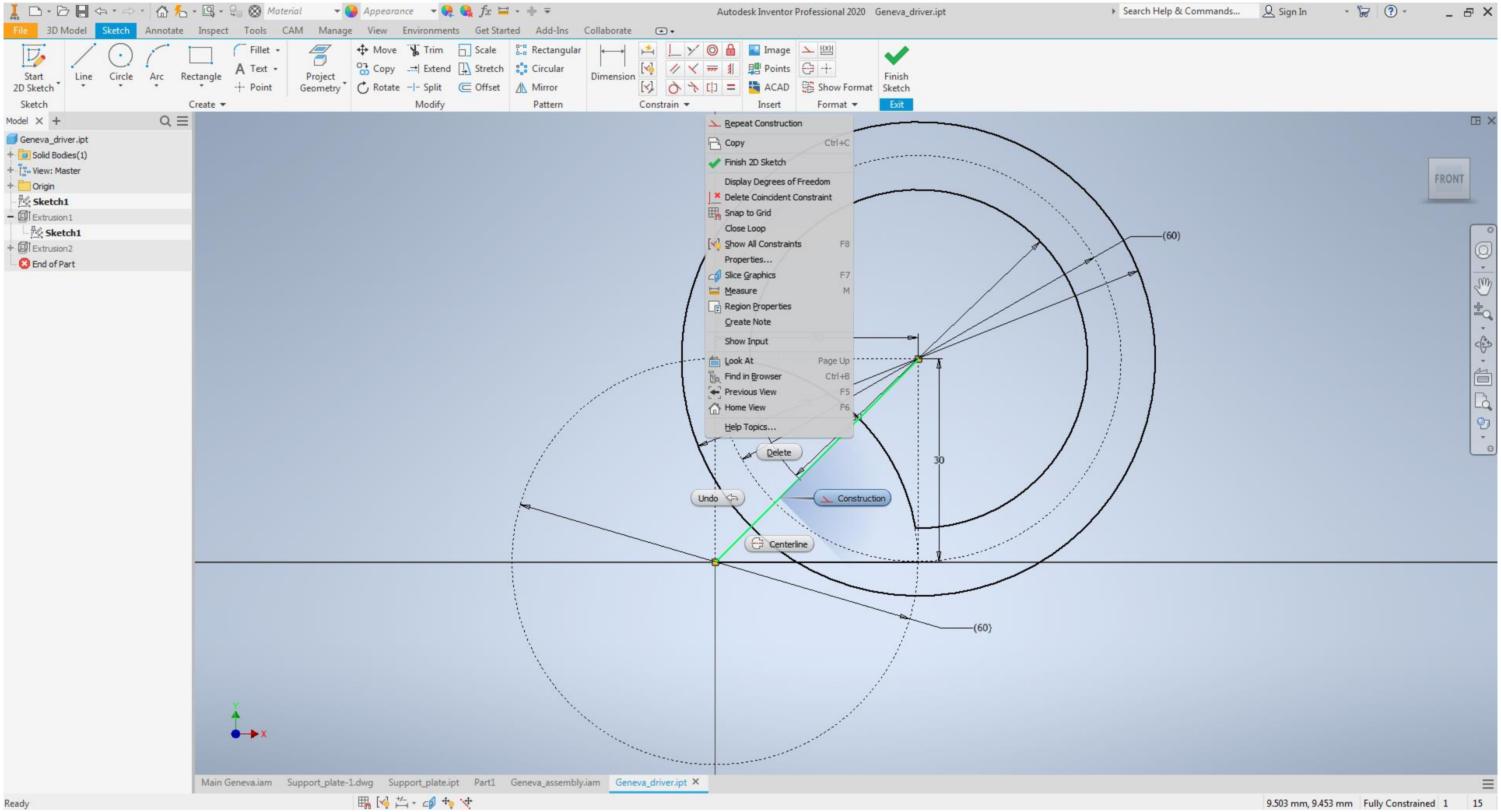


The part clicked on will be trimmed as shown below

Click on "Line" and draw a diagonal line between the top right and bottom left vertices of the square



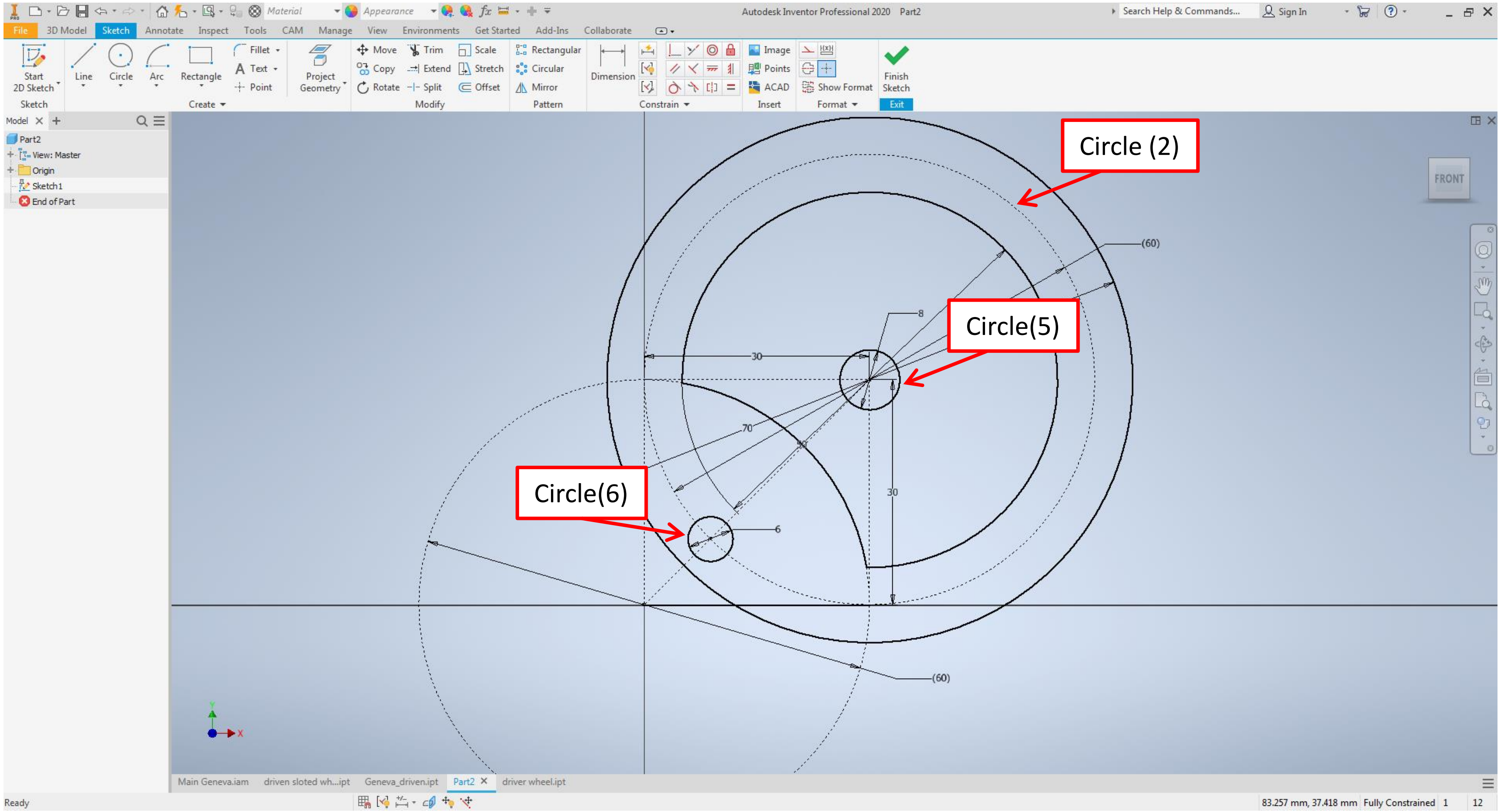
Right click on the line and select "Construction" to change the line to Construction line



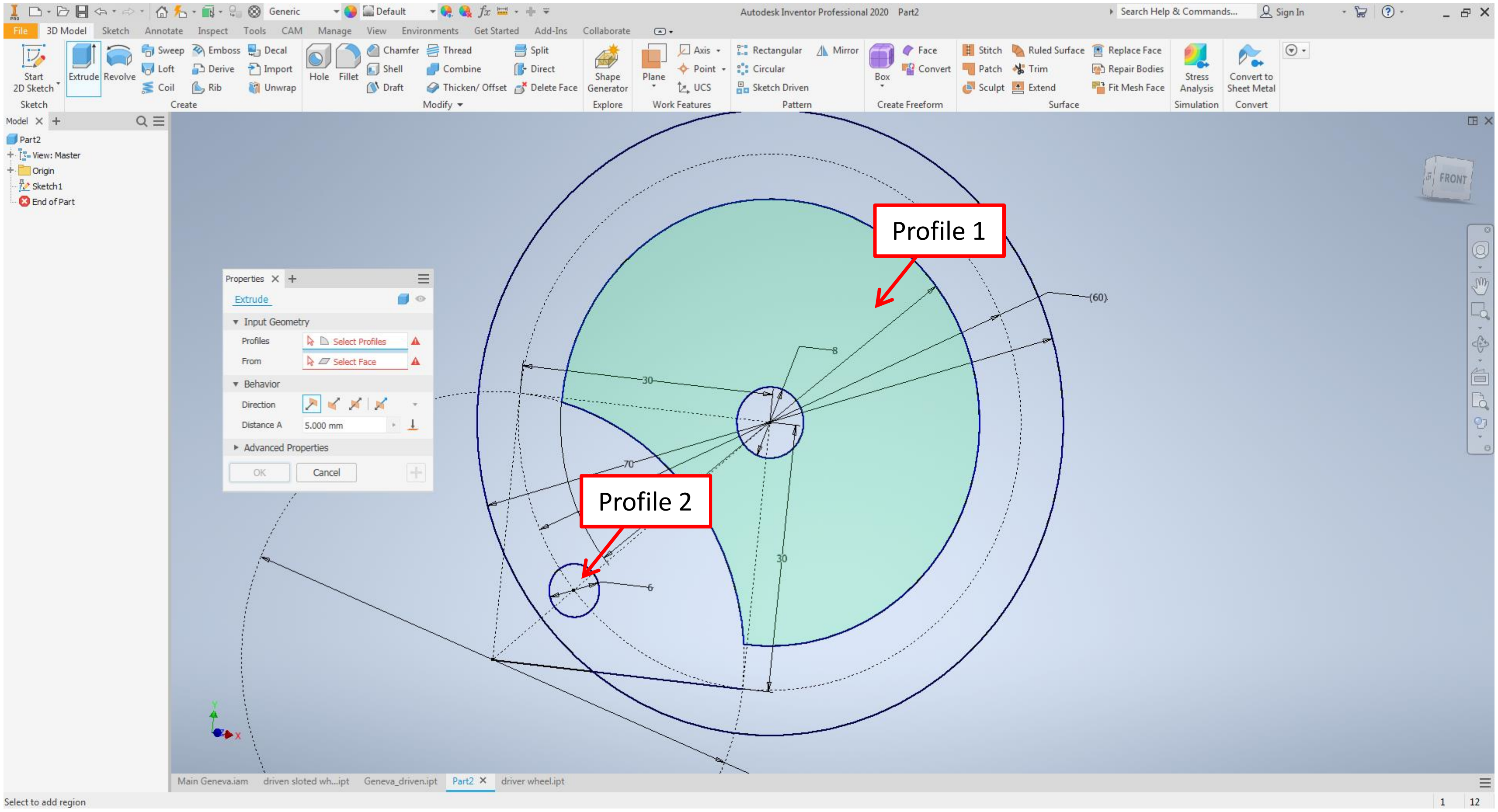
Click on “Circle” and draw a circle (5) of diameter 8mm concentric to circle(2)

Draw circle(6) of diameter 6mm with center at the point of intersection of circle(2) and diagonal of the square

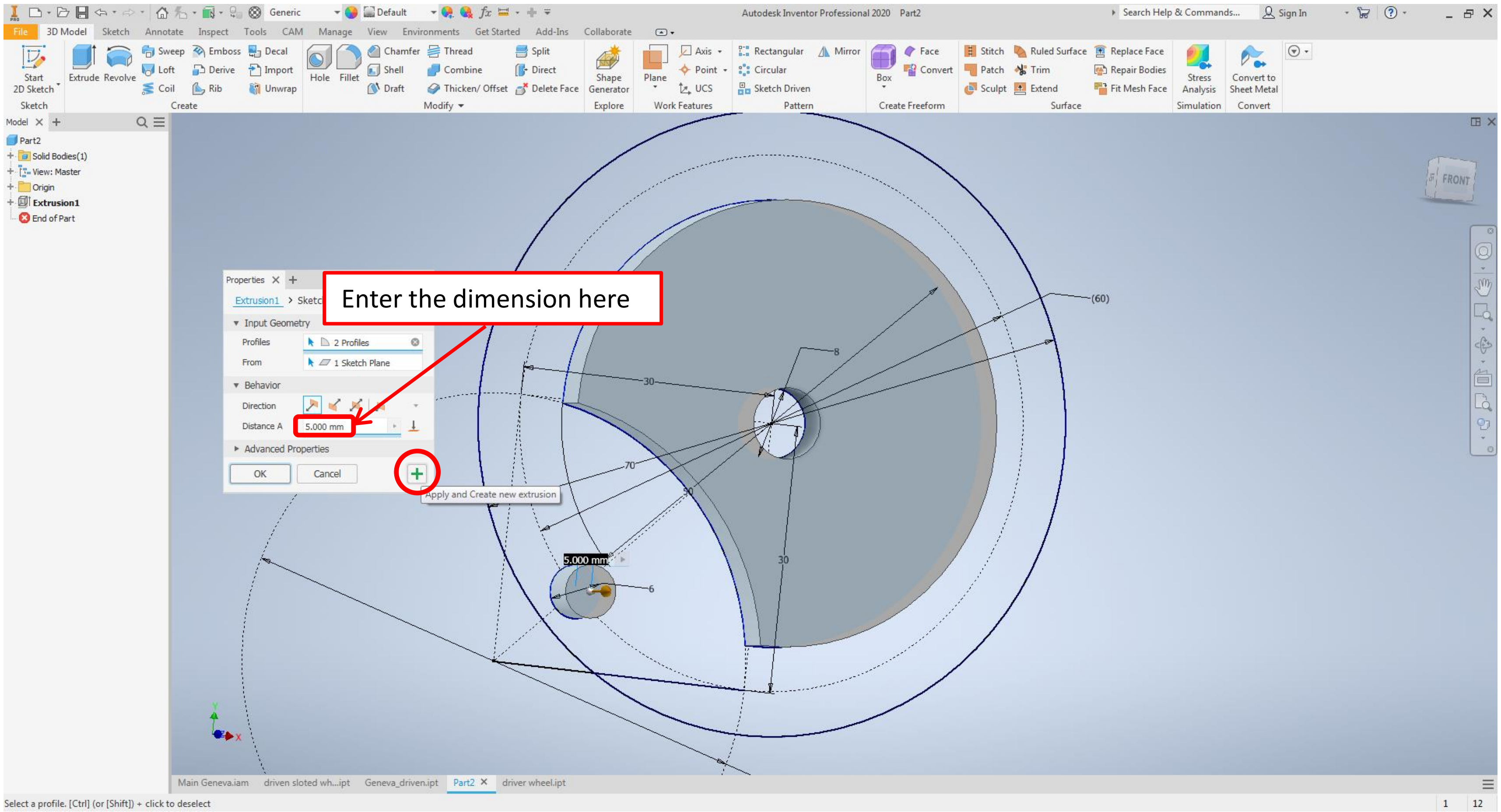
Click on “Finish Sketch”



Click on "Extrude" and select the crescent profile and the off-centered circle

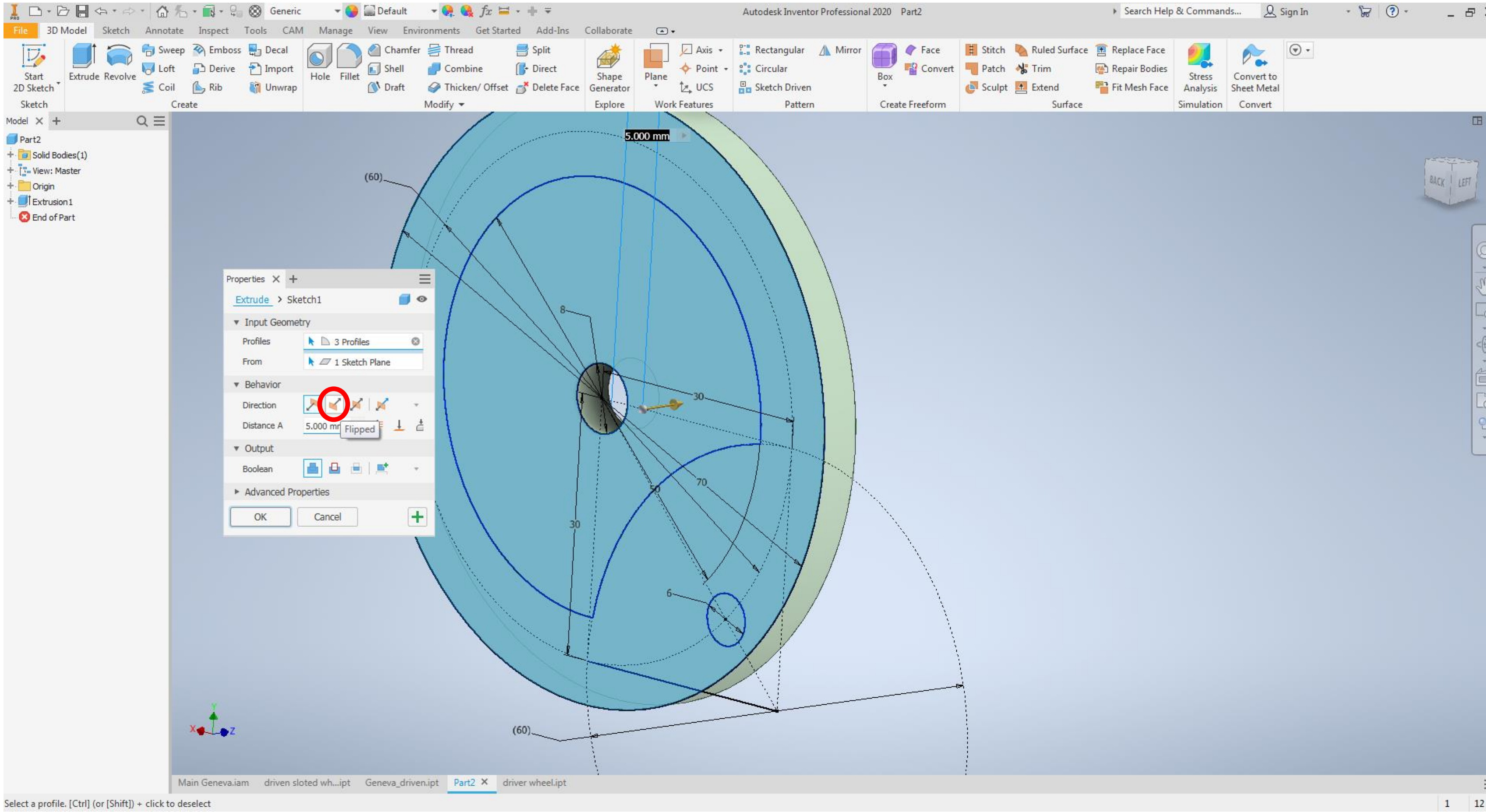


Enter the Extrusion distance as 5mm and click on "**+** (apply and create new extrusion)"

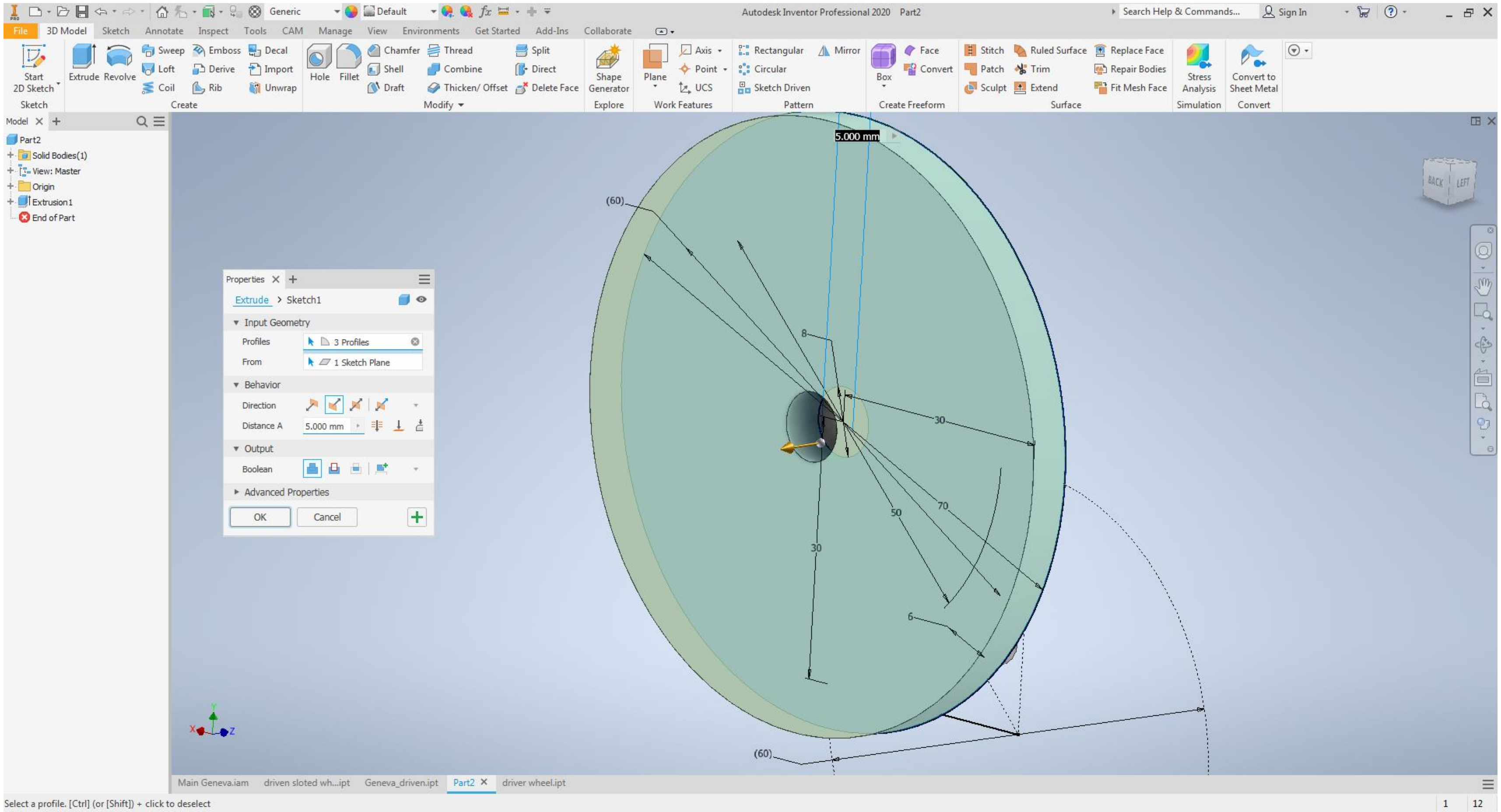


Your first side will be extruded

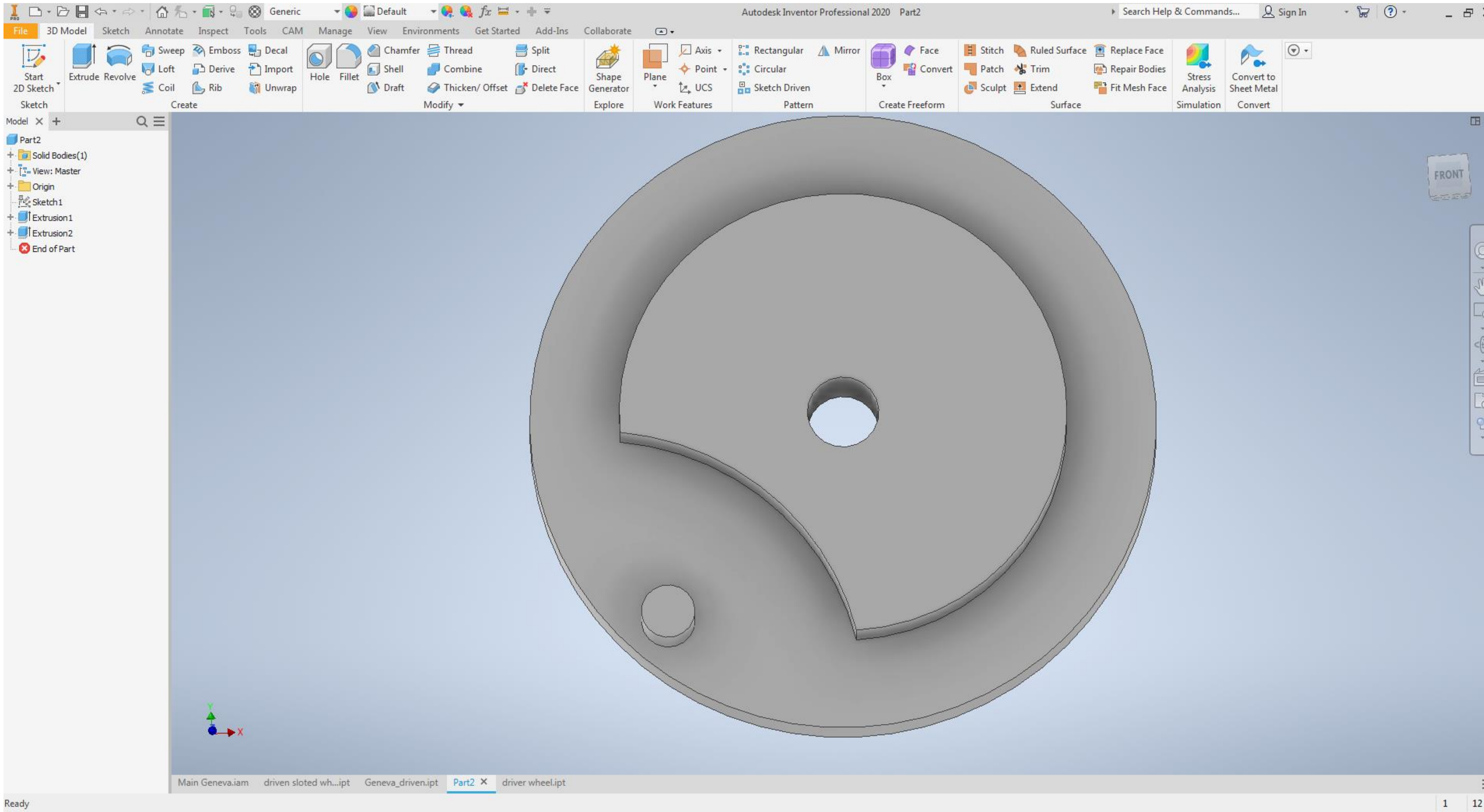
Now select the entire profile except the central hole and click on “Flipped”



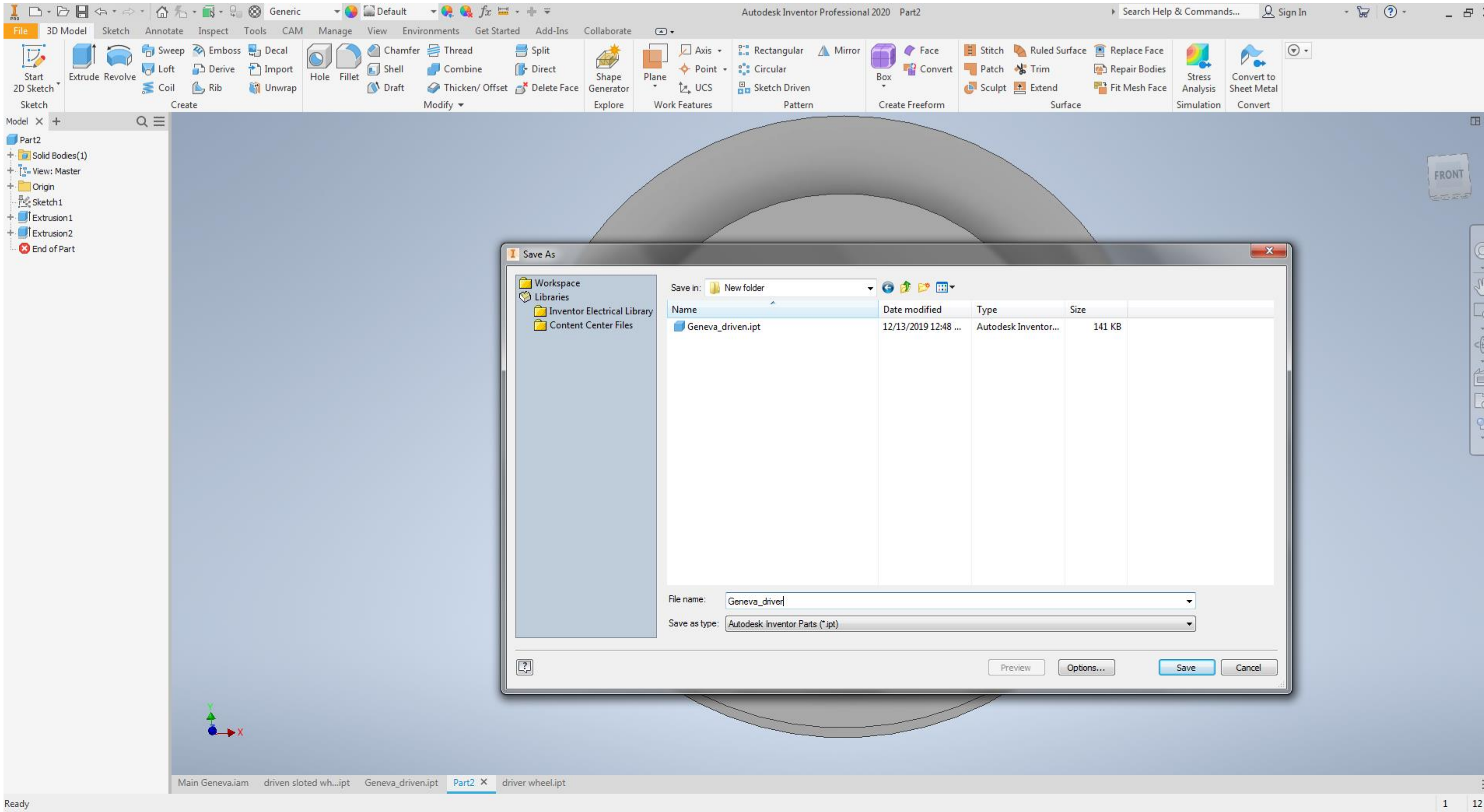
Your new extrusion will change the direction of extrusion
Enter the extrusion distance as 5mm and click “OK”



Your part should look like this

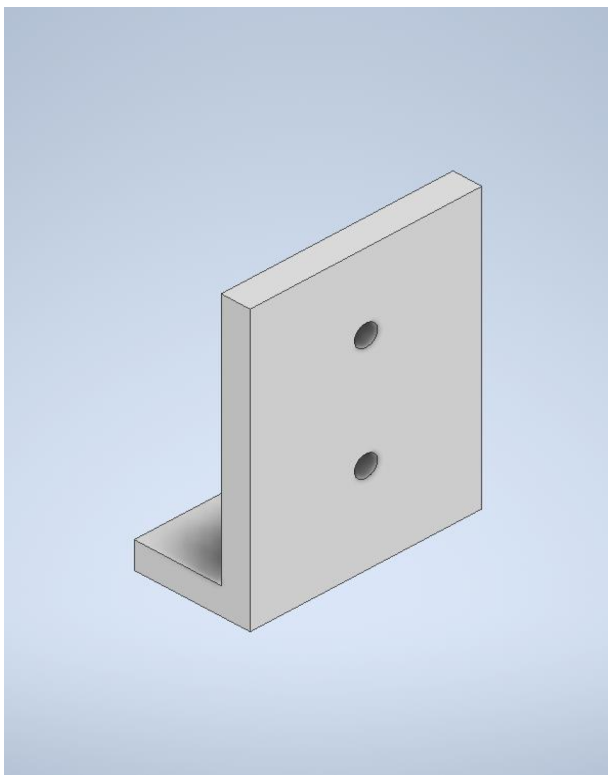


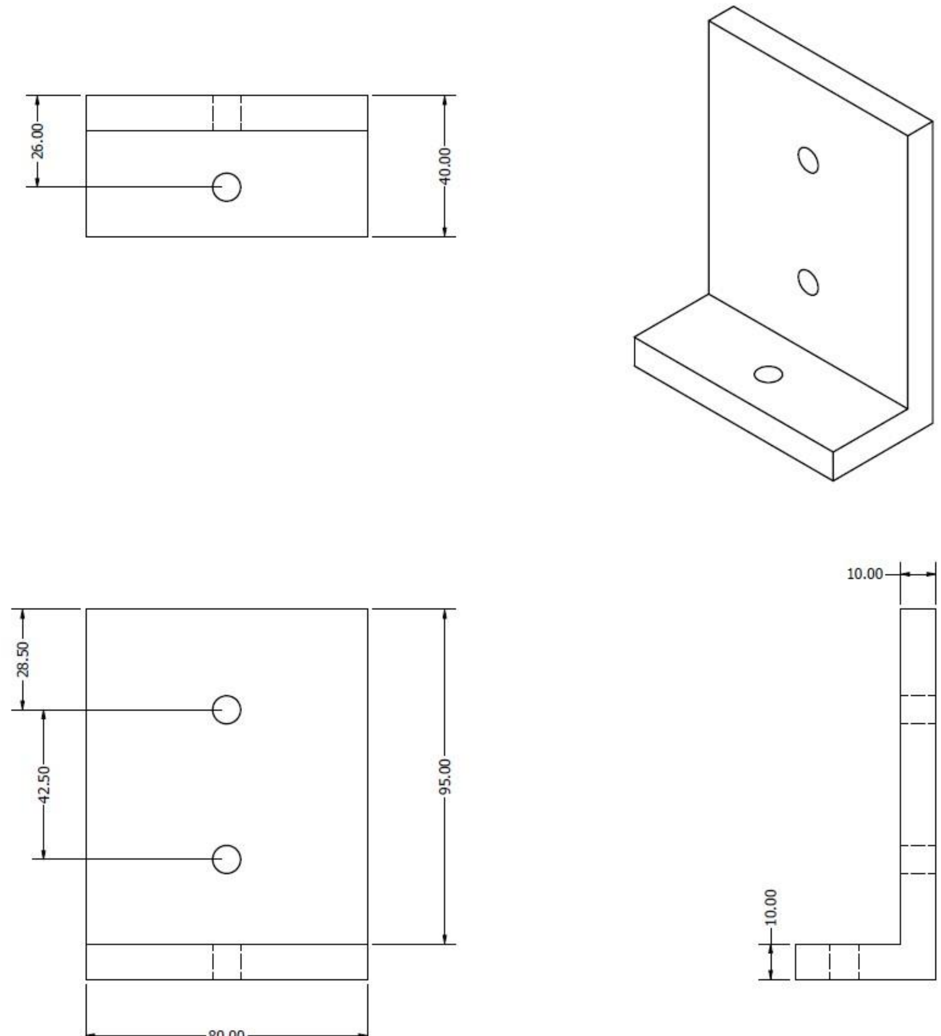
Click “File” and then click “Save”
Enter the part name and click “Save”



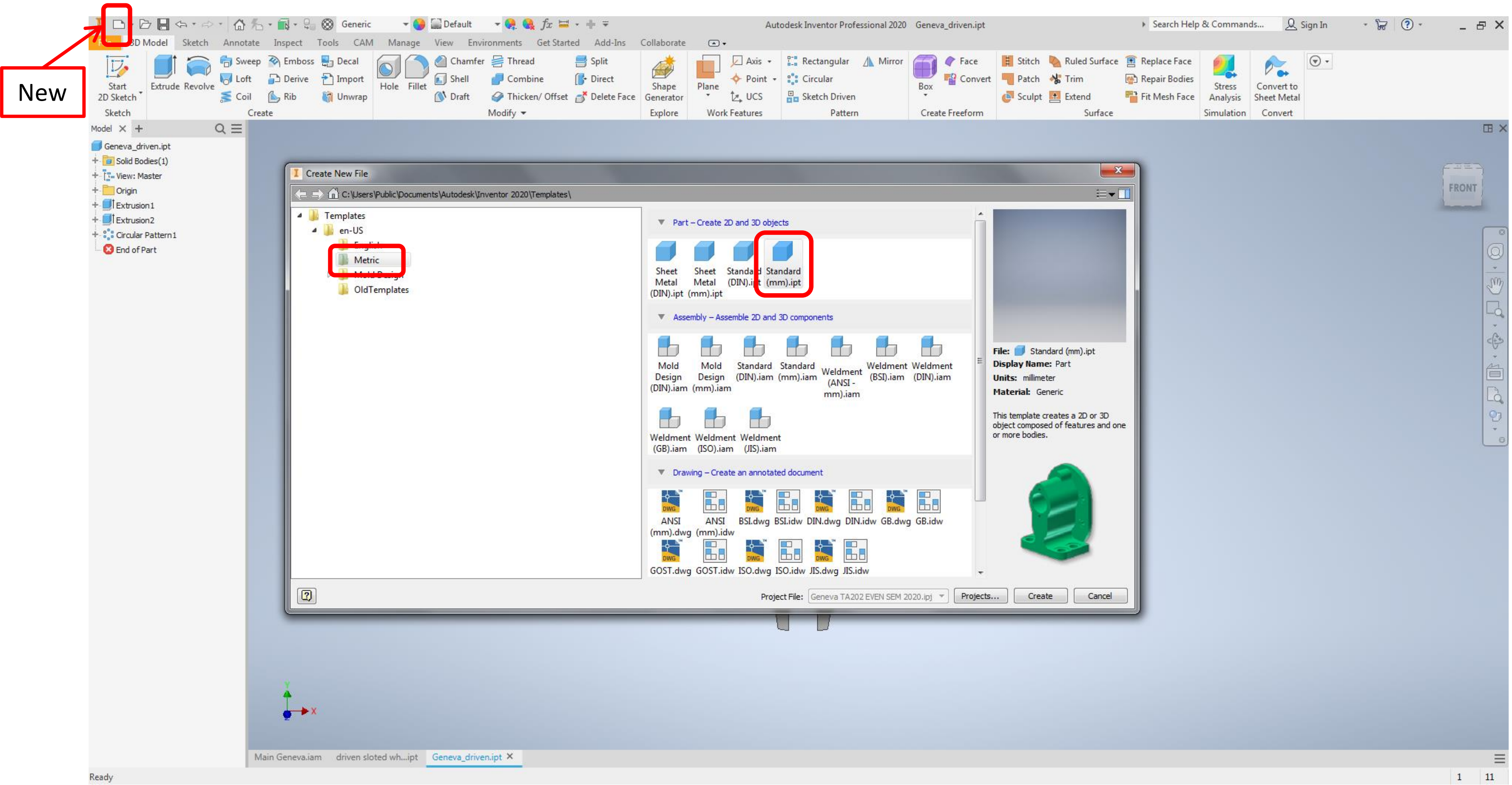
This ends your task 2. Please show your progress to your guide/TA/Tutor. Proceed after.

Task 3
Make a Support Plate

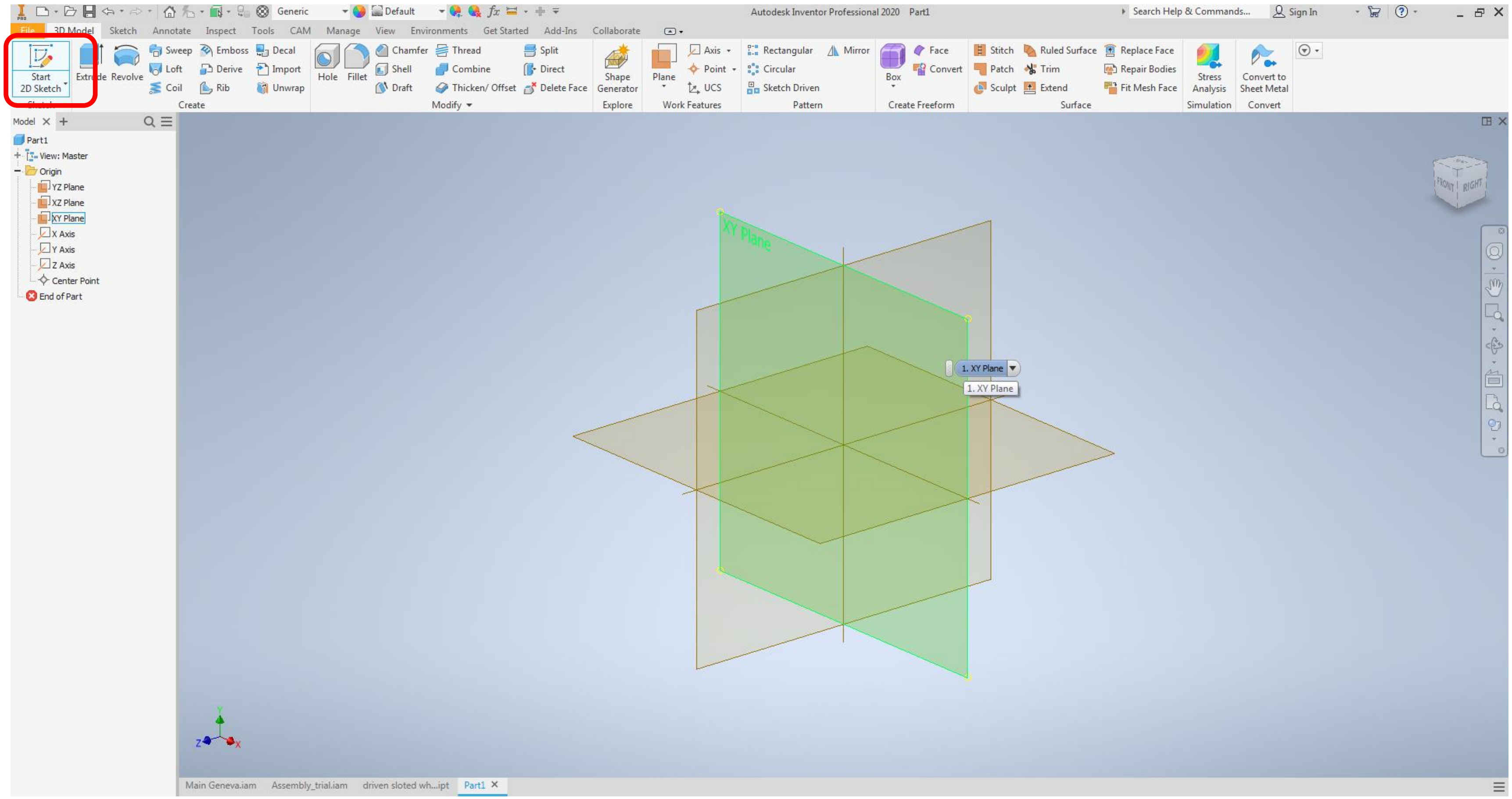




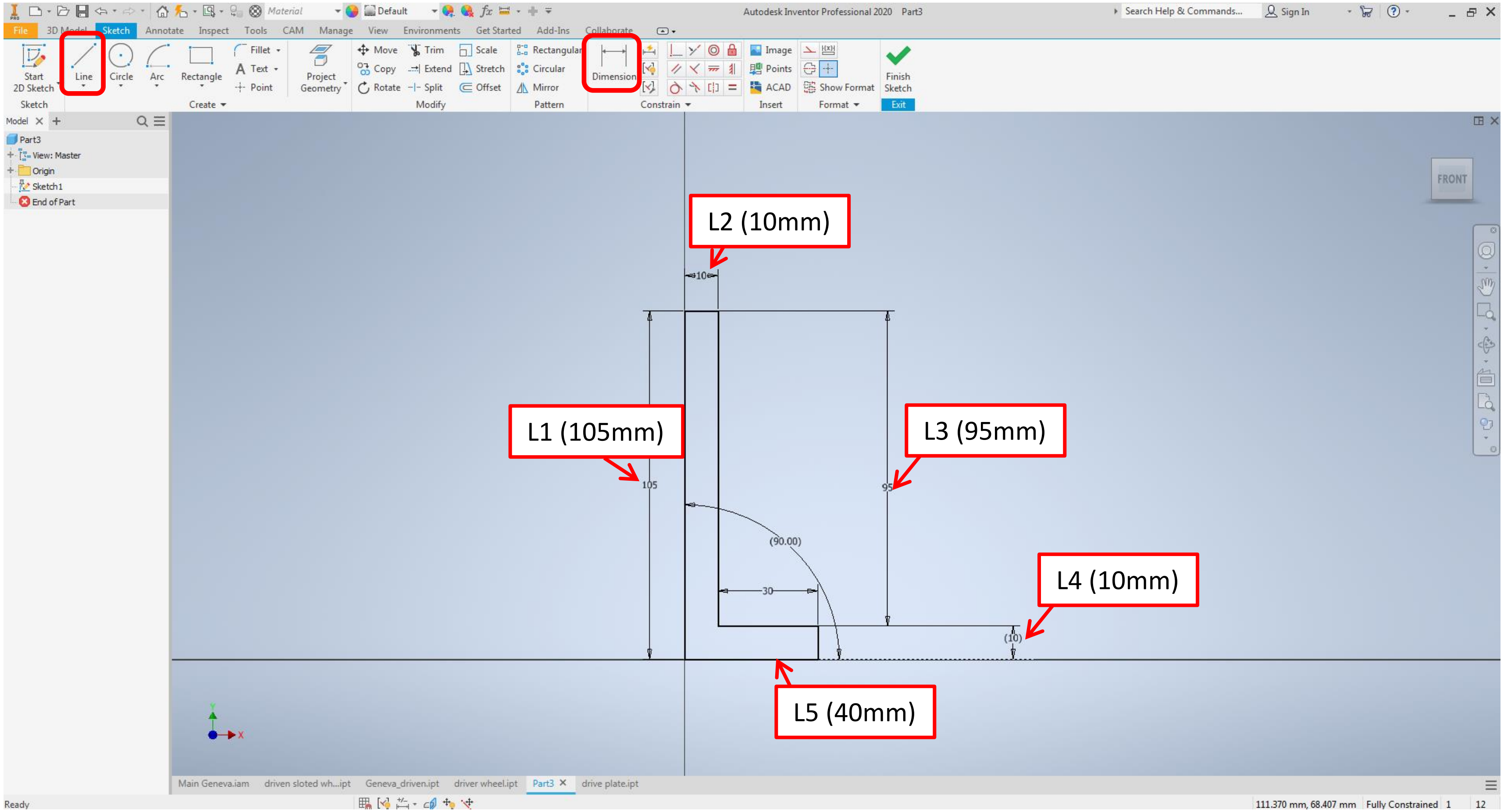
Click on “New” icon, select “Metric” and then select “Standard(mm).ipt”



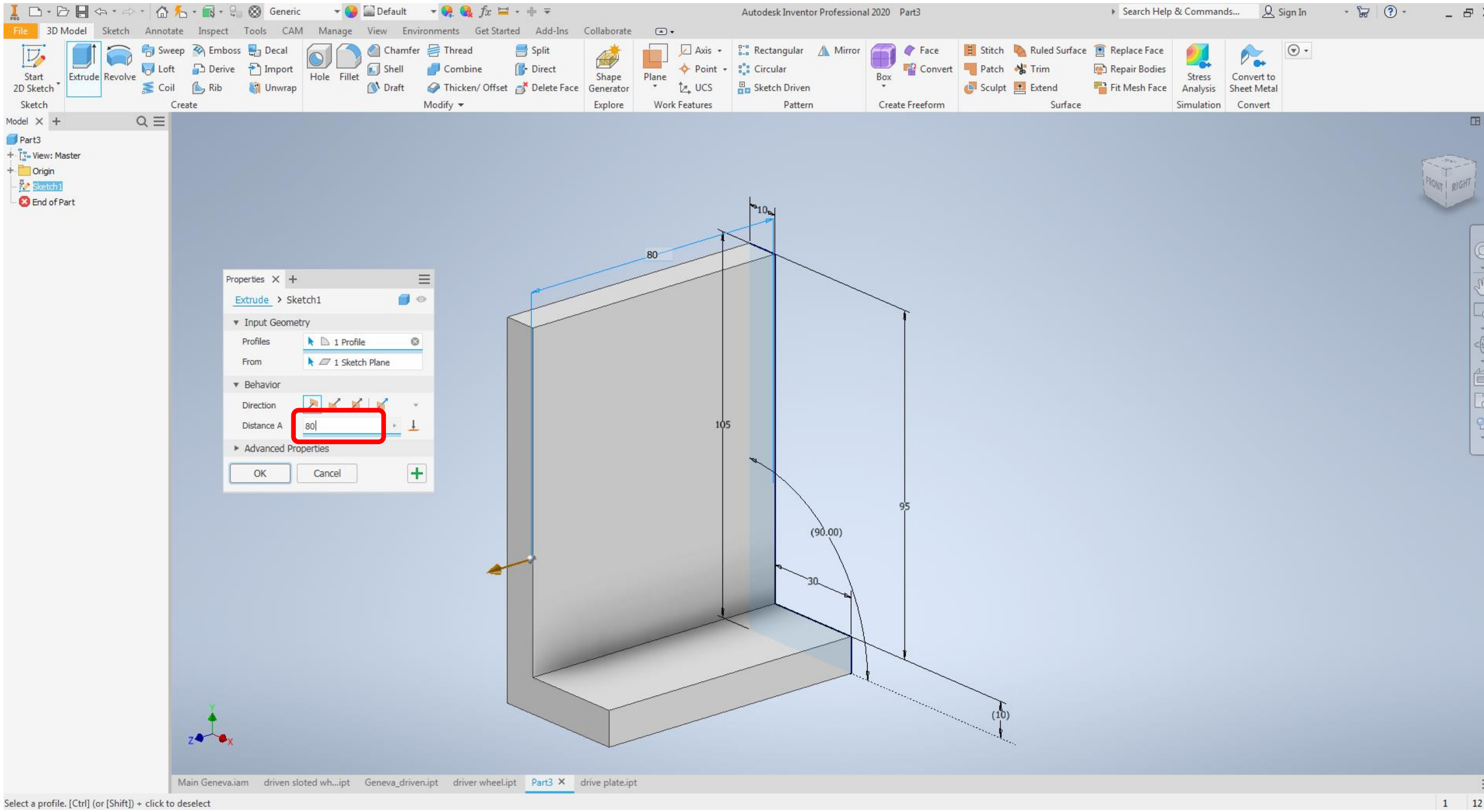
Click on Start 2D sketch and select the XY plane



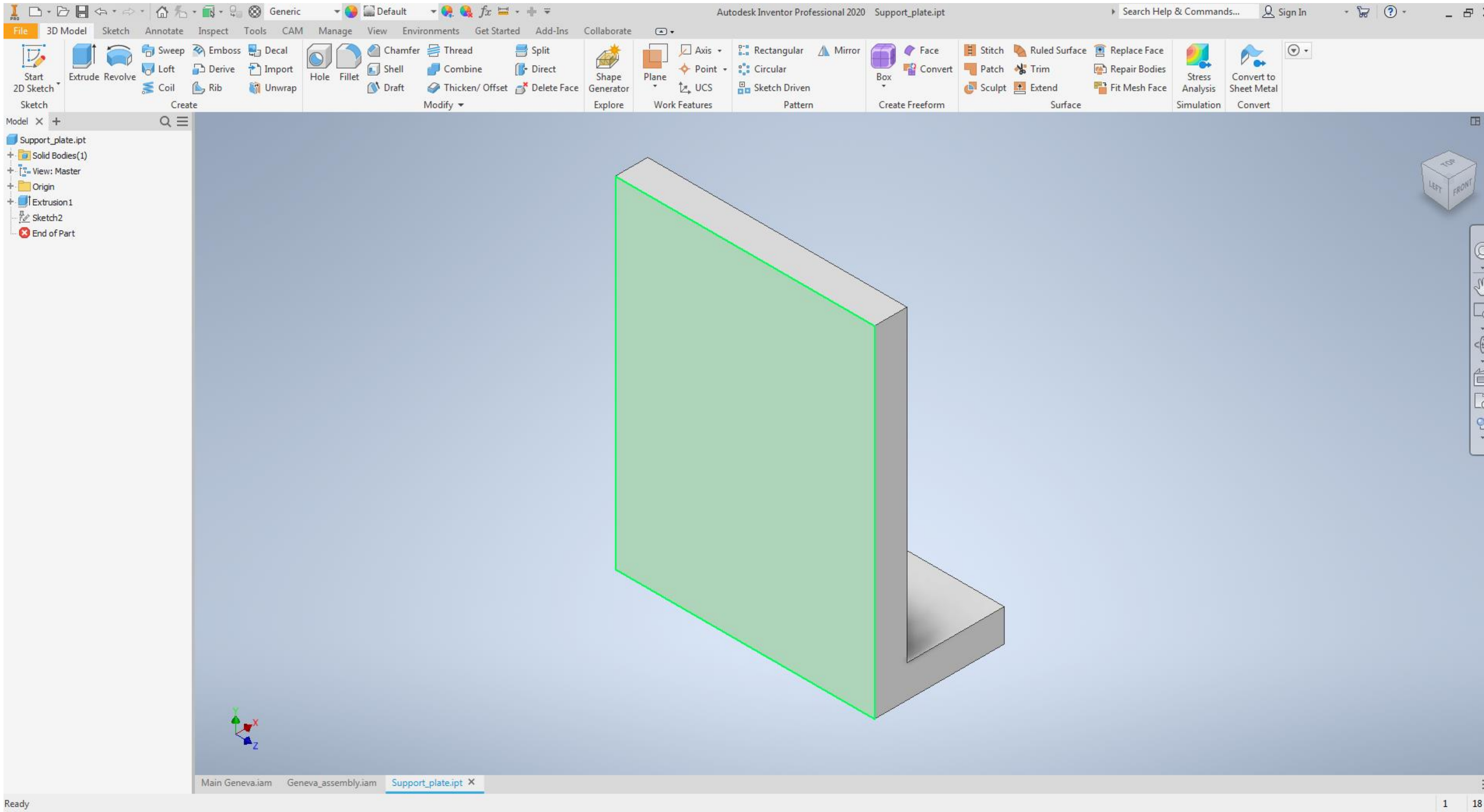
Click on “Line” and draw the sketch as shown below using lines
Click on “Dimensions” and add the dimensions as shown below



Click “ Extrude” and extrude the profile to a distance of 80mm
Click “OK”



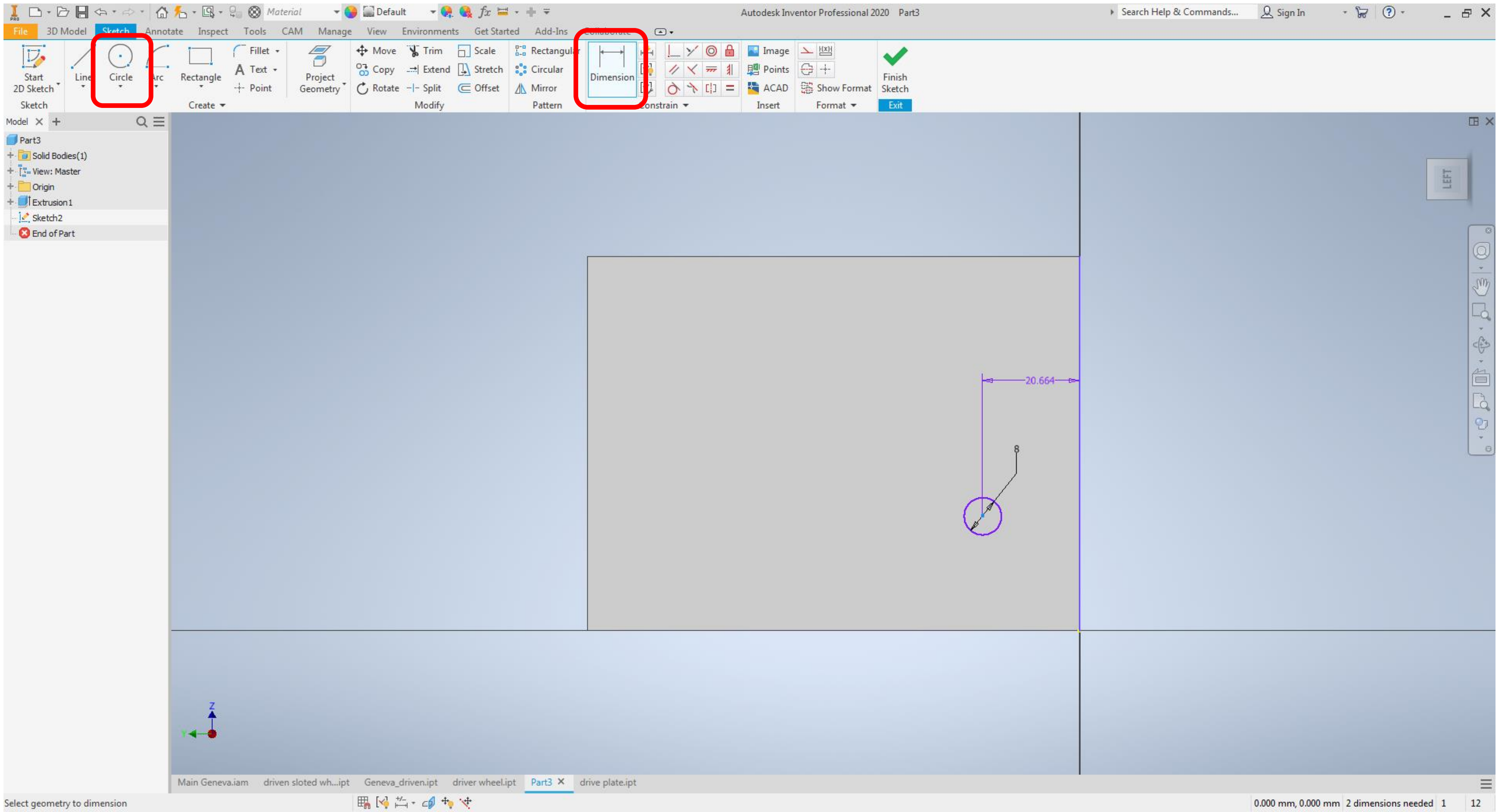
Click on “Start 2D sketch” and select the face(highlighted below)



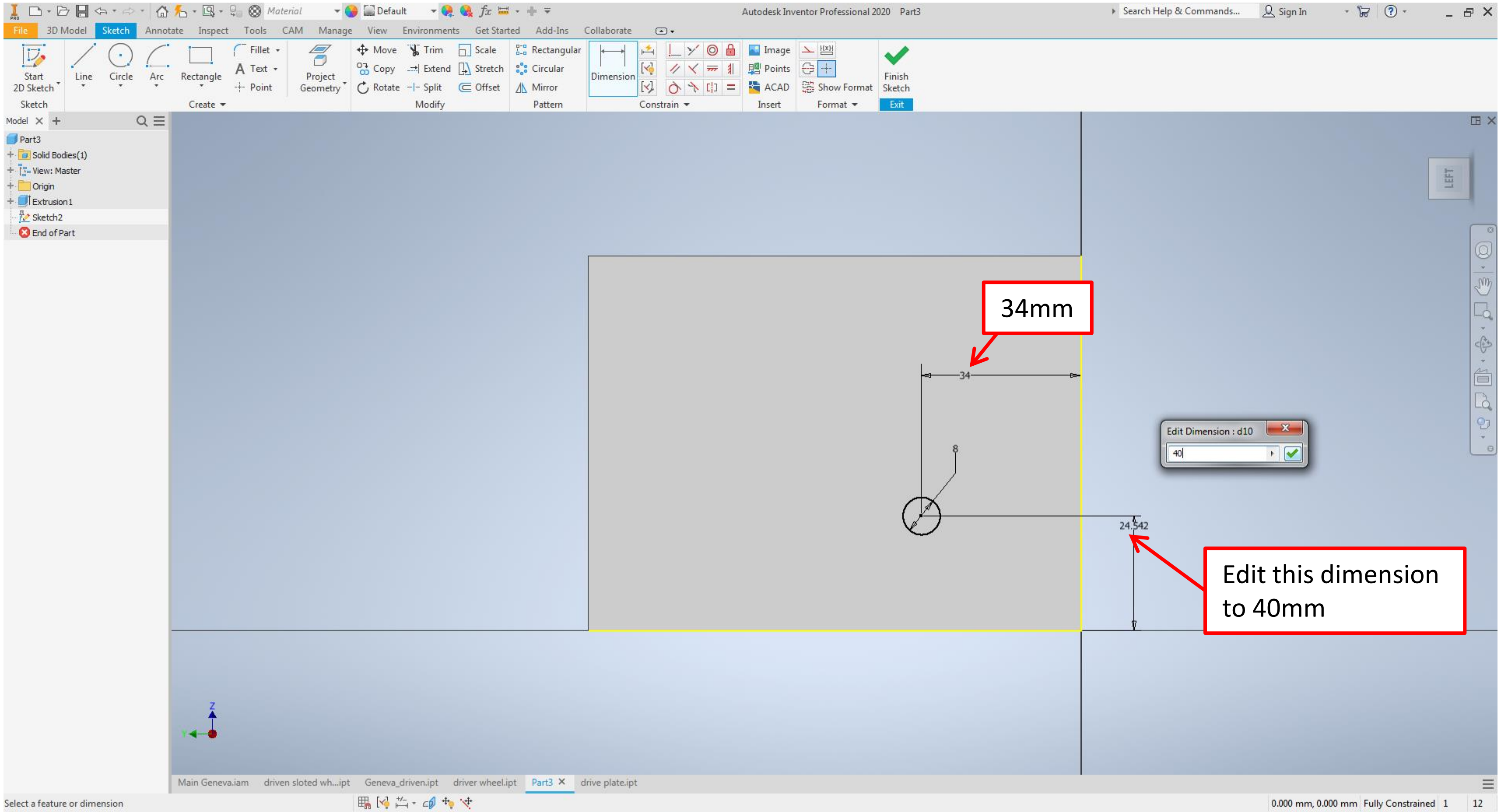
Ensure that the sketch orientation is the same as shown below

Click on “Circle and create a circle of diameter 8mm

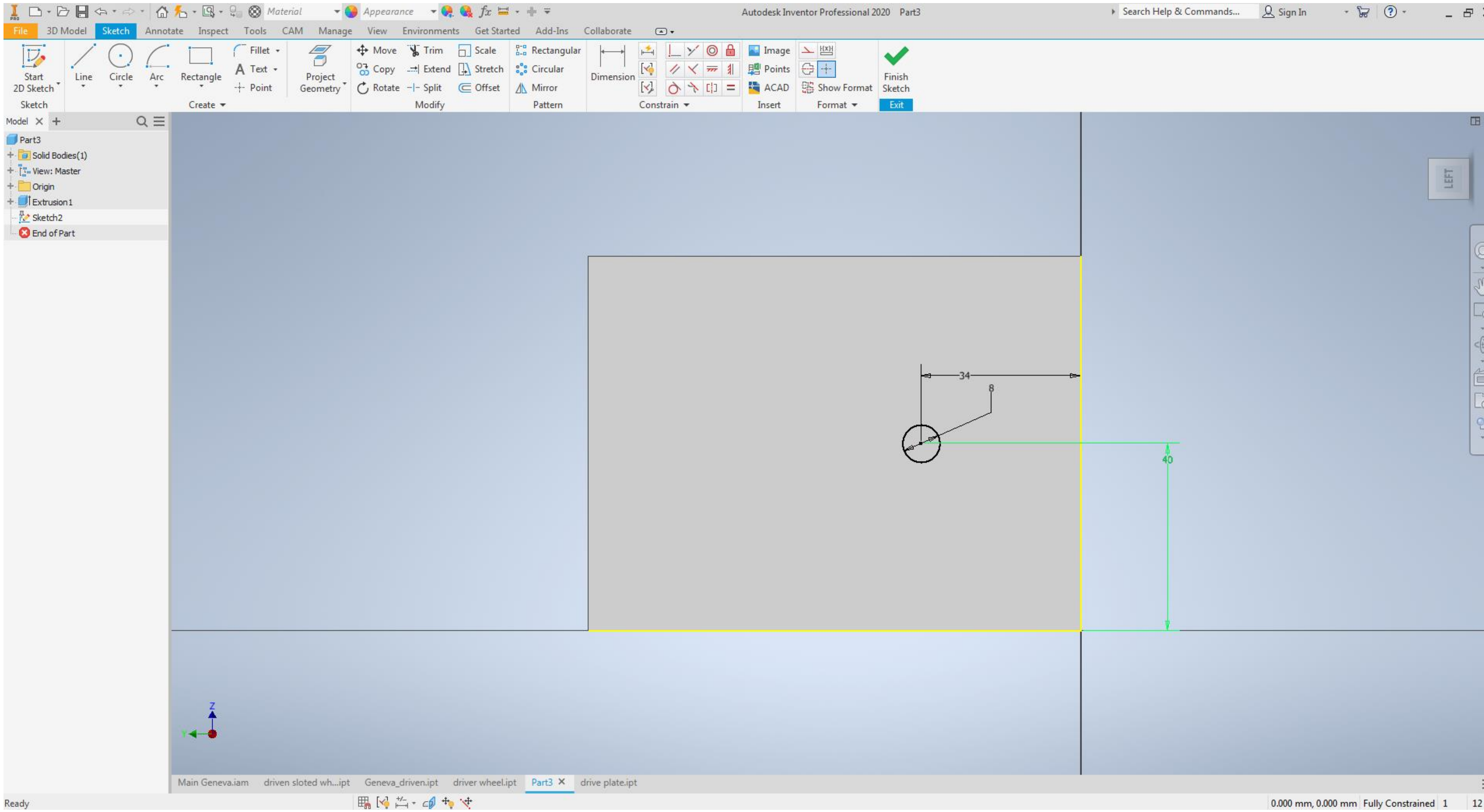
Click on “Dimension” and click on the center of the circle and the vertical line then click on a blank place to create the dimension



Double click the dimension to edit the value and set the value to 34mm
Add the dimension from the center of the circle to the horizontal axis and edit the dimension to 40mm



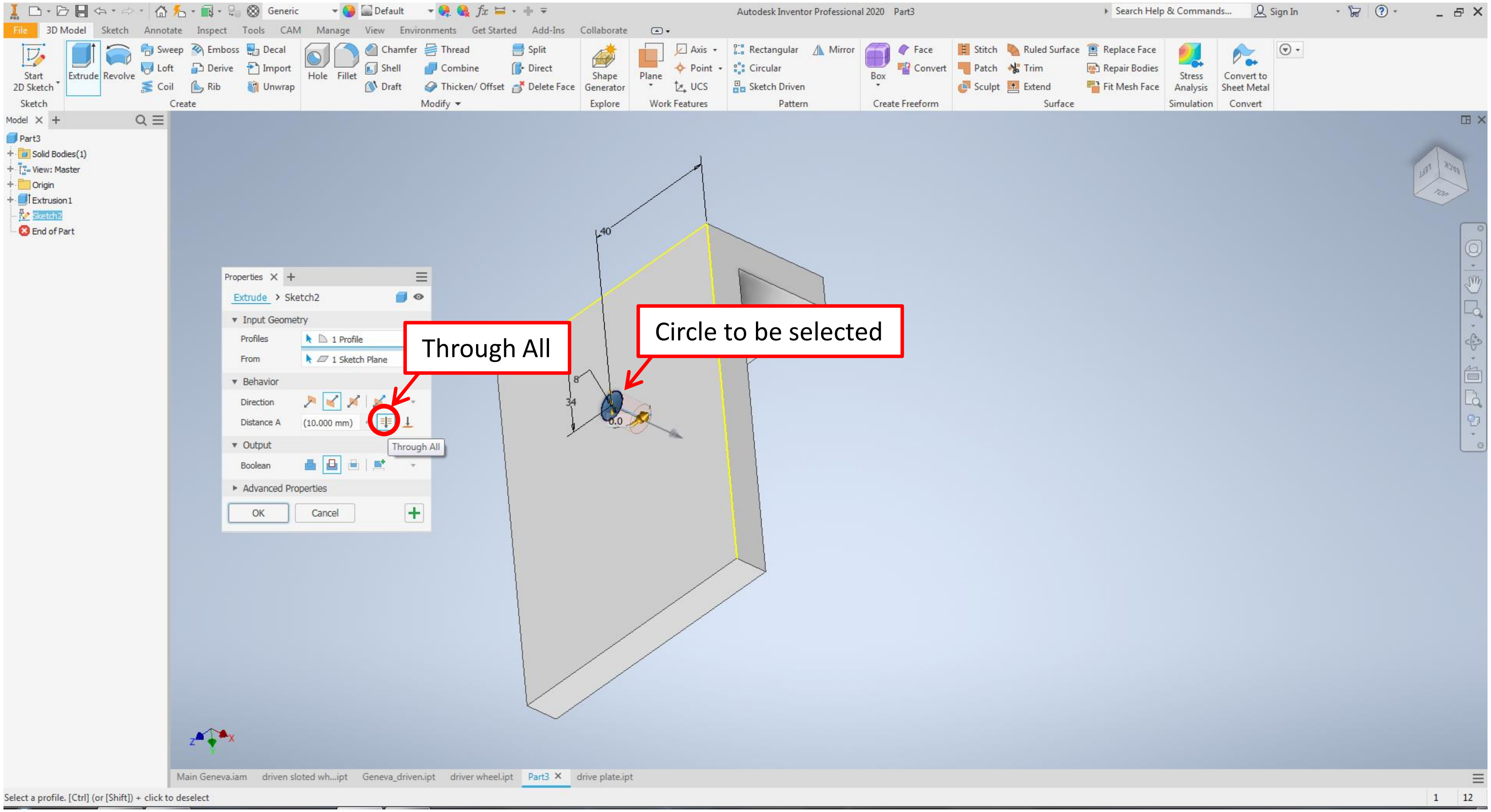
Ensure that the dimensions are as shown below and click on “Finish Sketch”



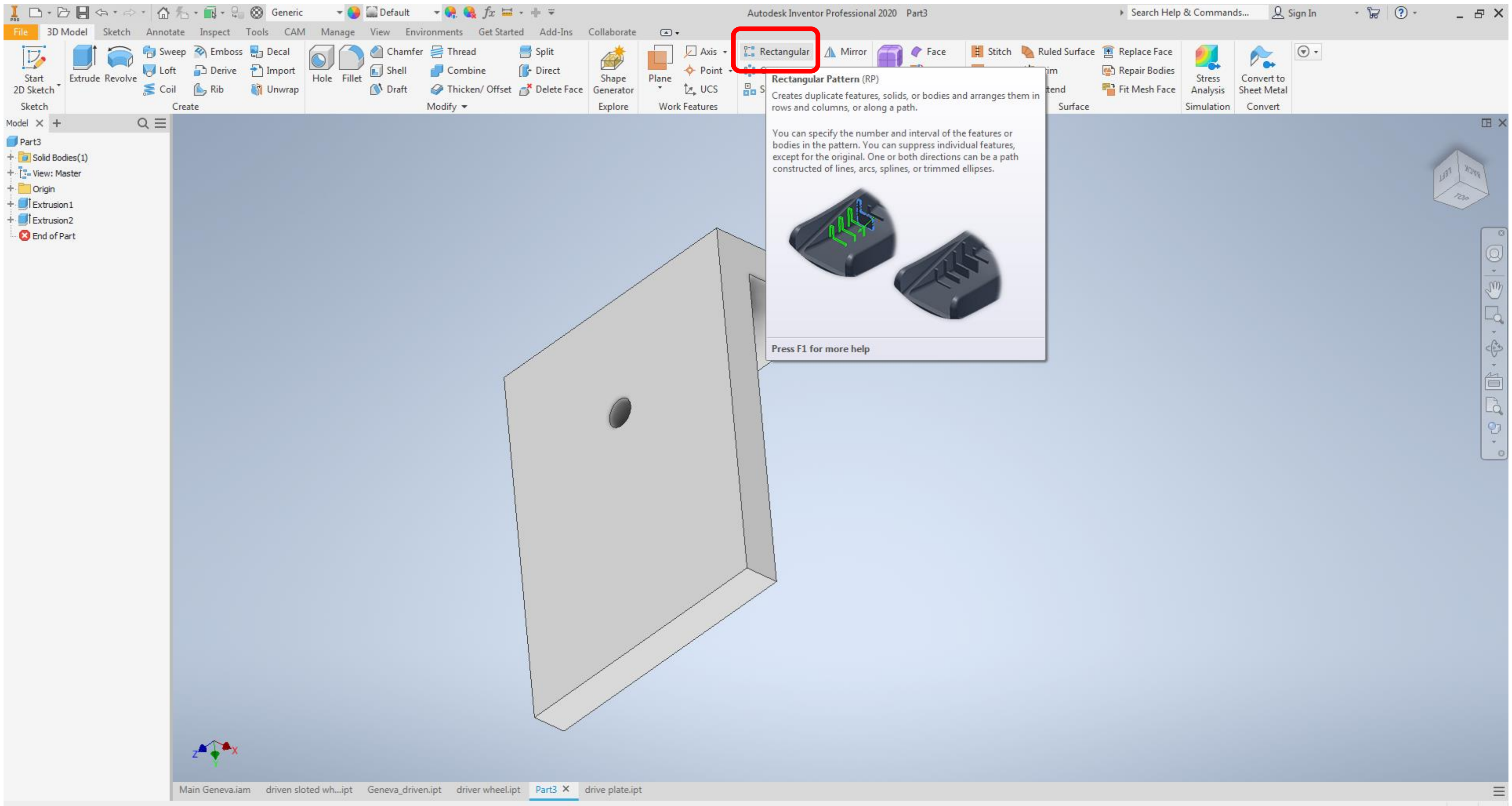
Click "Extrude" and select the circle

Click on "Through All" to create a hole through the entire solid

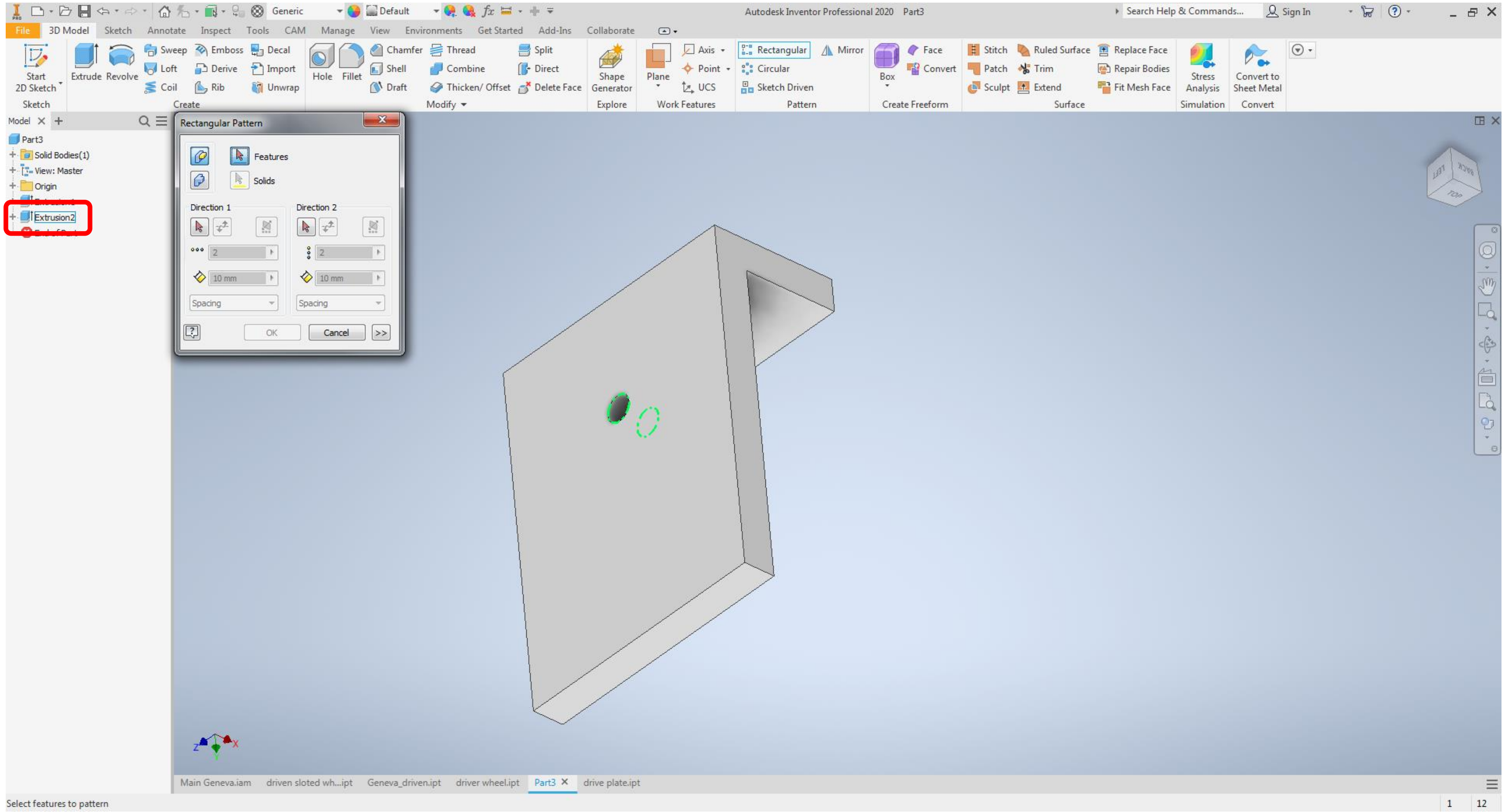
Click "OK"



Click on “Rectangular Pattern”



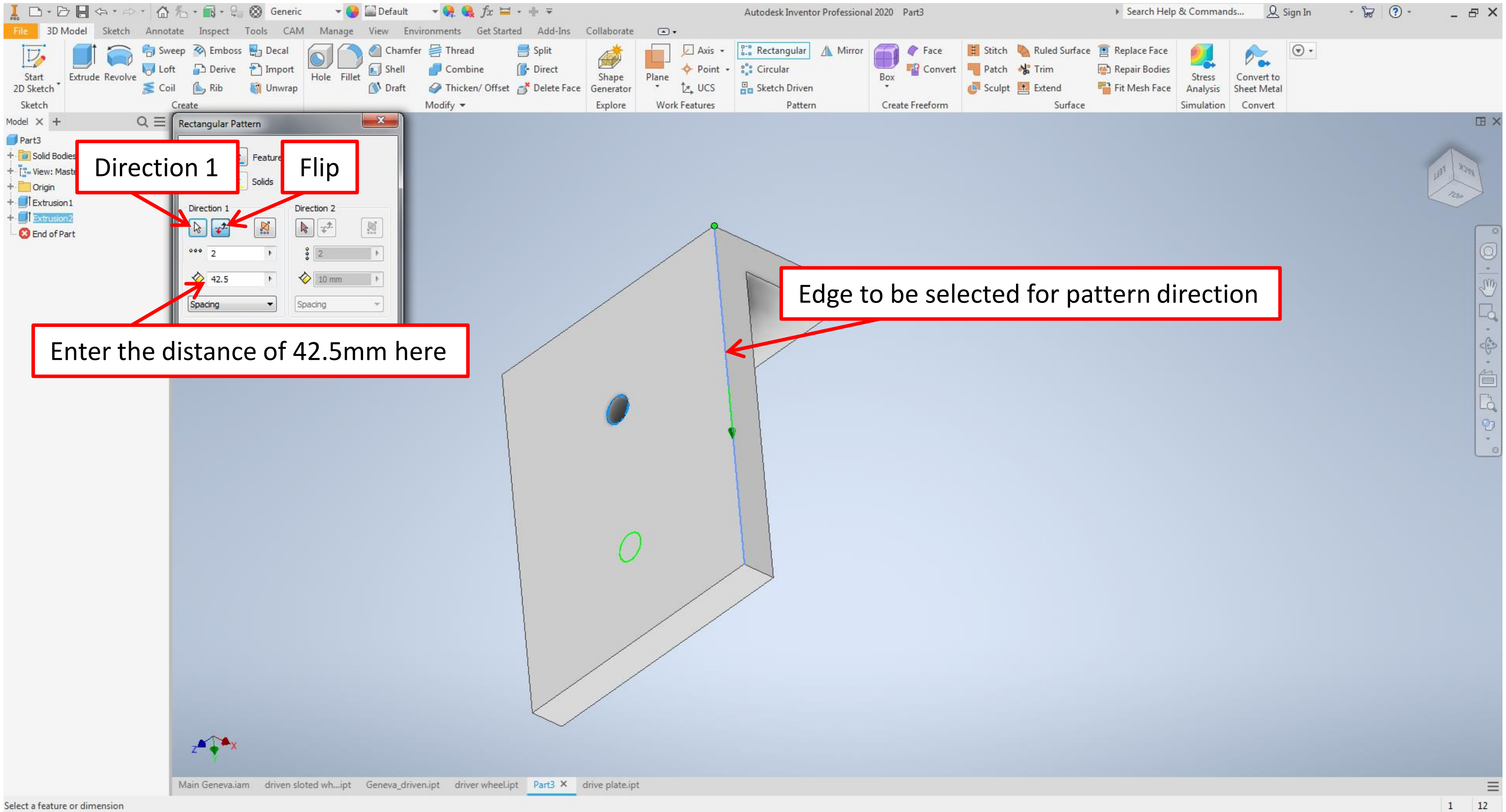
Click on “Extrusion 2” in the model browser



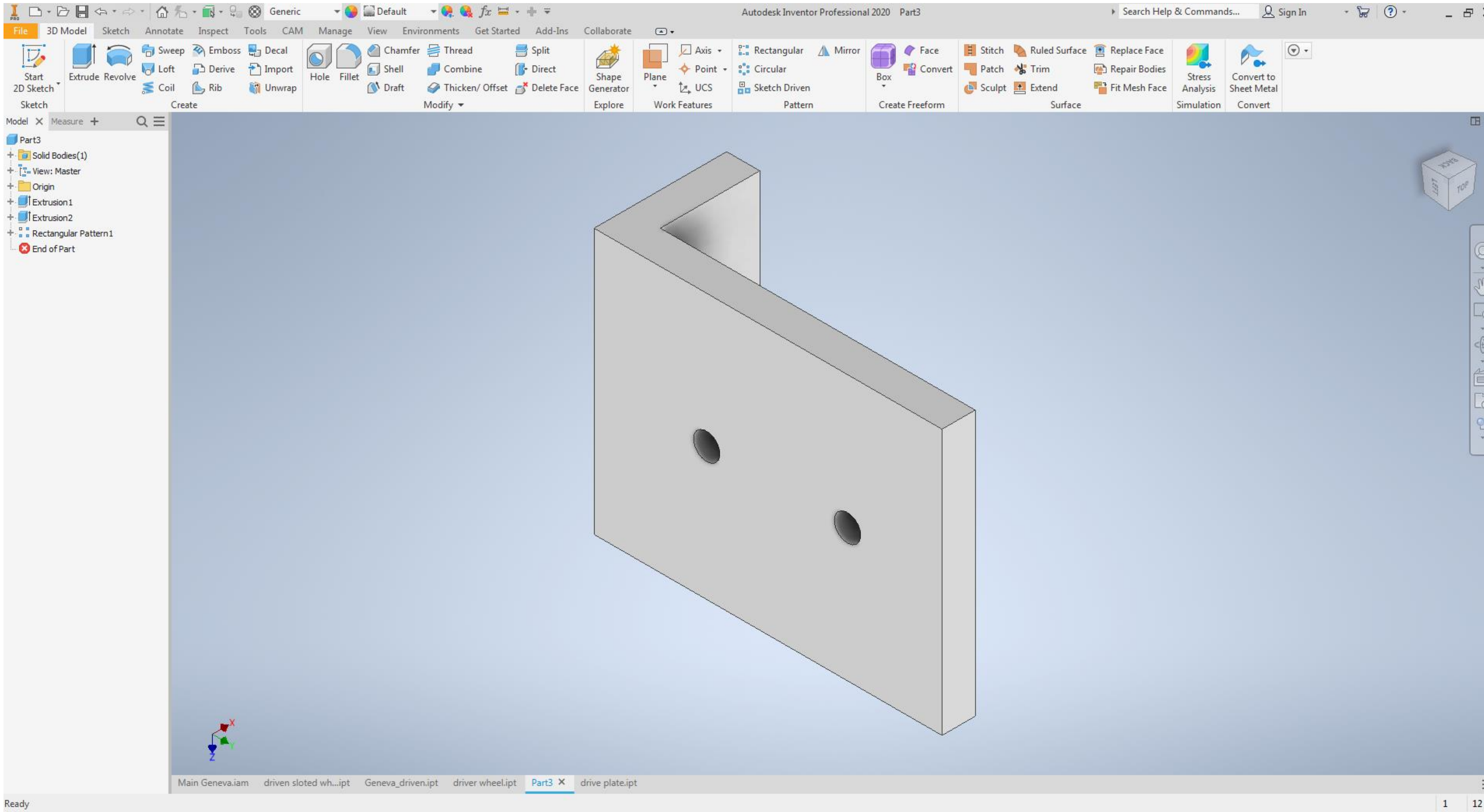
Click on the “Direction 1” and select the edge as shown below to create a pattern along the same direction

Click on “Flip” if the direction of pattern is not as shown

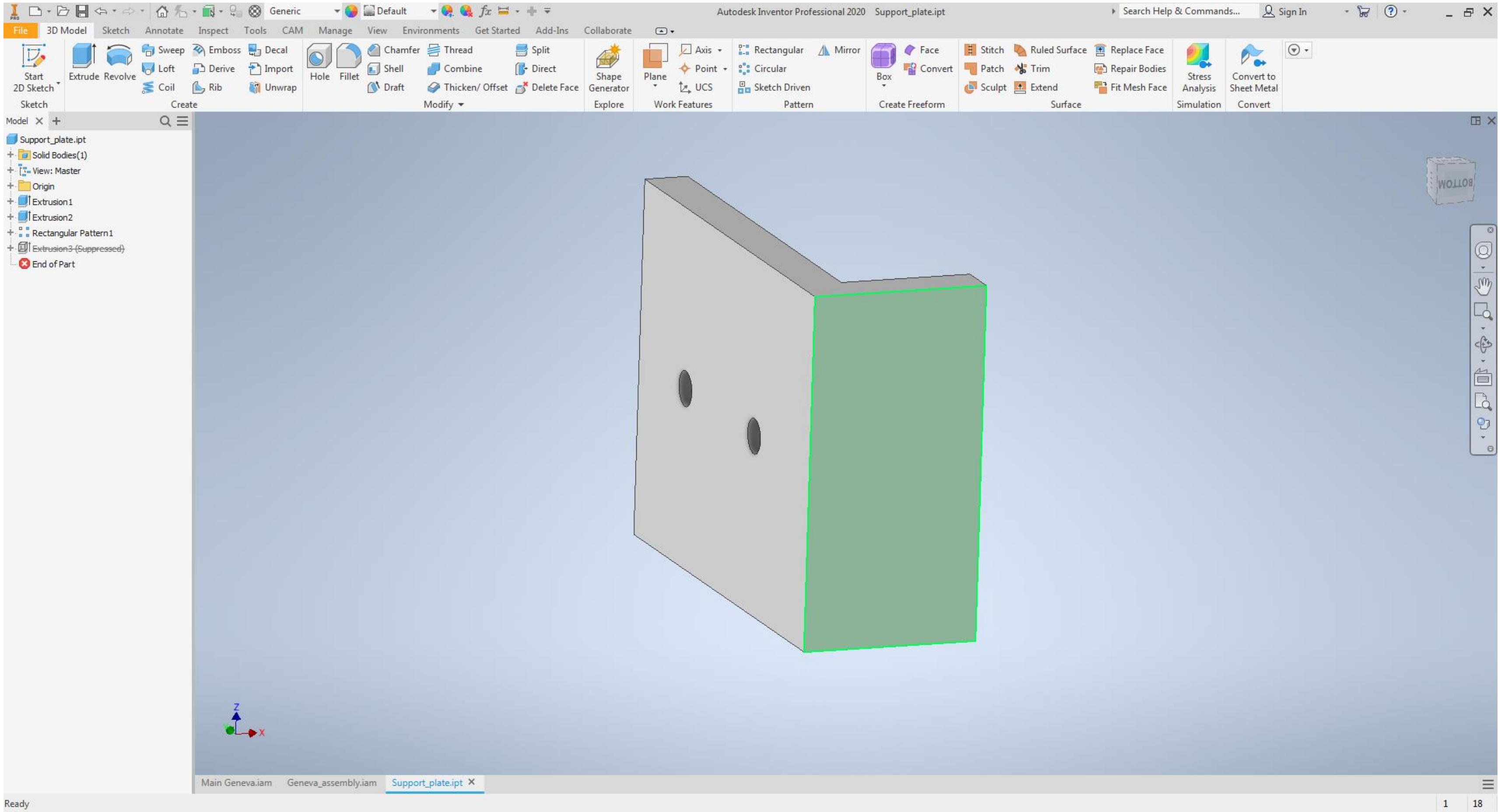
Enter the Distance 42.5mm in the distance box and click “OK”



This is how the part should look



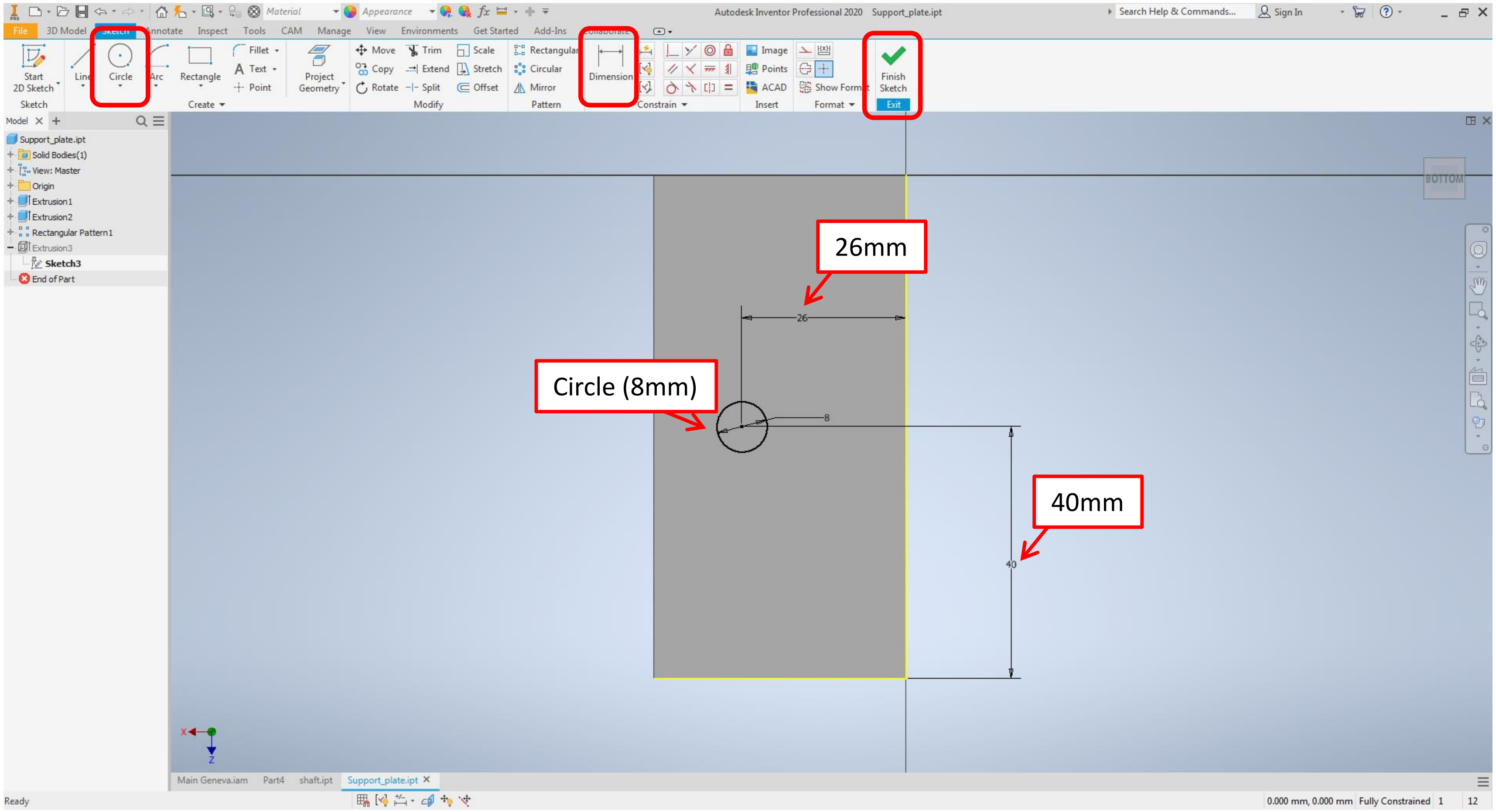
Click on “Start 2D sketch” and select the plane shown below



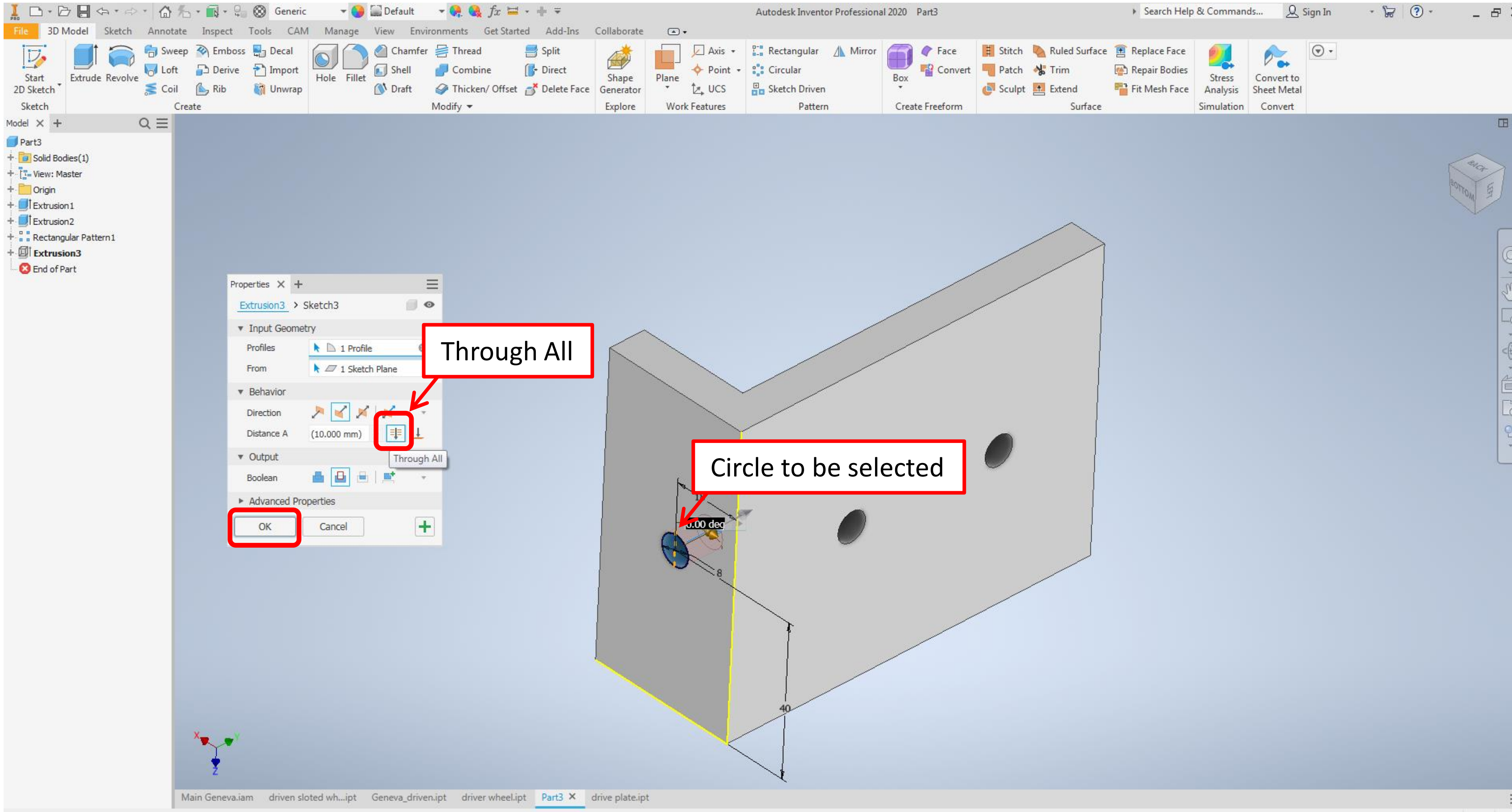
Click on "Circle" and draw a circle of 8 mm diameter

Click on dimension and add the dimensions as shown below

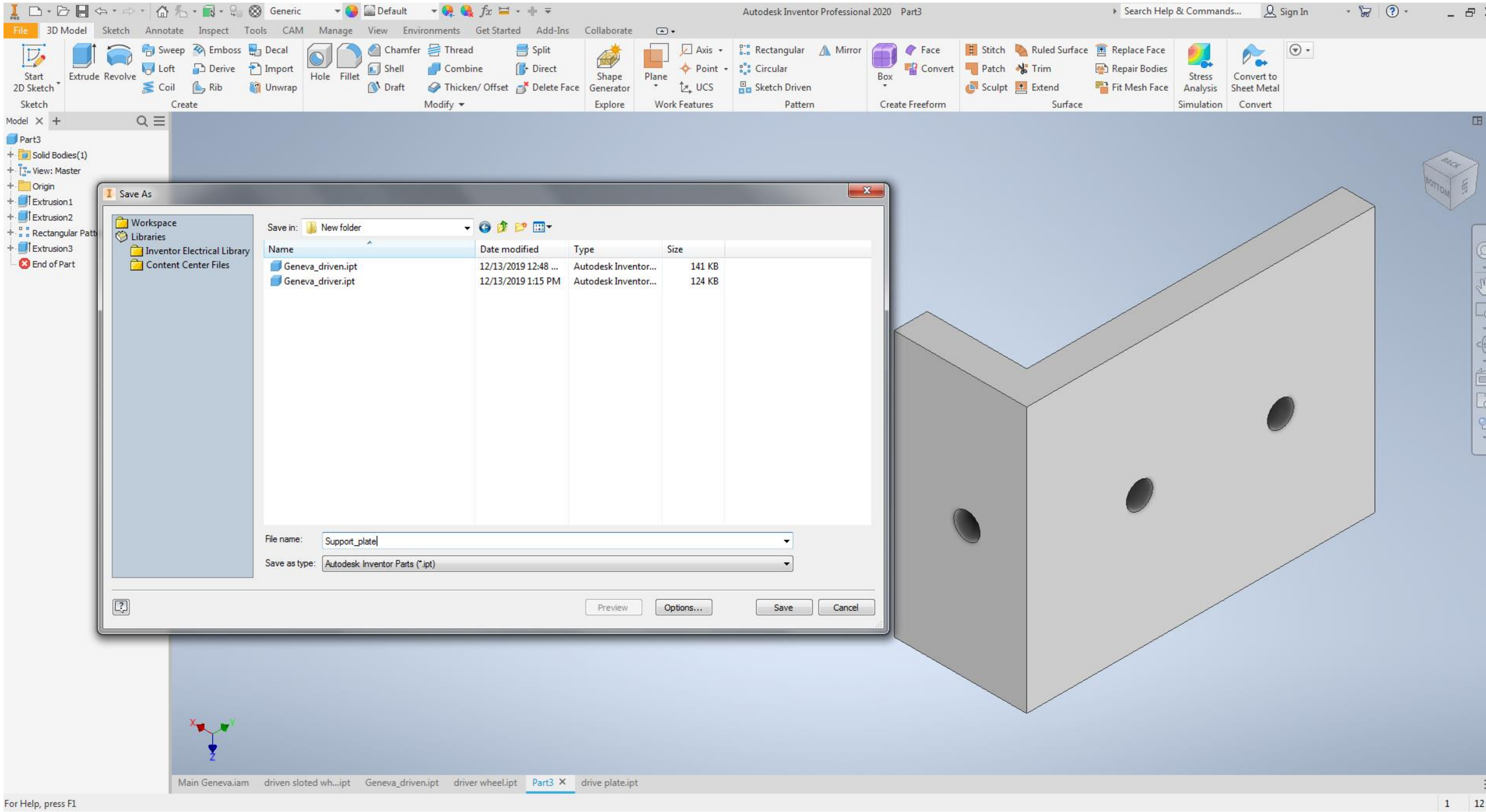
Click on "Finish Sketch"



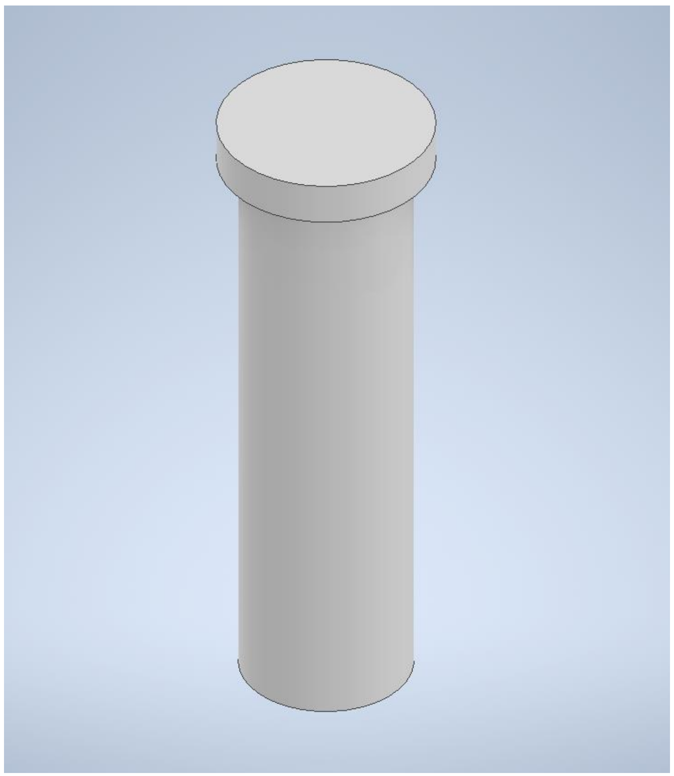
Click “Extrude” and select the circle to create a hole
Click on “Through all” and click “OK”

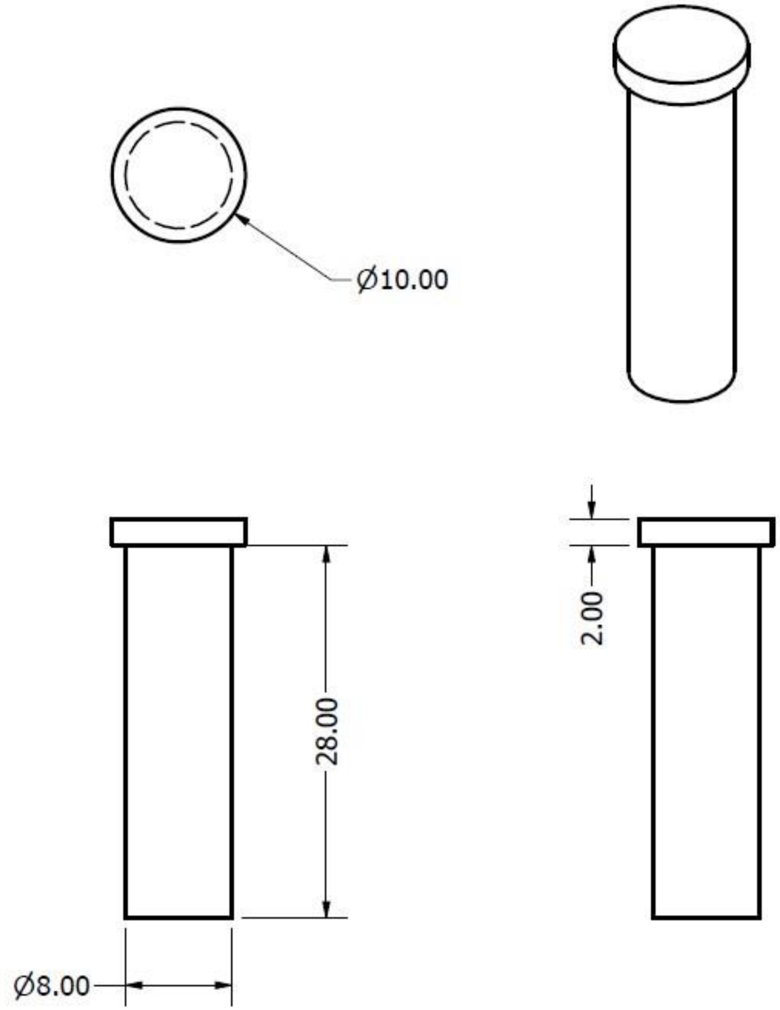


Click on “File”, select “Save” and save the part

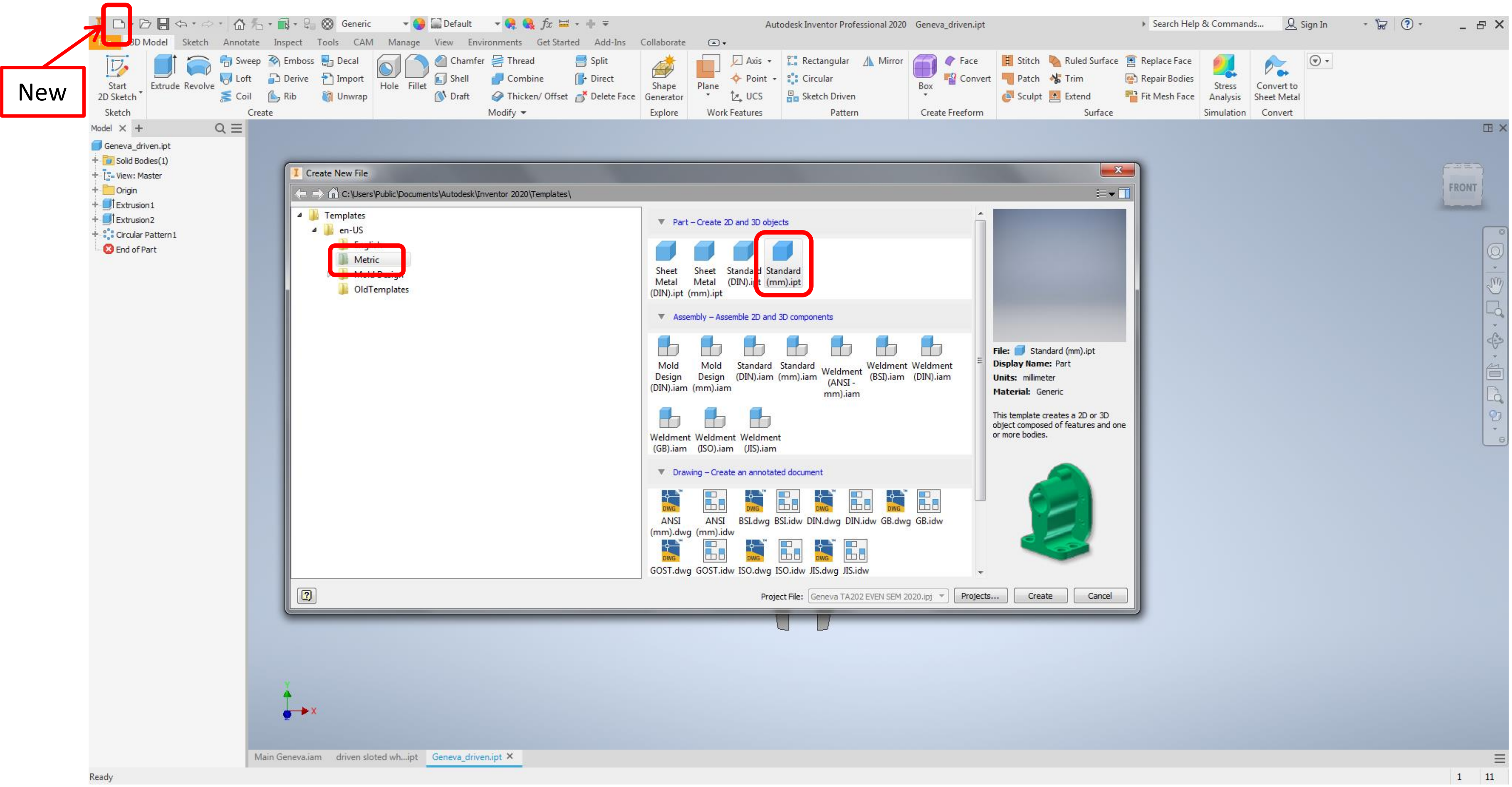


Task 4
Make a Shaft

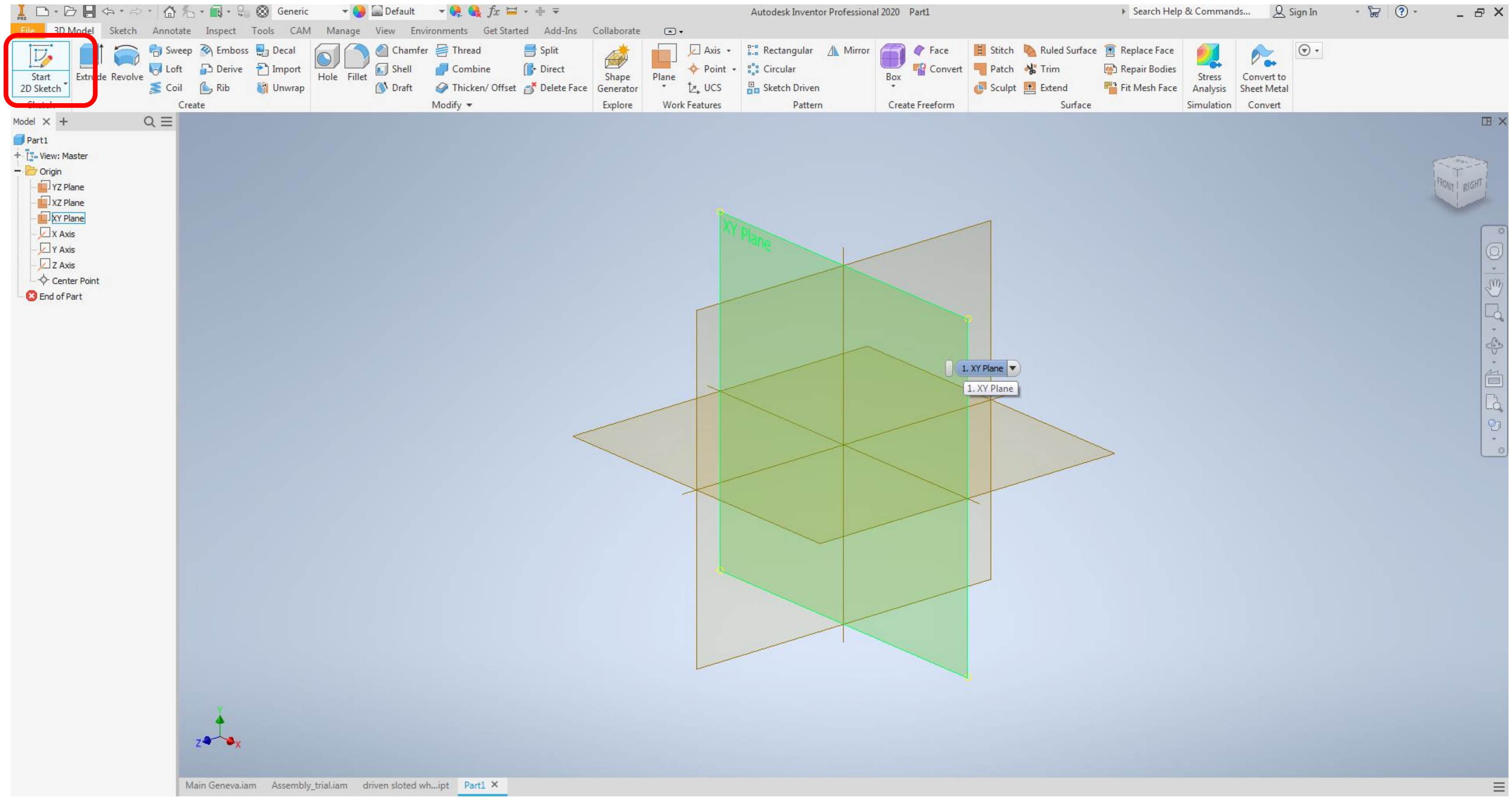




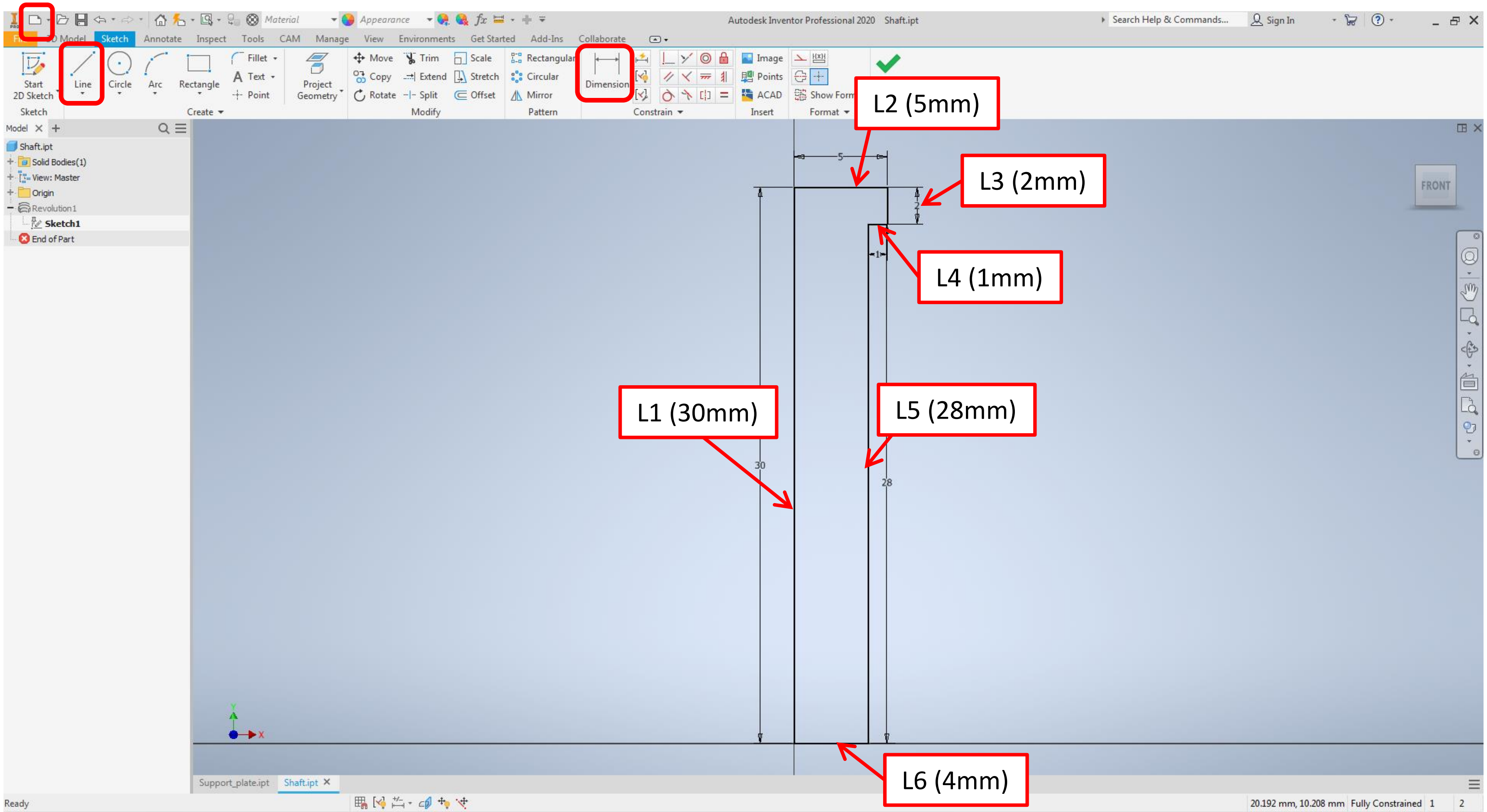
Click on “New” icon, select “Metric” and then select “Standard(mm).ipt”



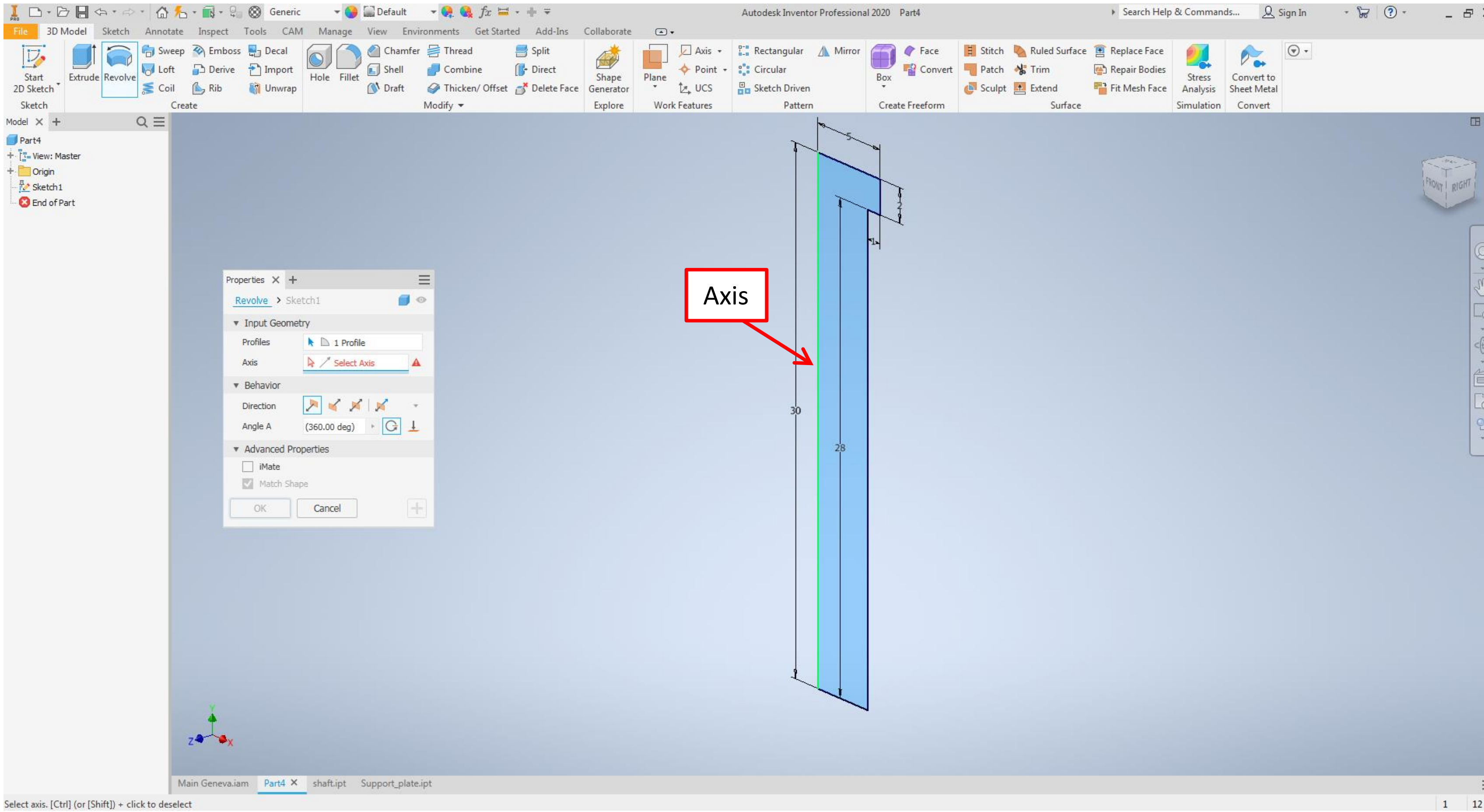
Click on Start 2D sketch and select the XY plane



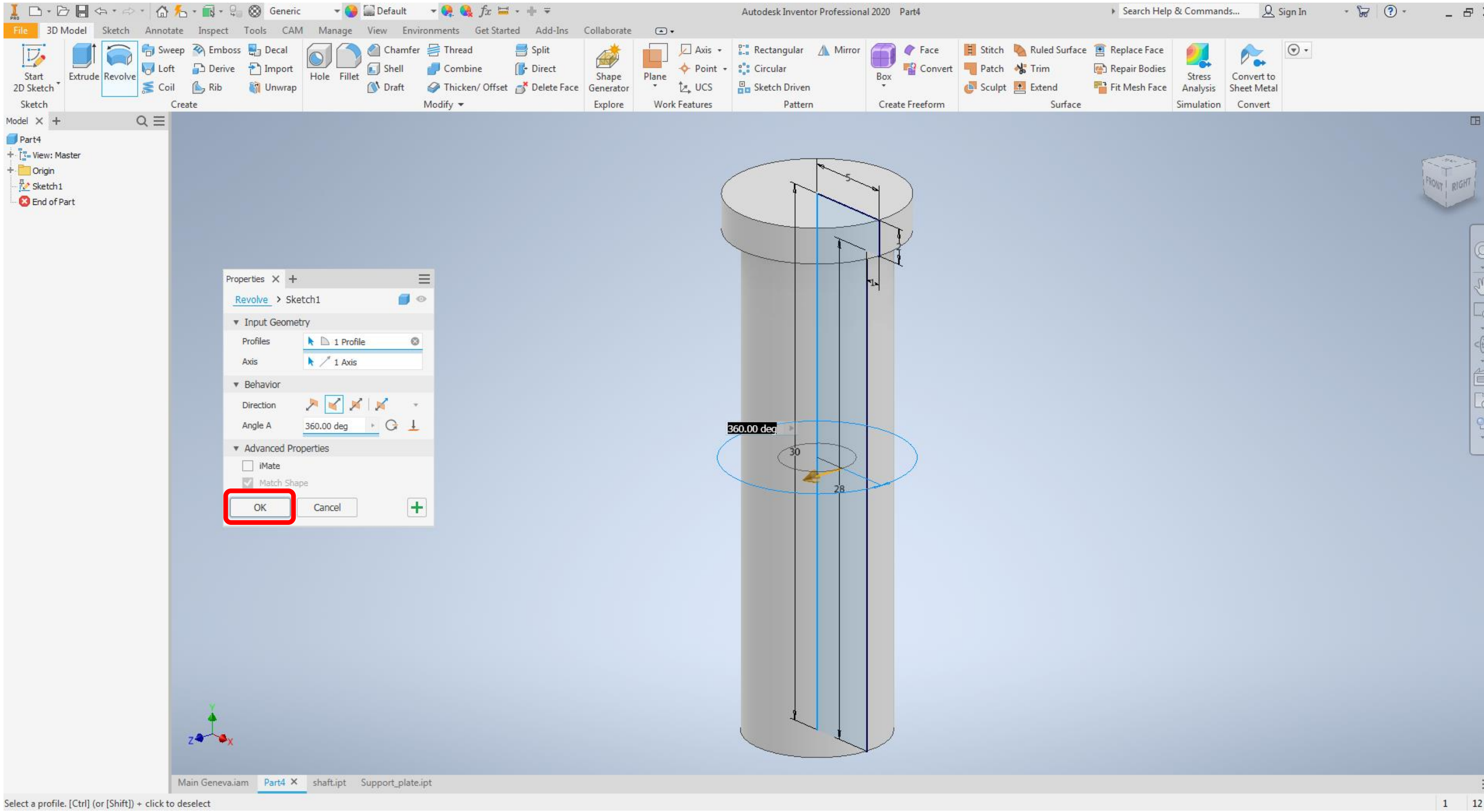
Click on “Line” and using lines, draw the sketch as shown below with dimensions
* Make sure to create a closed sketch by drawing all six lines (L1 to L6) as shown



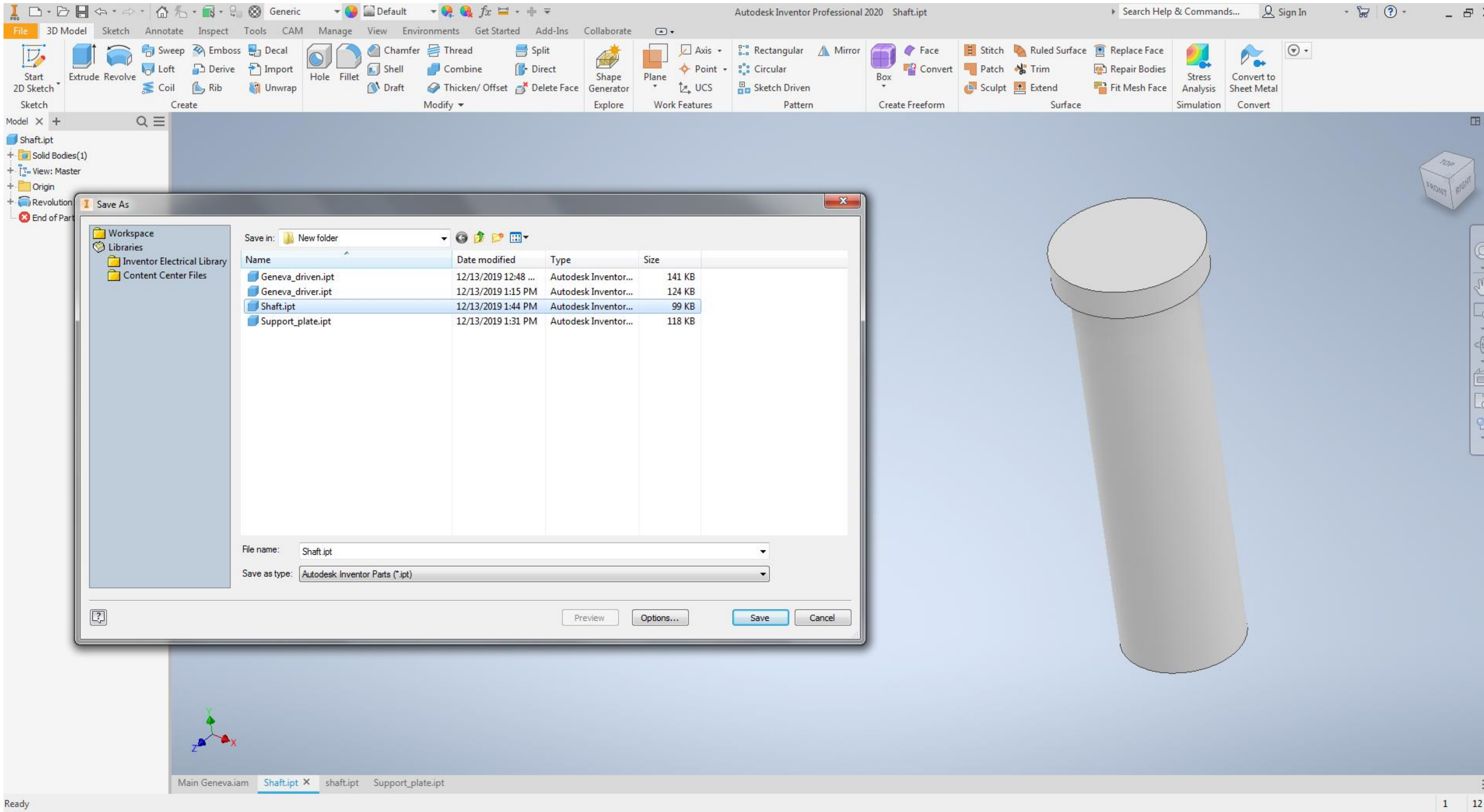
Click on “Revolve” and select the axis about which the profile is to be revolved



Click on “OK”

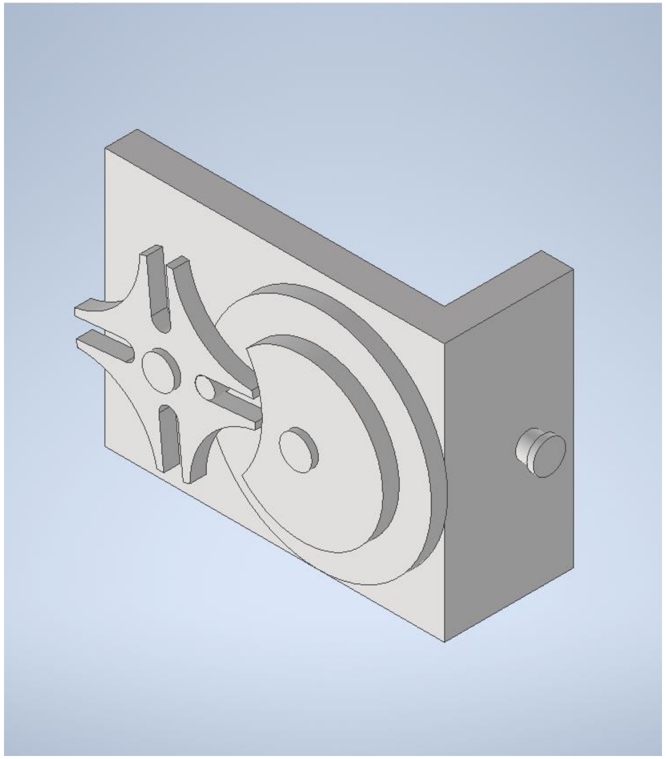


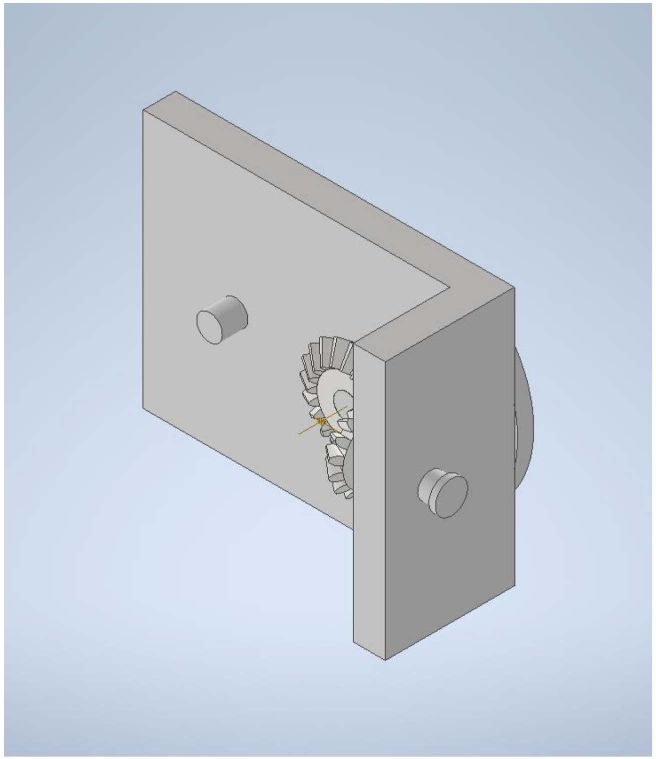
Click on “File”, select “Save” and save the part



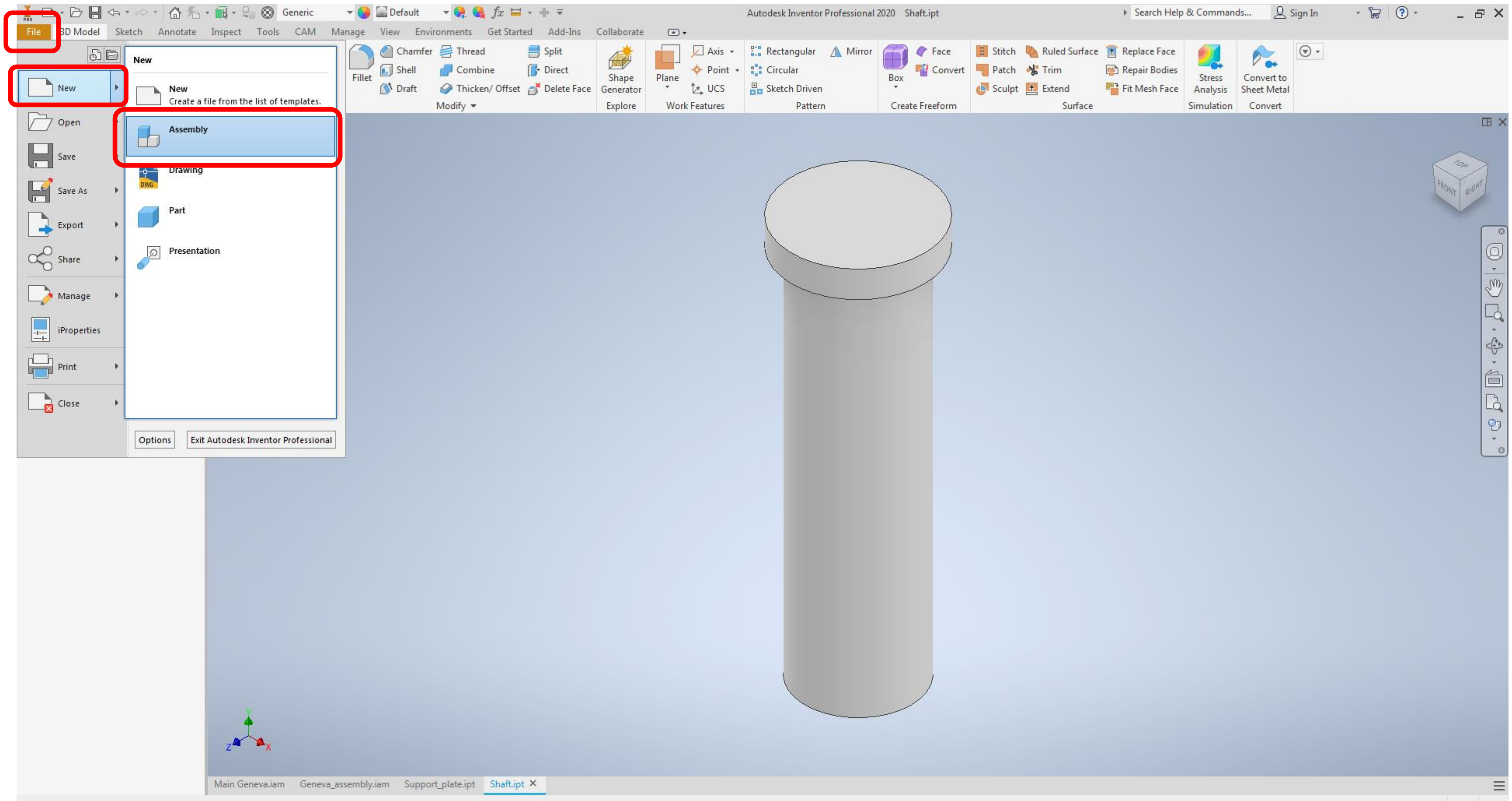
This ends your task 3 and 4 too. Please show your progress to your guide/TA/Tutor. Proceed after.

Task 5
Assemble all the parts

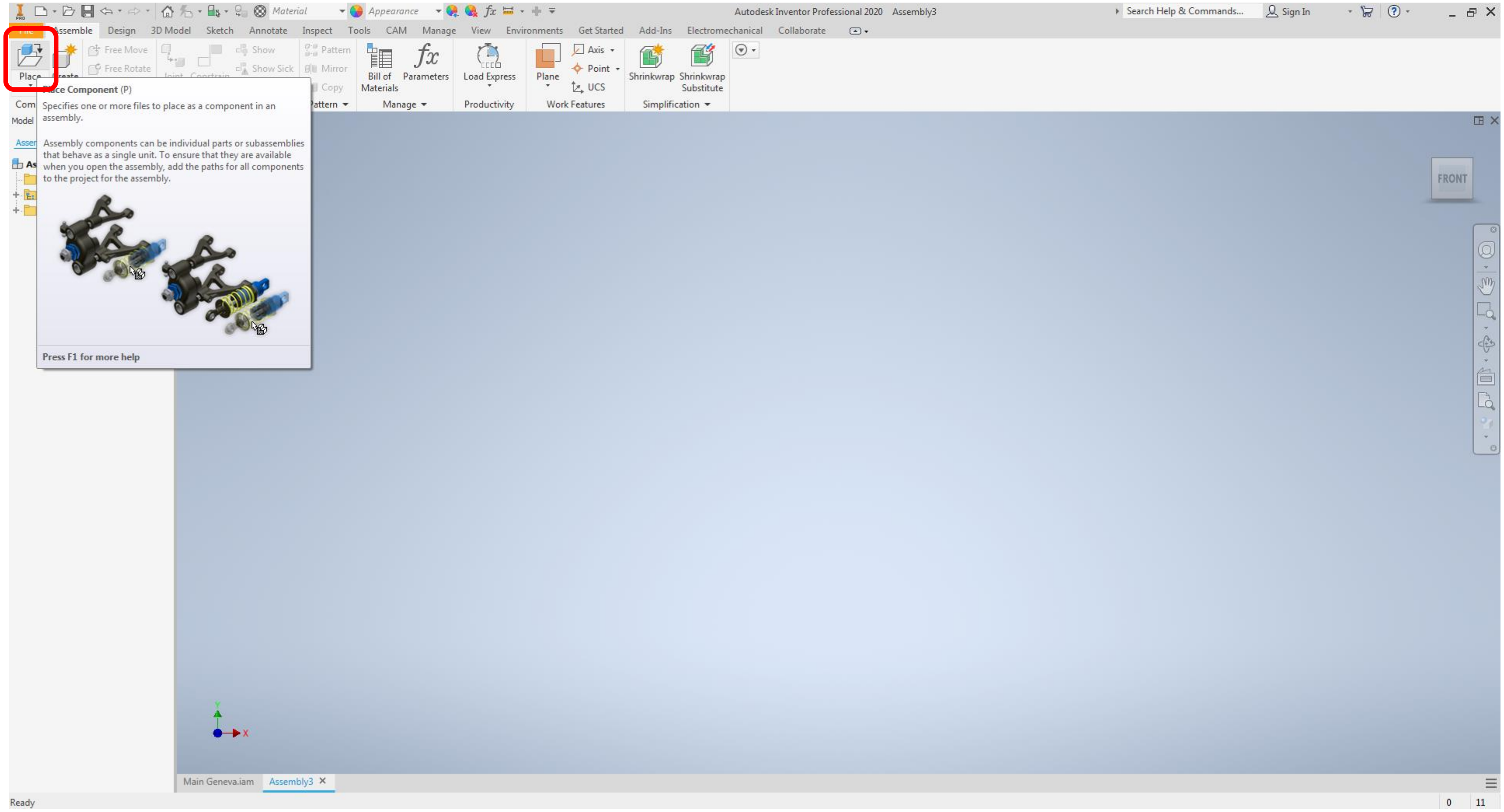




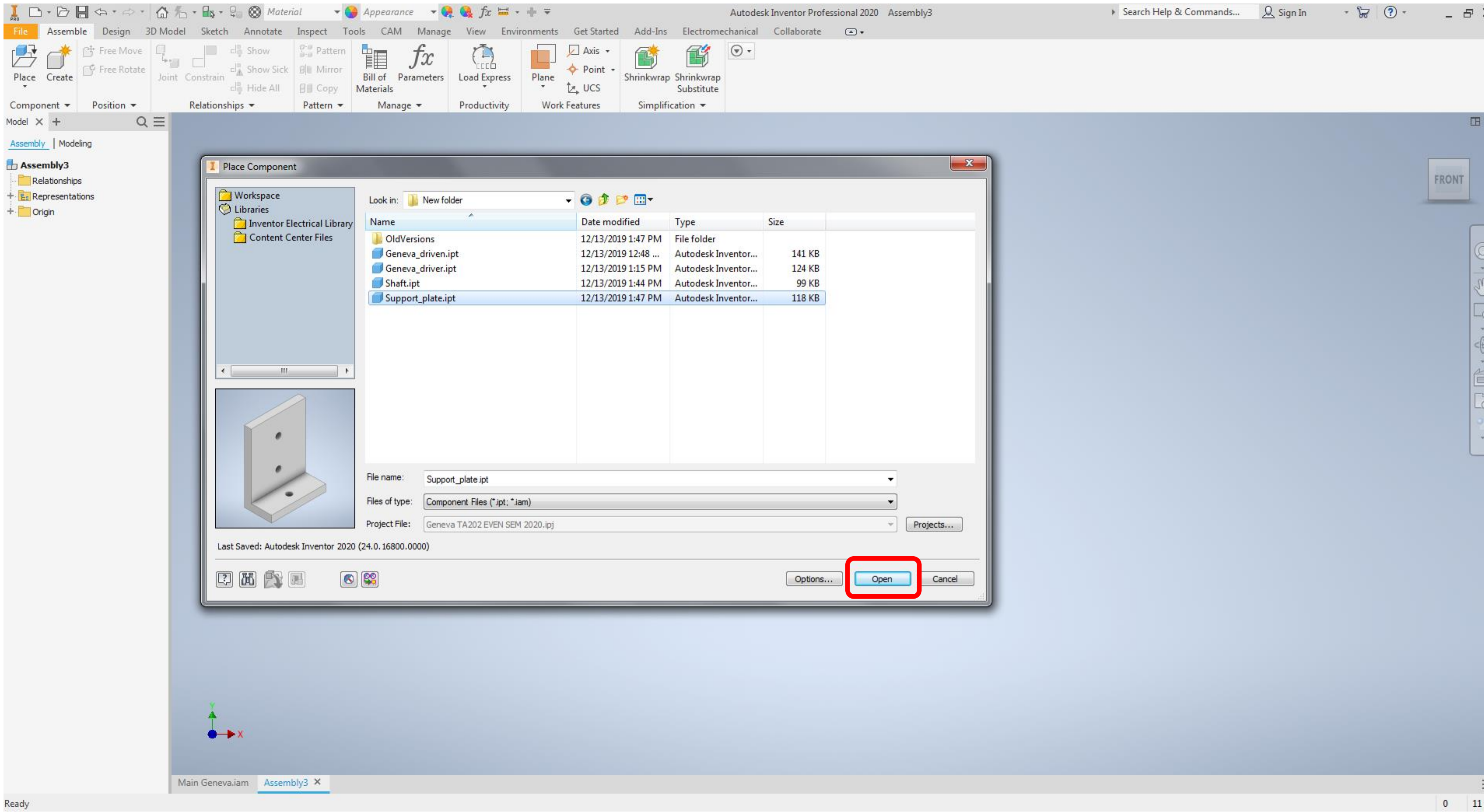
Click on “File”, click on “New” drop down and select “Assembly”



Click on “Place” to place the parts in the Assembly

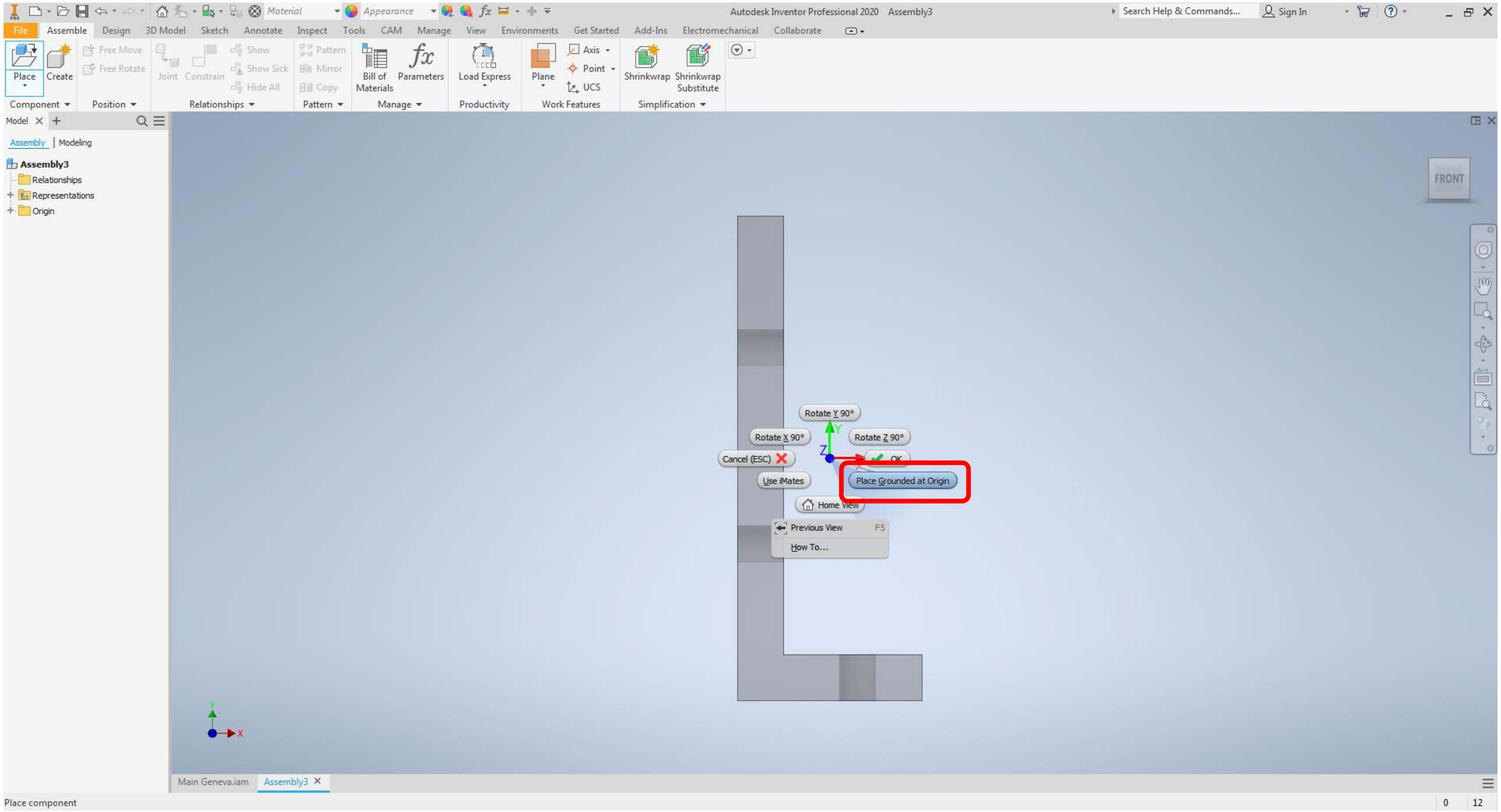


Select the “Support plate” and click on “Open”

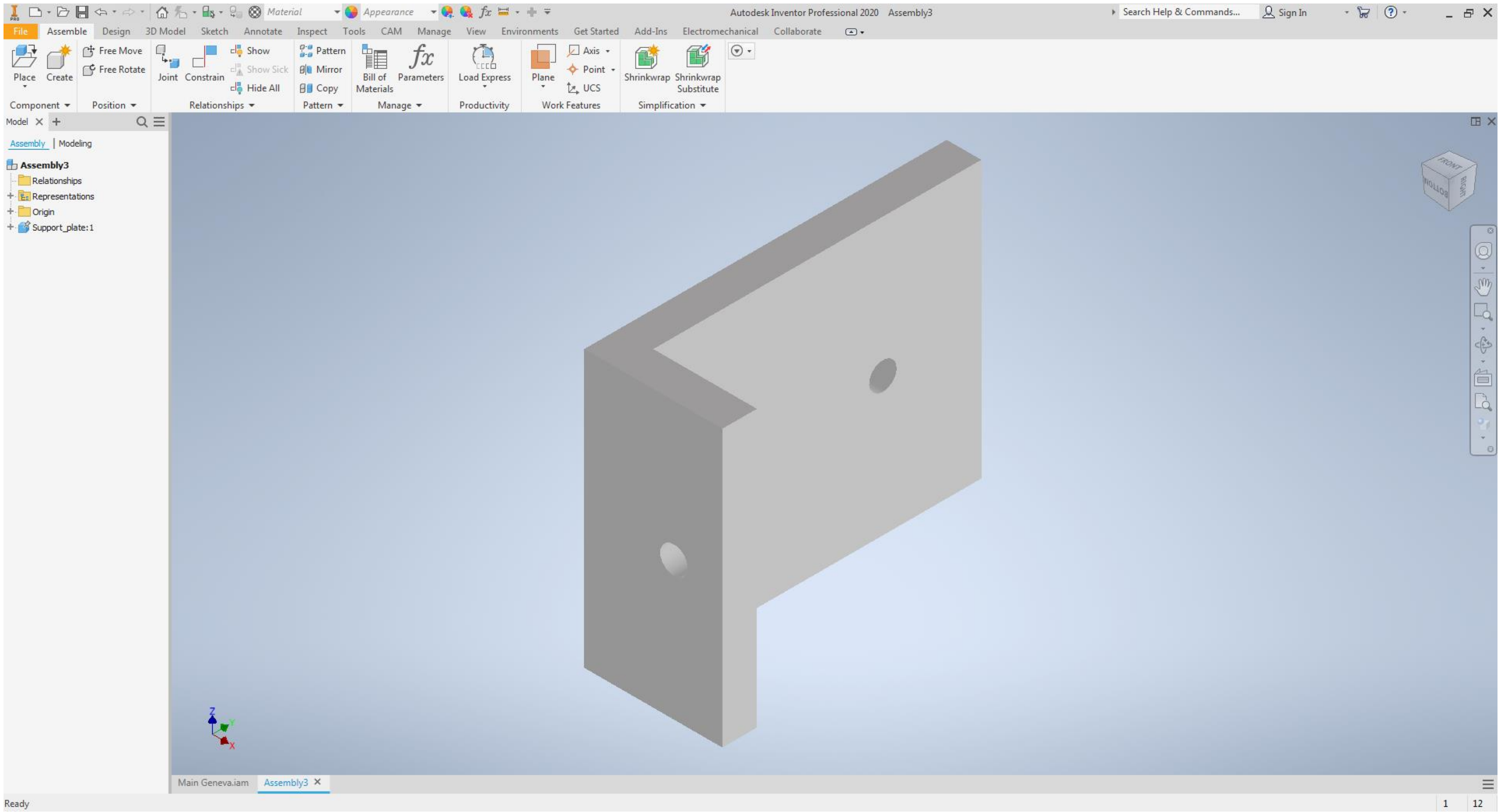


Right click and select "Place Grounded at Origin" to fix the position of the part at the origin

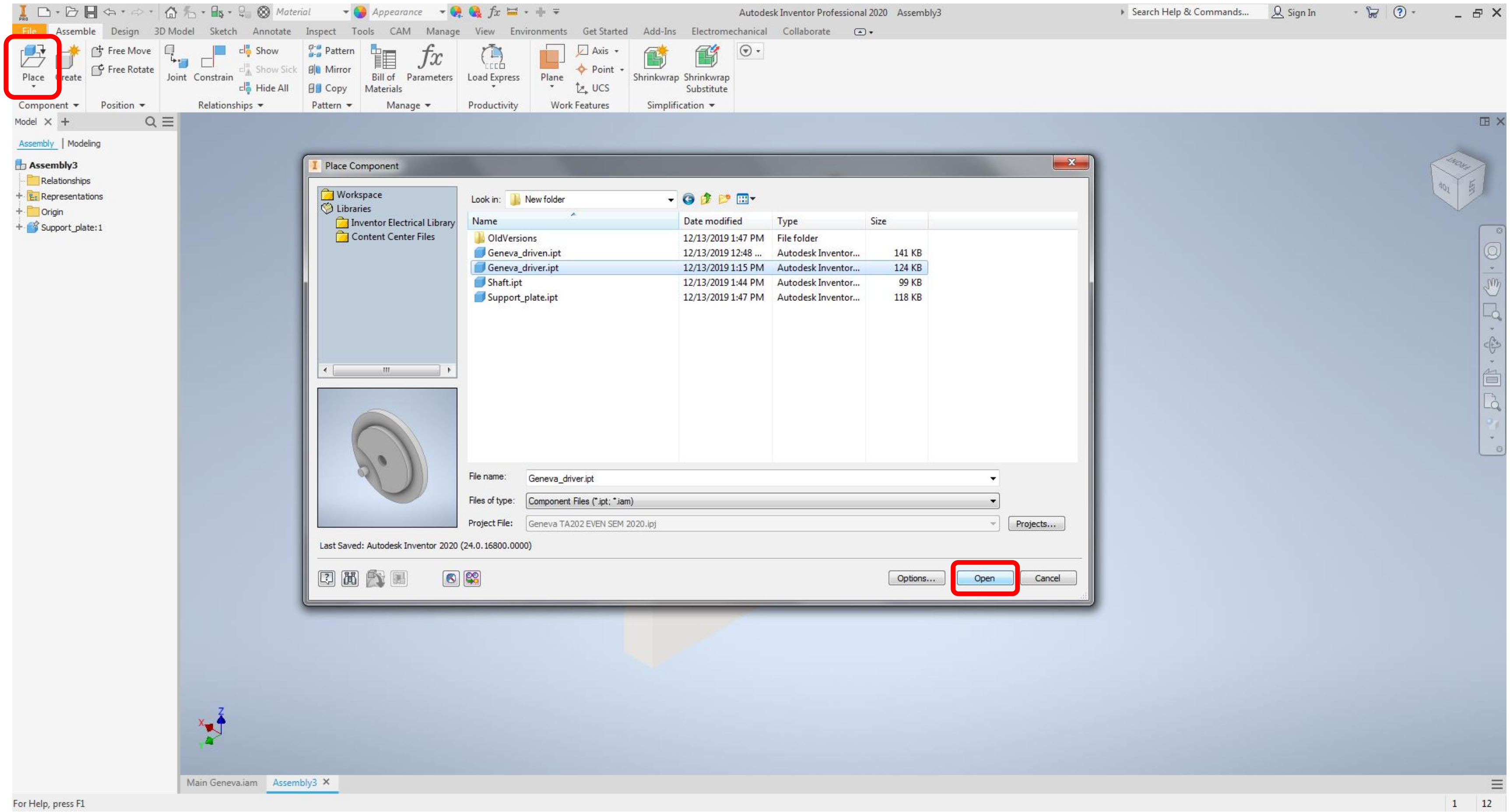
Press "Esc"



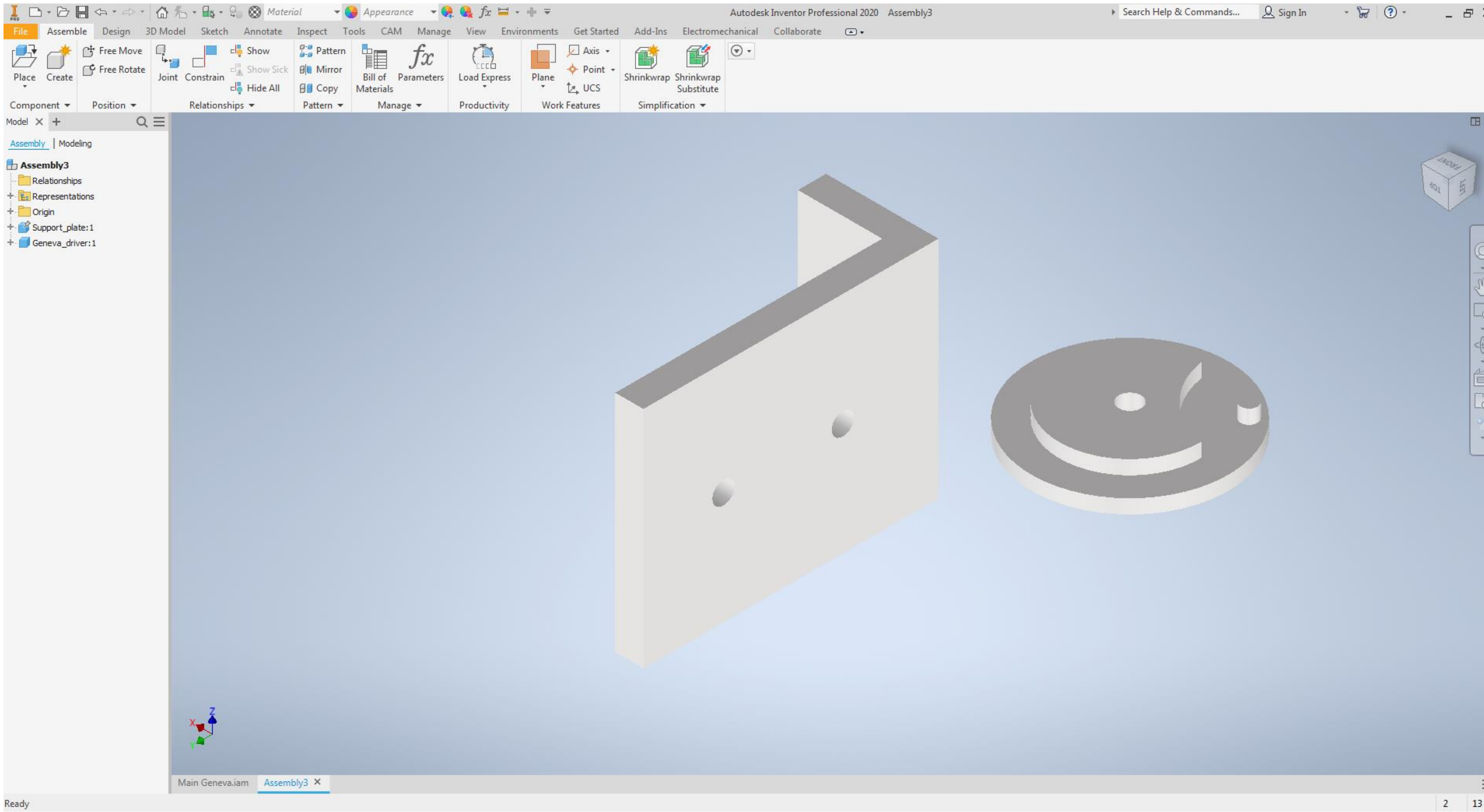
Your screen should look like this once you have placed the part



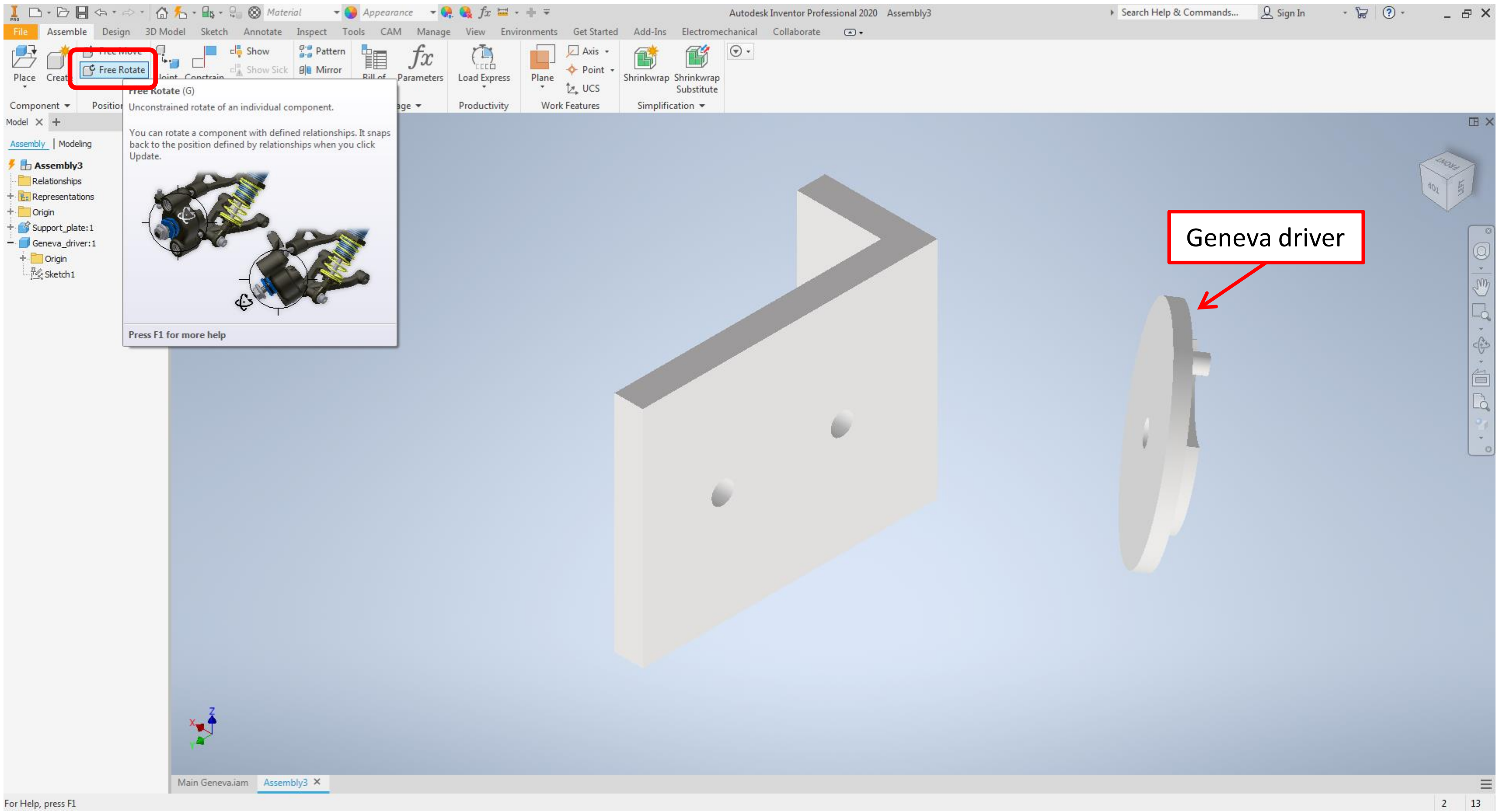
Click on “Place”, select “Geneva driver” and click “Open”



Click on the screen where the part is to be placed and press “Esc”
Geneva driver will be placed in the assembly

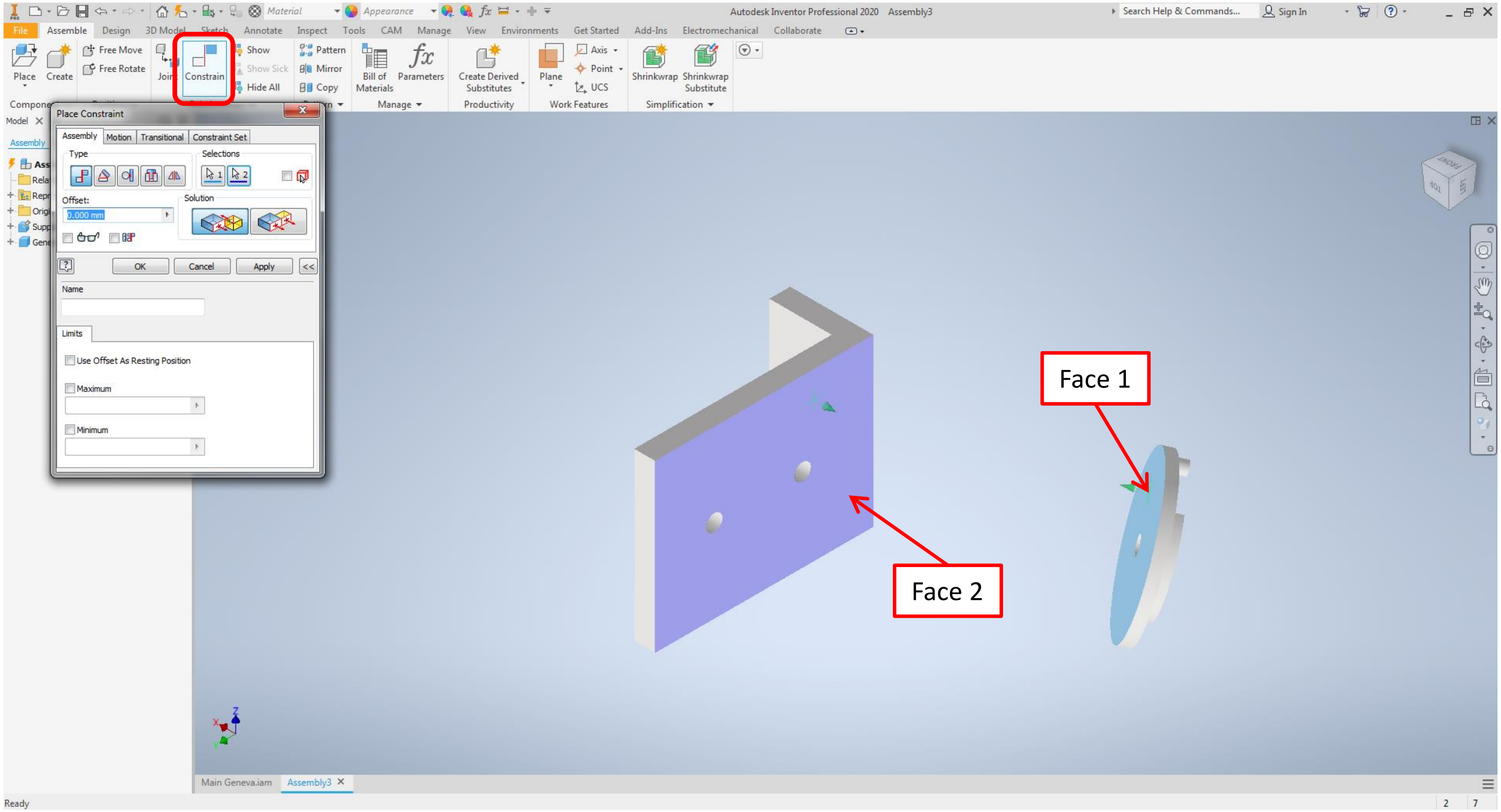


Click on “Free Rotate” and rotate the Geneva driver in order to align it as shown below

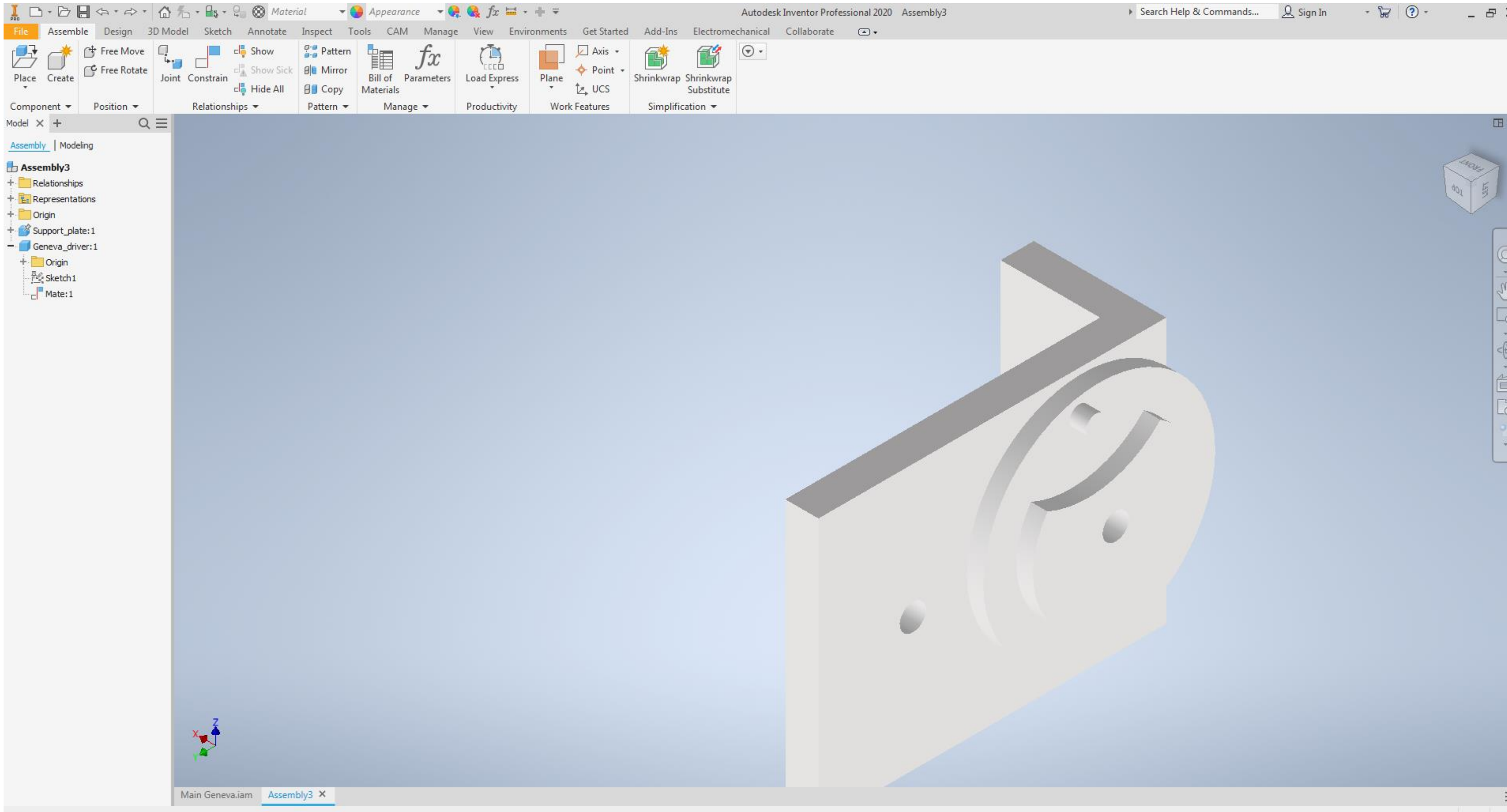


Click on “Constrain” and select the faces as shown below

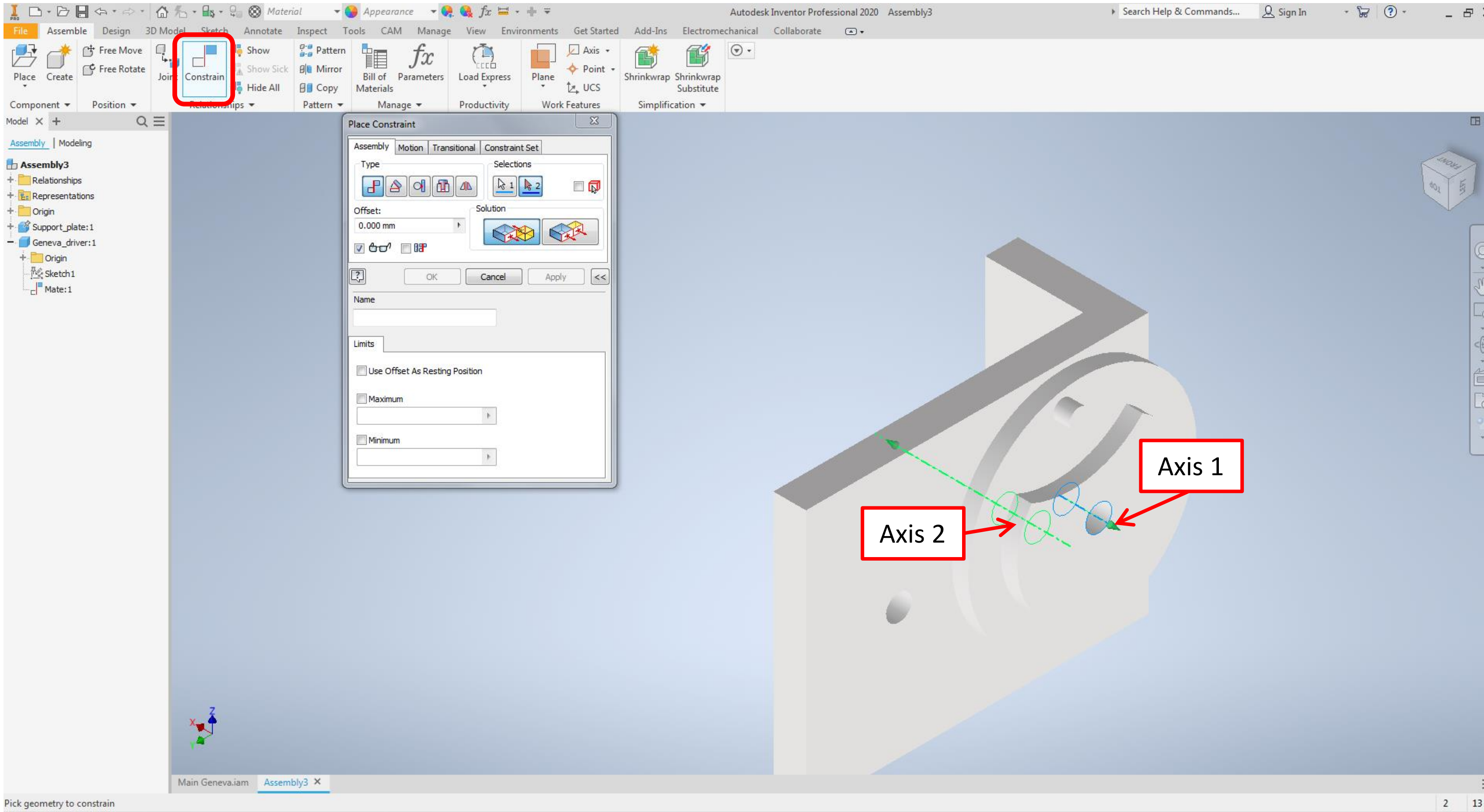
Click “OK”



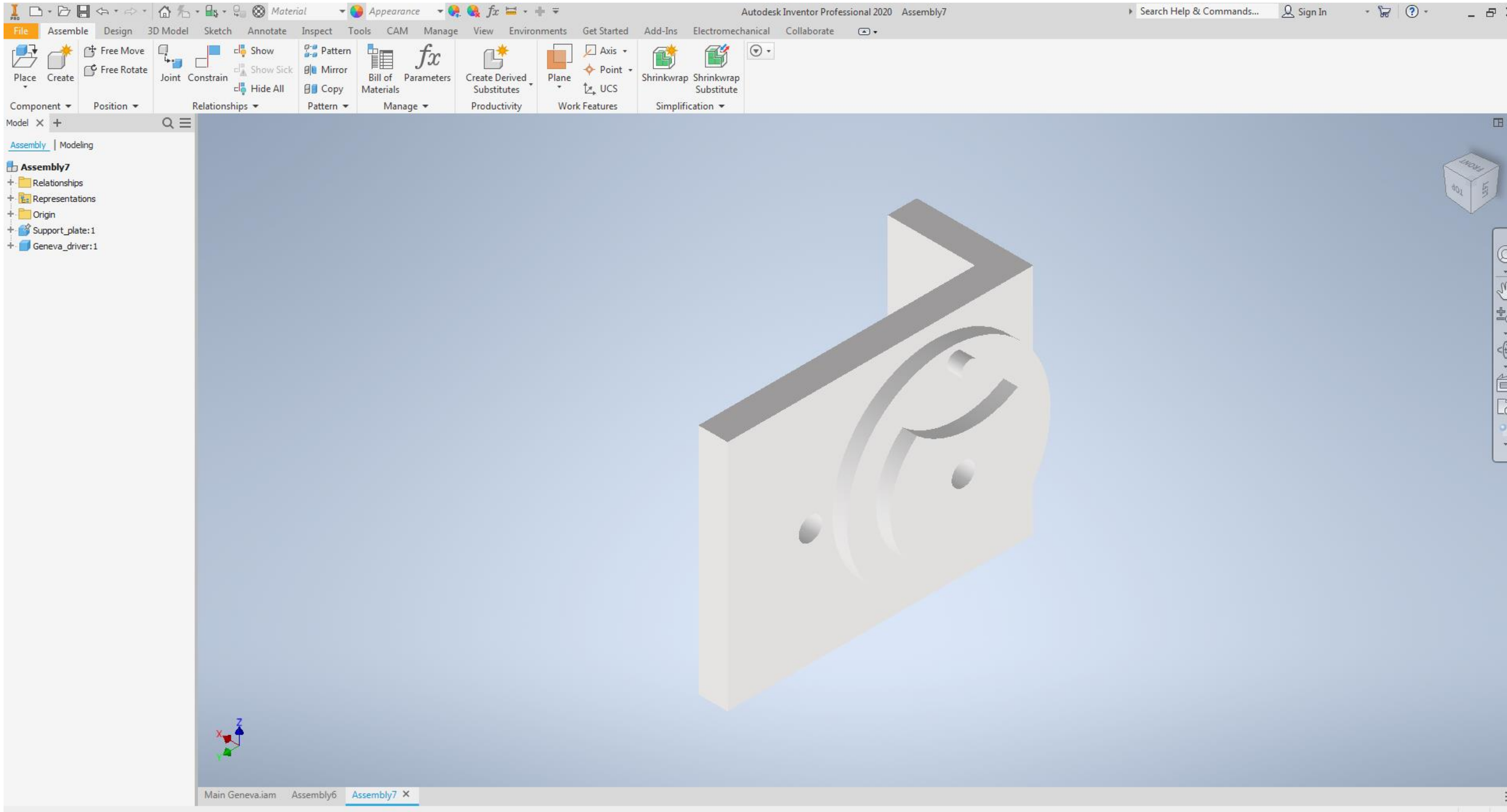
At this stage, your assembly should look like this



Click on "Constrain" select the axis of holes as shown below
Click "OK"
*The hole behind the Geneva driver may not be visible, hover the mouse pointer around the hole in order to select the hole axis on the Support plate

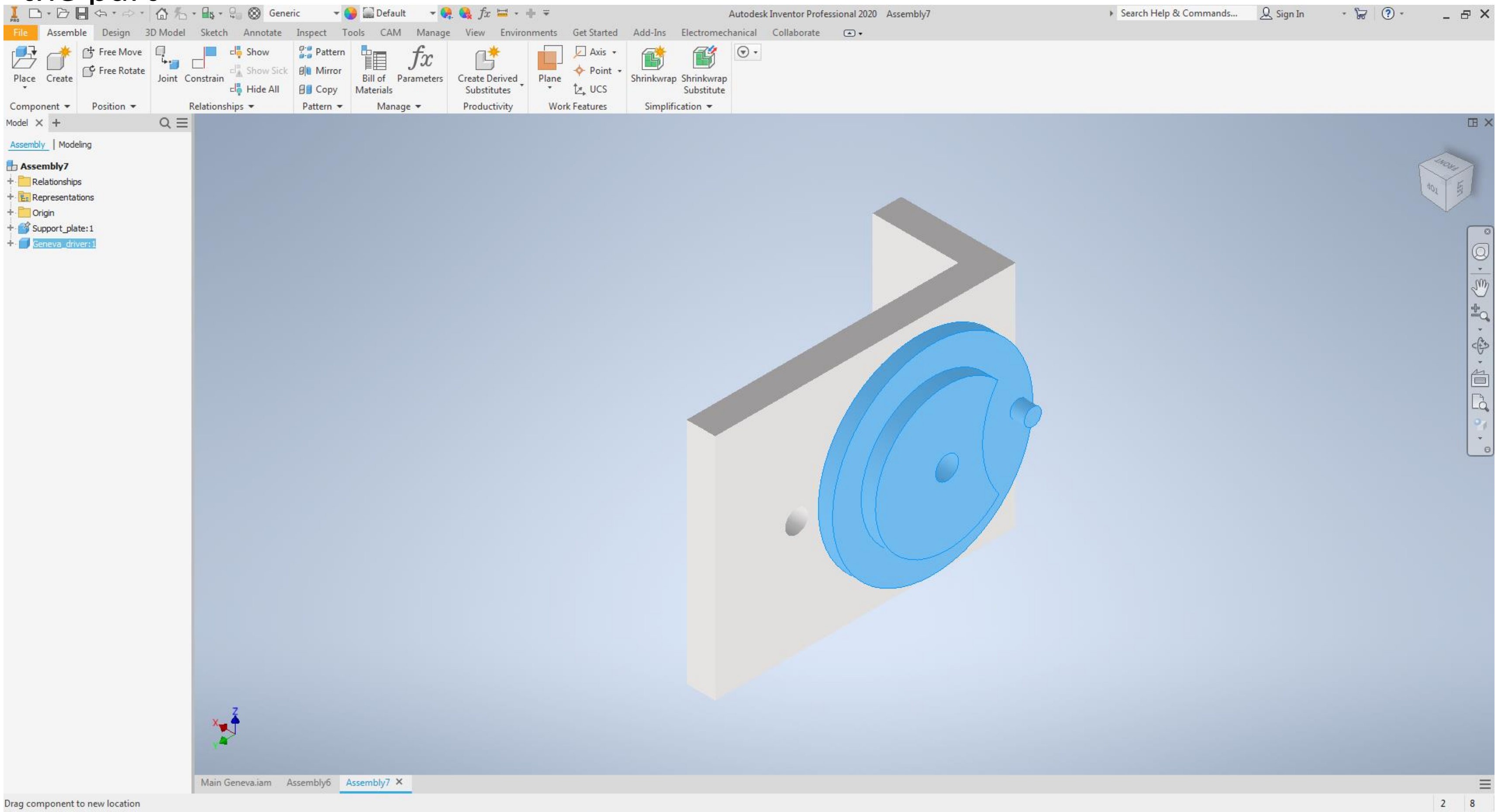


At this stage, your assembly should look like this

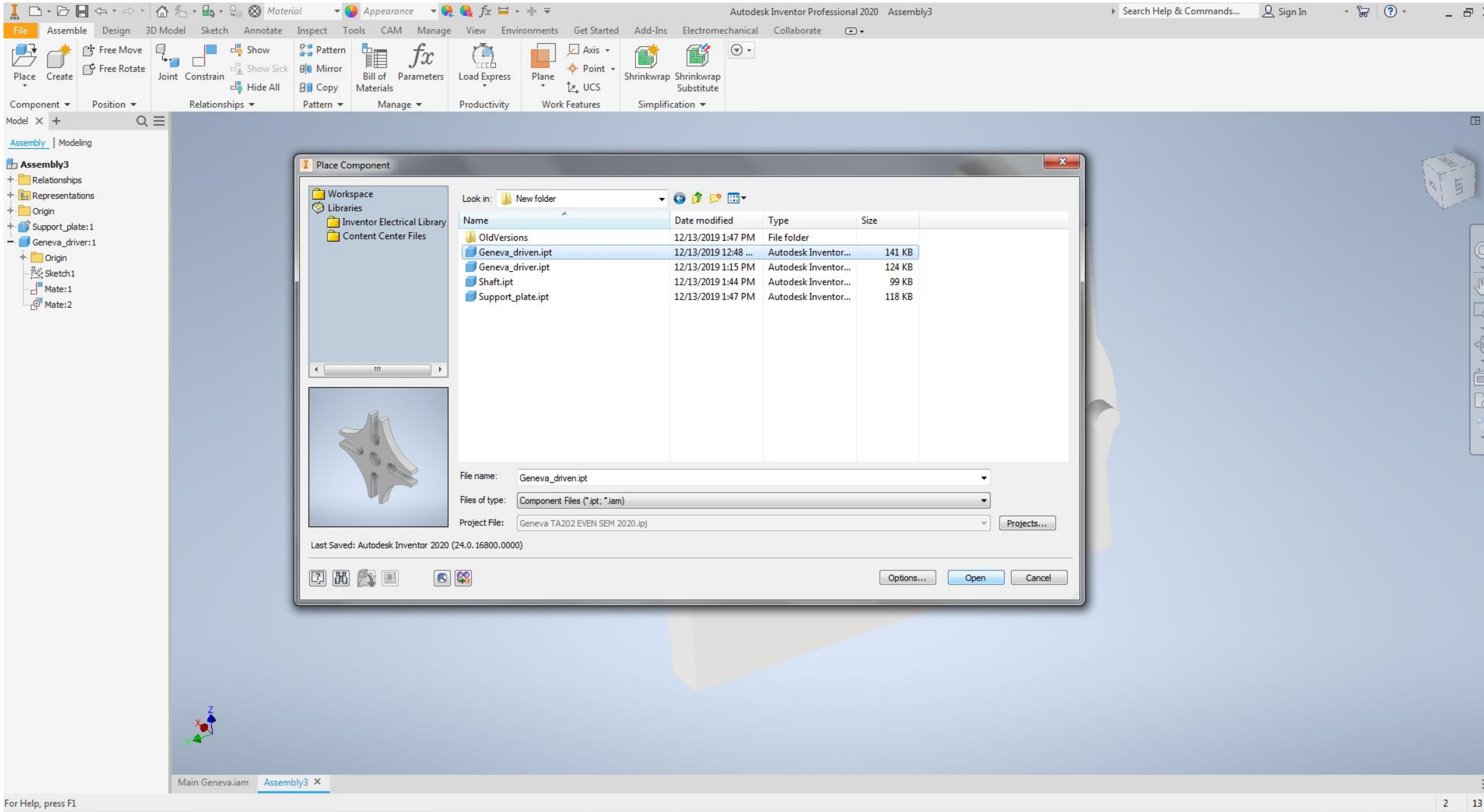


You can now rotate the Geneva Driver Wheel about its axis

*Left click on the part and hold the mouse button. Now, move your mouse to rotate the part

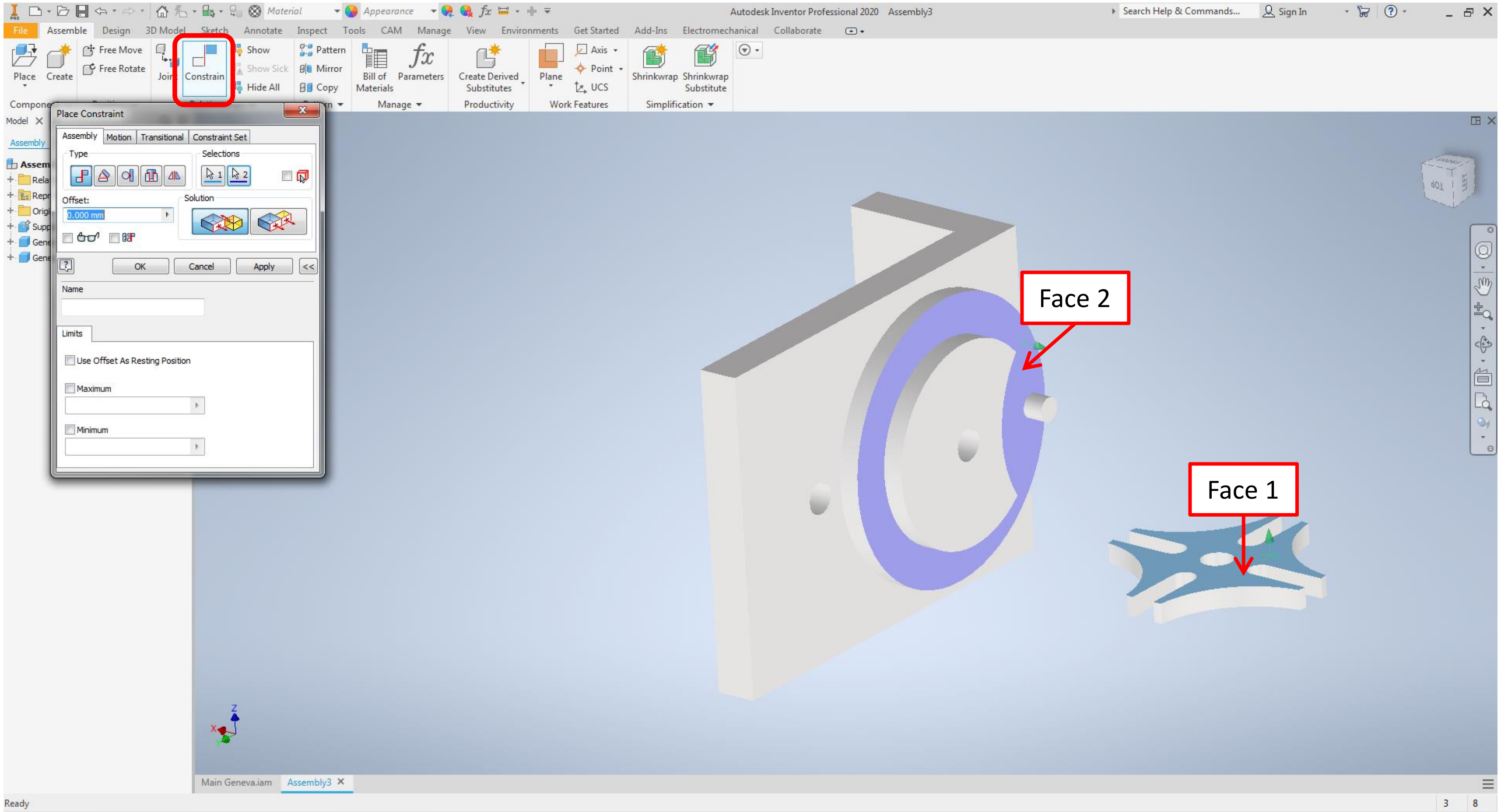


Click on “Place”, select “Geneva driven wheel” and click “Open”
Click on the screen where the part is to be placed and press “Esc”

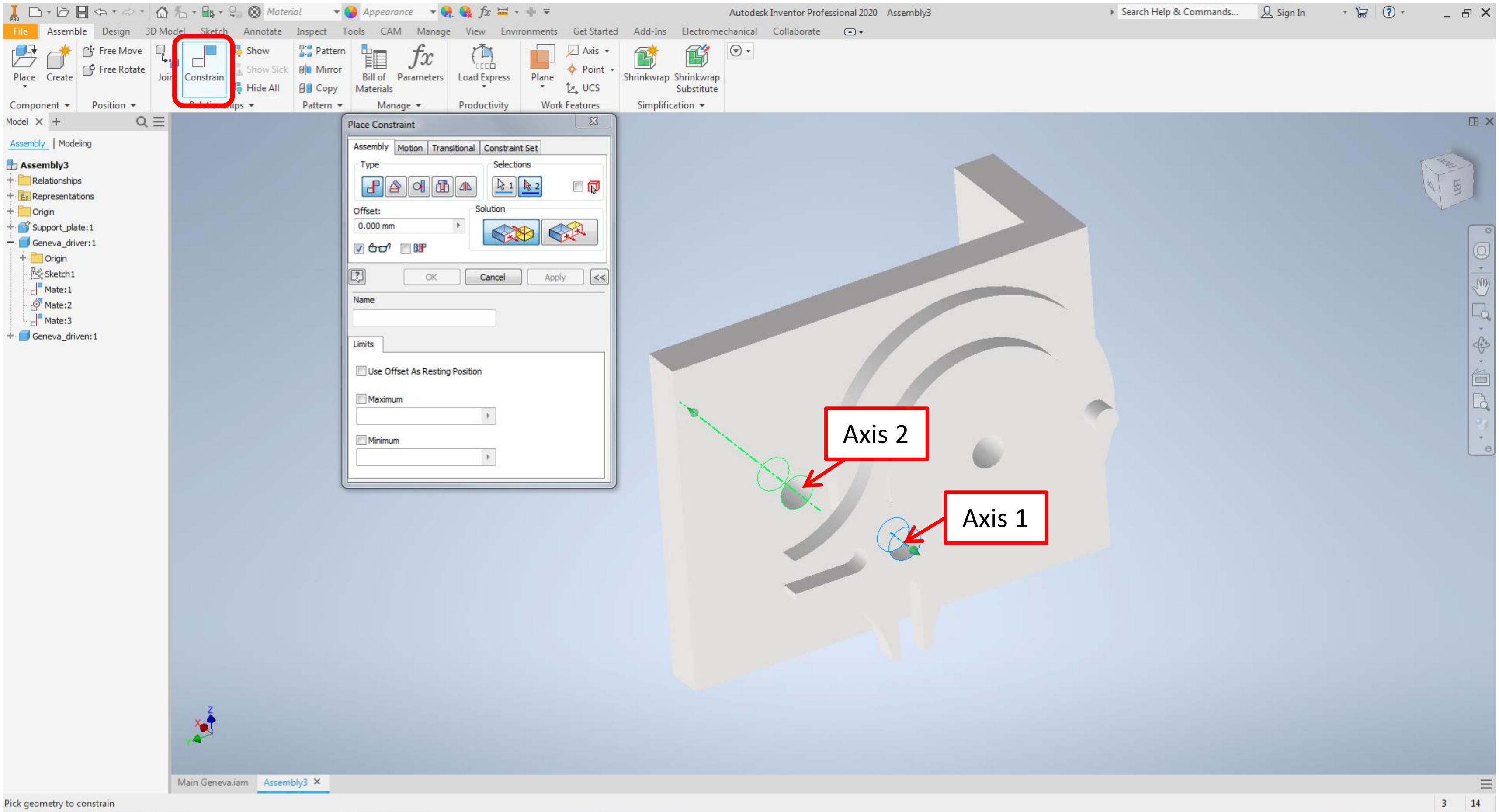


Click on “Constrain” and select the two faces as shown below

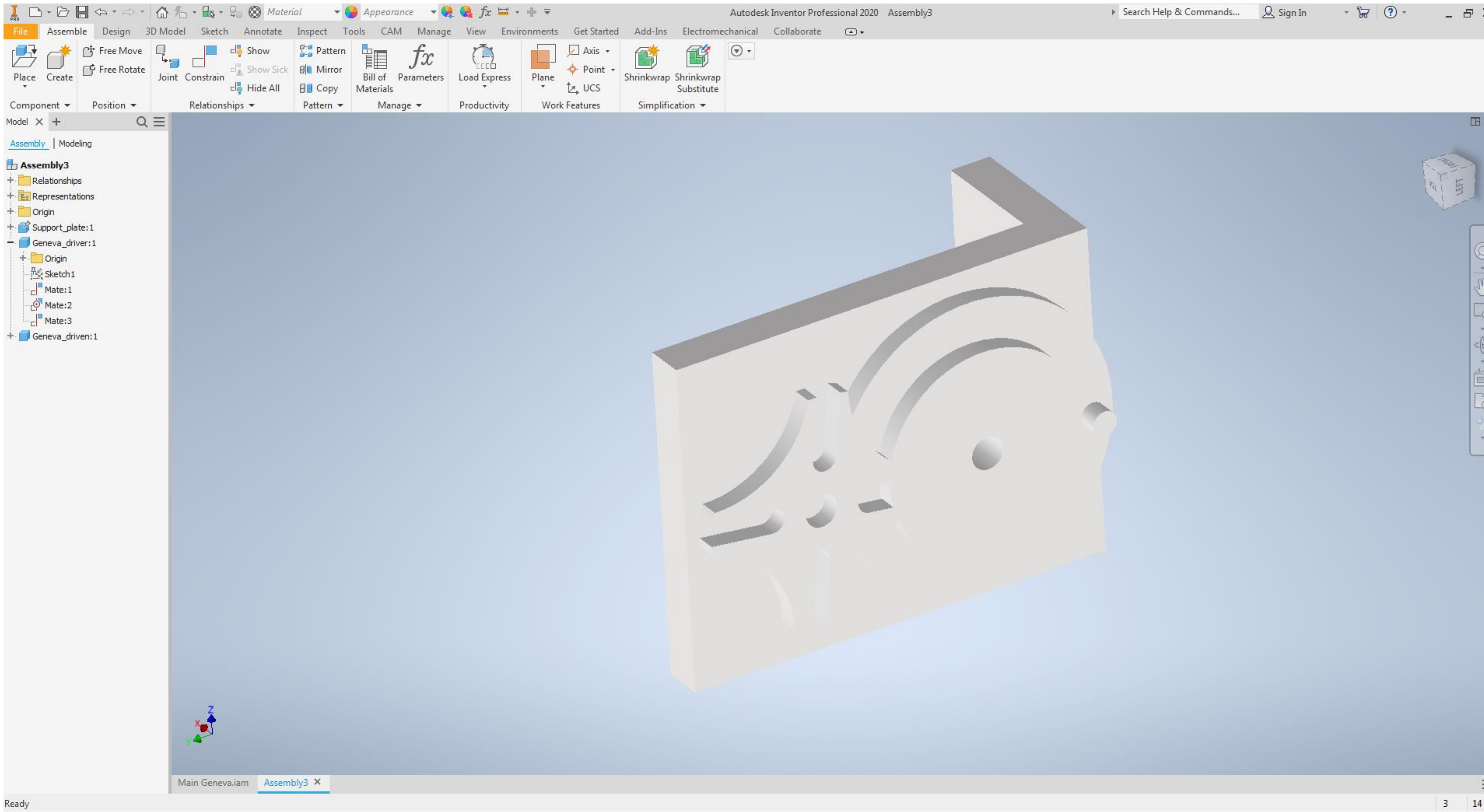
Click on “OK”



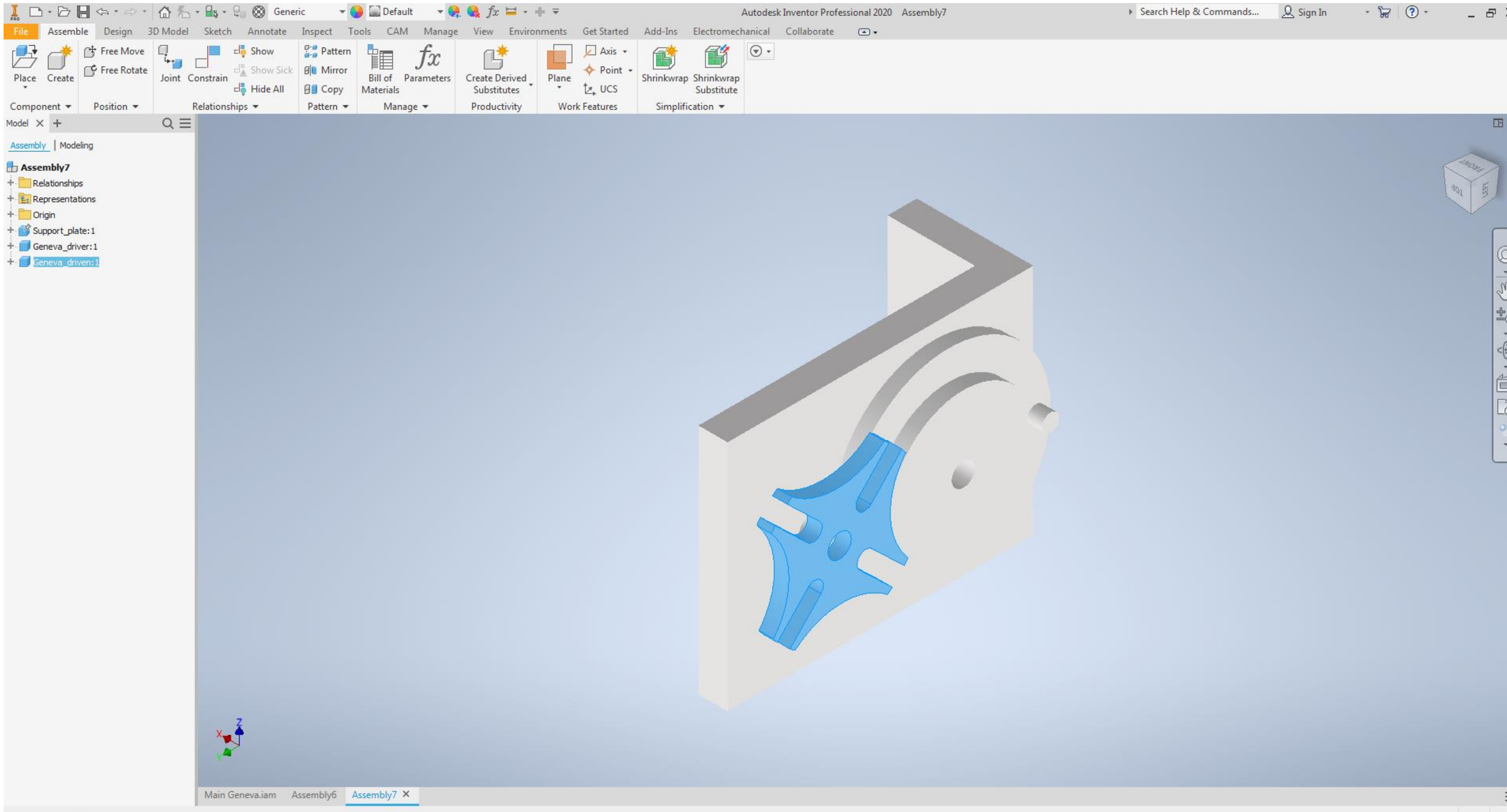
Click on “Constrain” and select the axis of the holes as shown
Click “OK”



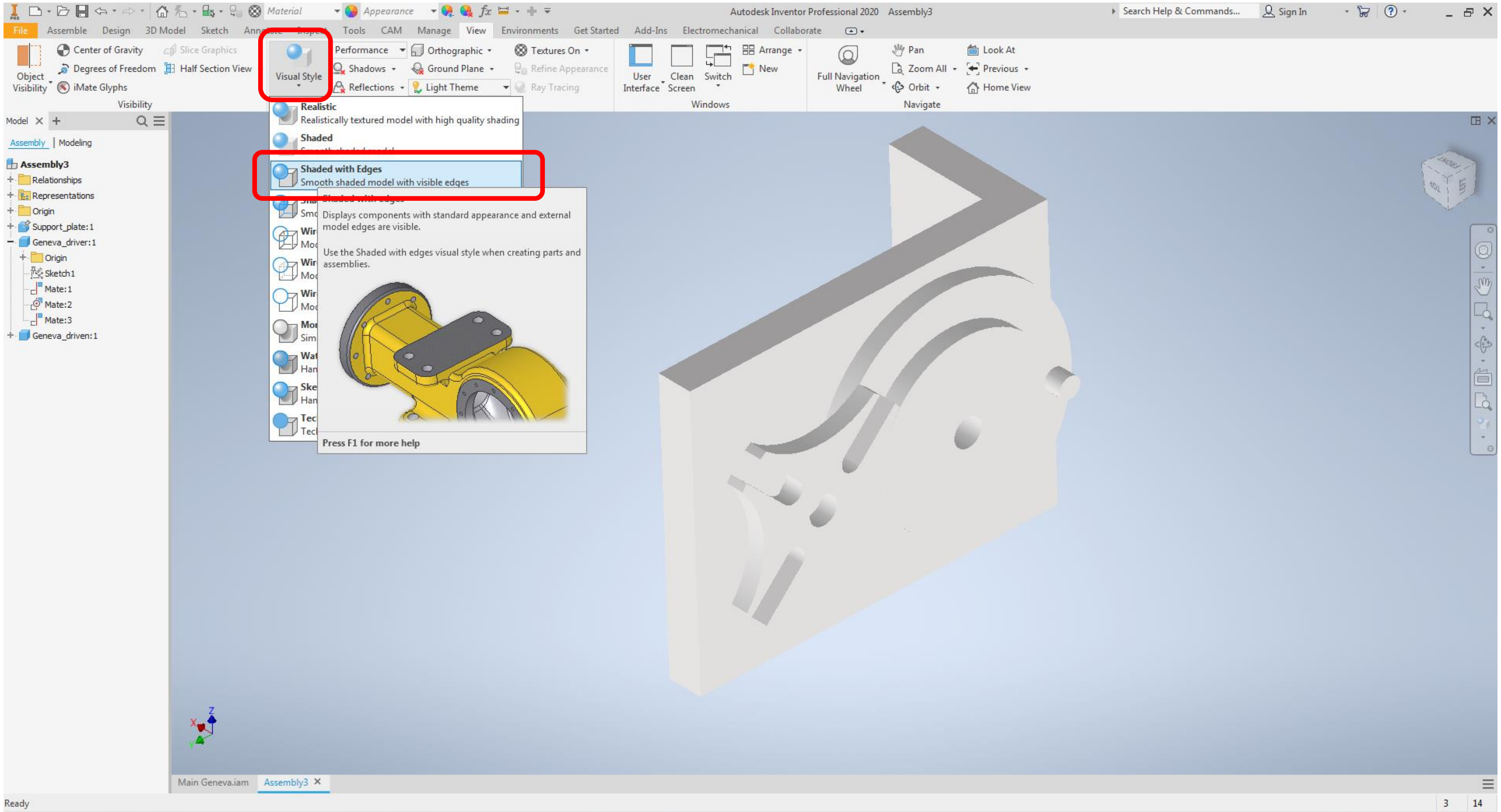
At this stage, your assembly should look like this



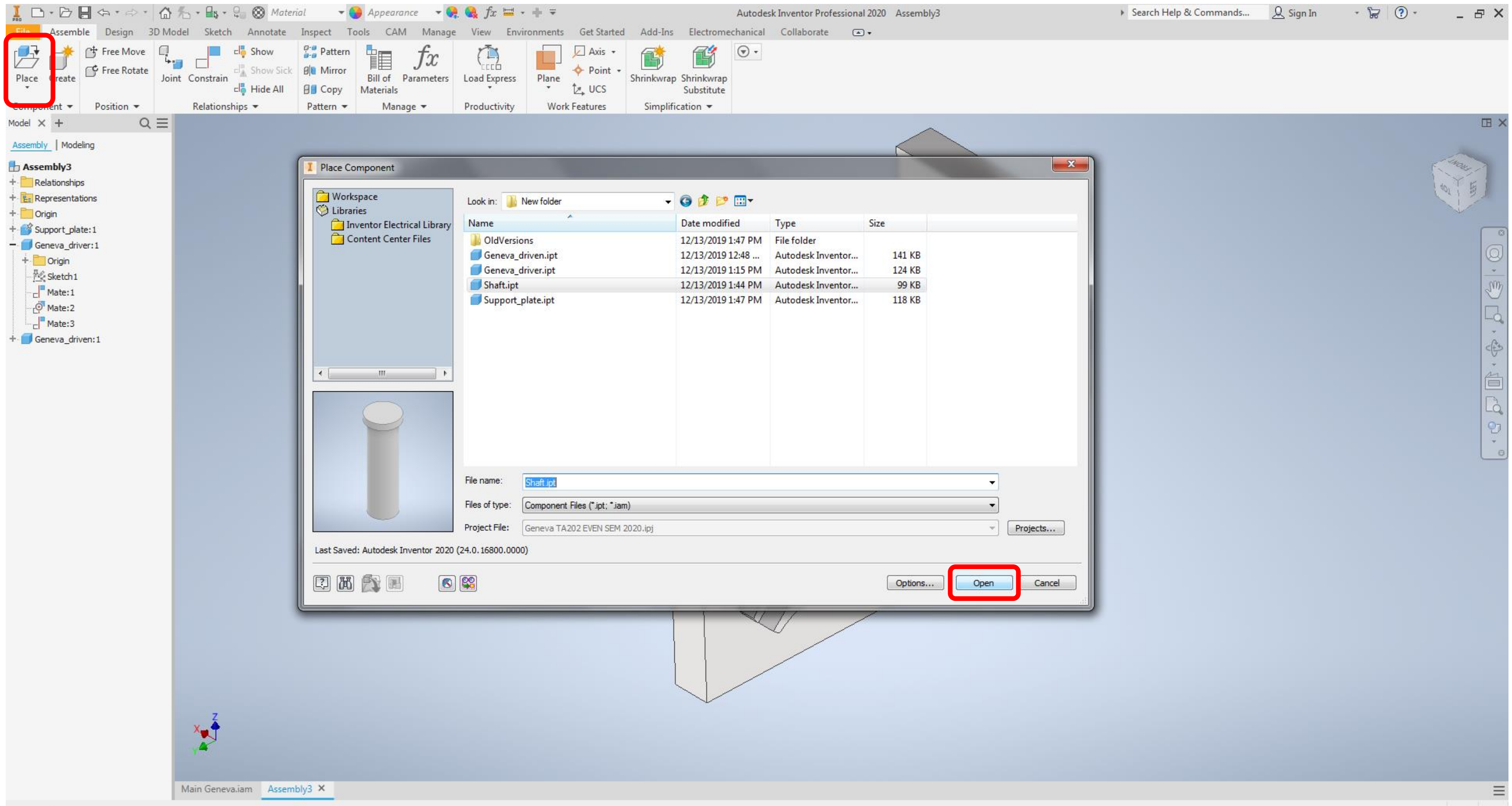
You will now be able to rotate and orient the Driven Wheel about it’s axis as shown below



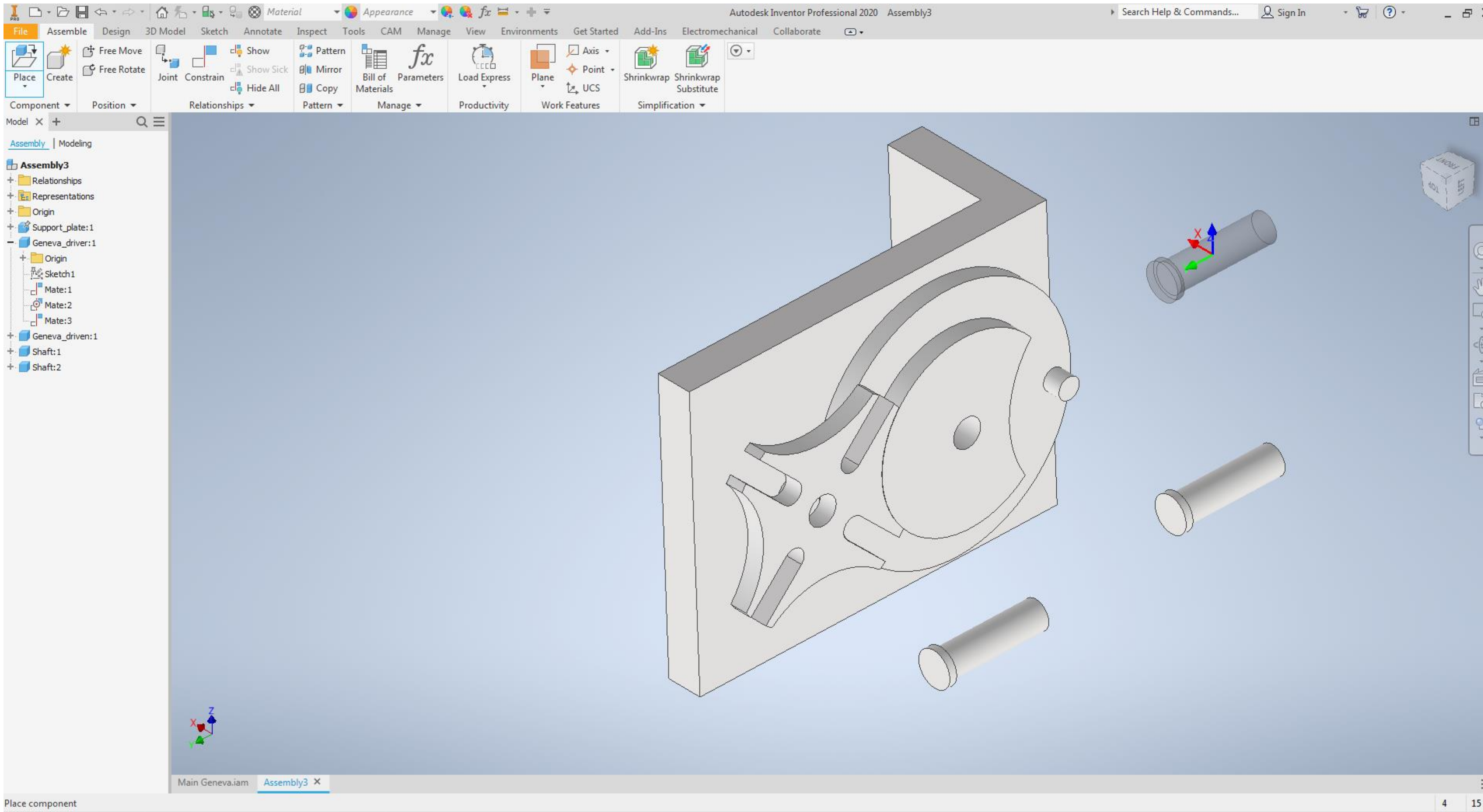
You can click on “View” and change View Style to “Shaded with Edges” for better visibility



Click on “Place”, select the “Shaft” and click “Open”

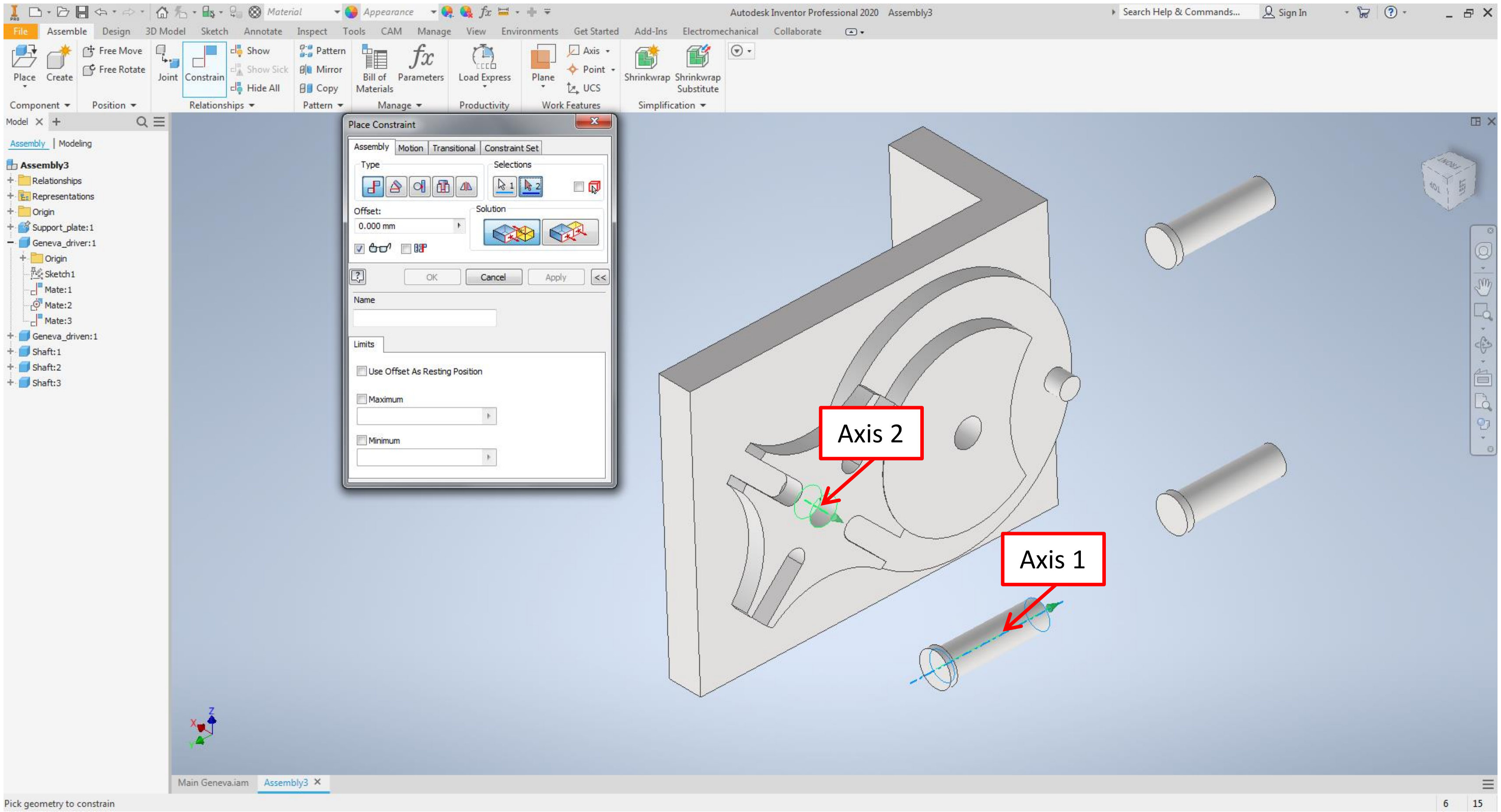


Click on the screen where the part is to be placed
Click at three different places on the screen as shown below and press "Esc"

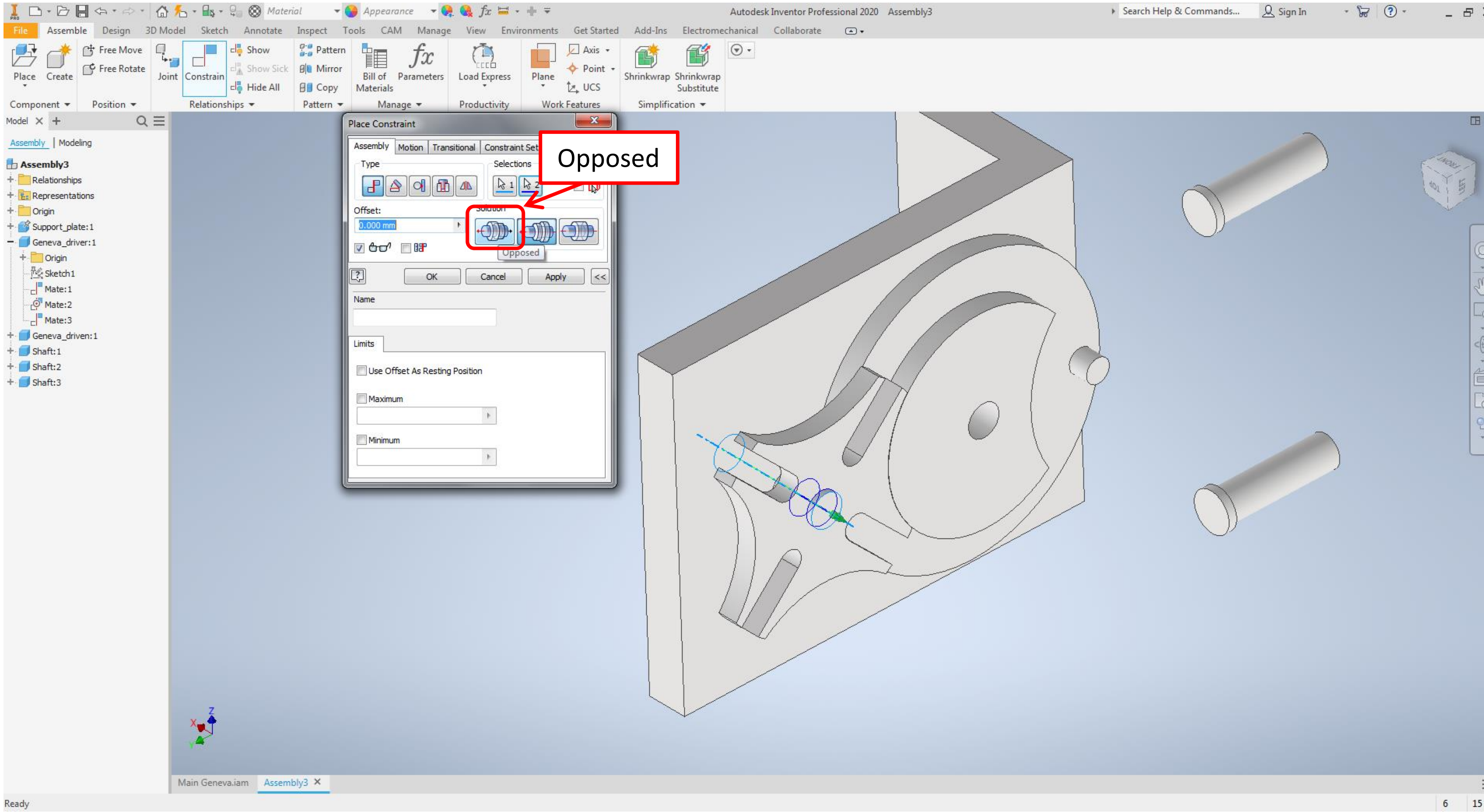


Click on “Constrain” and select the axis of the shaft and axis of hole in the driven wheel as shown

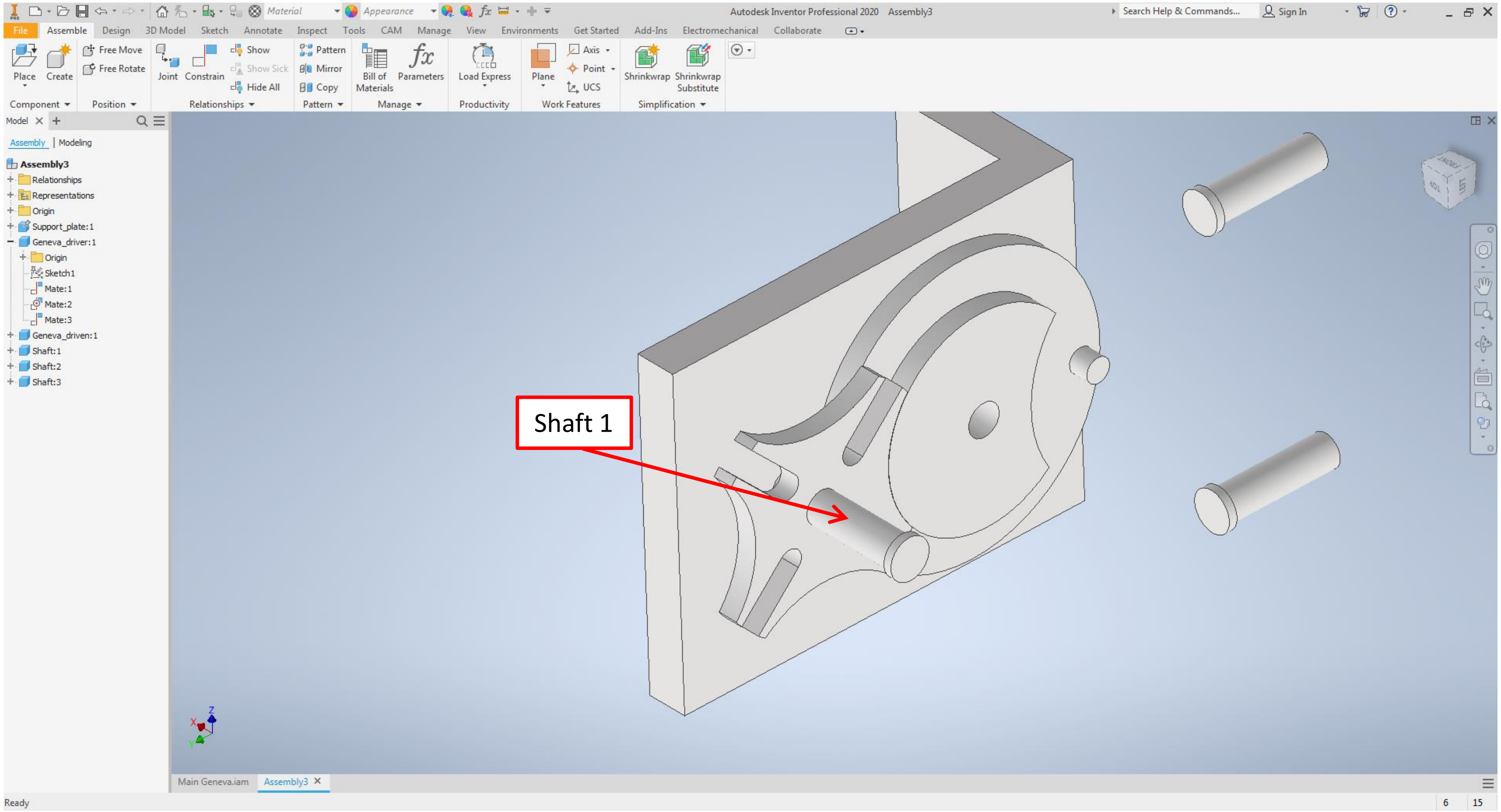
Click “OK”



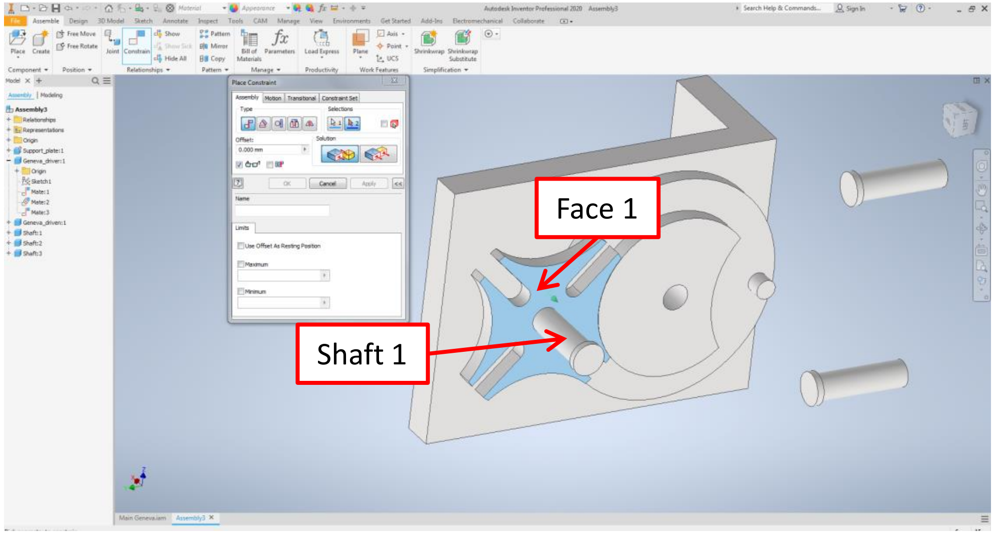
If the shaft is inverted as shown, click on “Opposed” to set the orientation right
Click “OK”

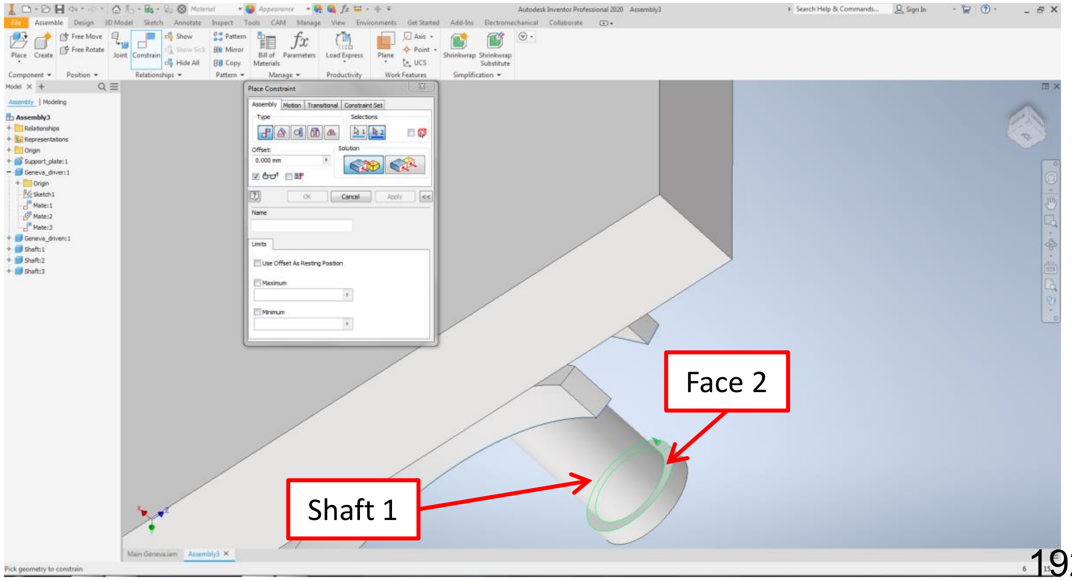


Check for orientation of Shaft 1

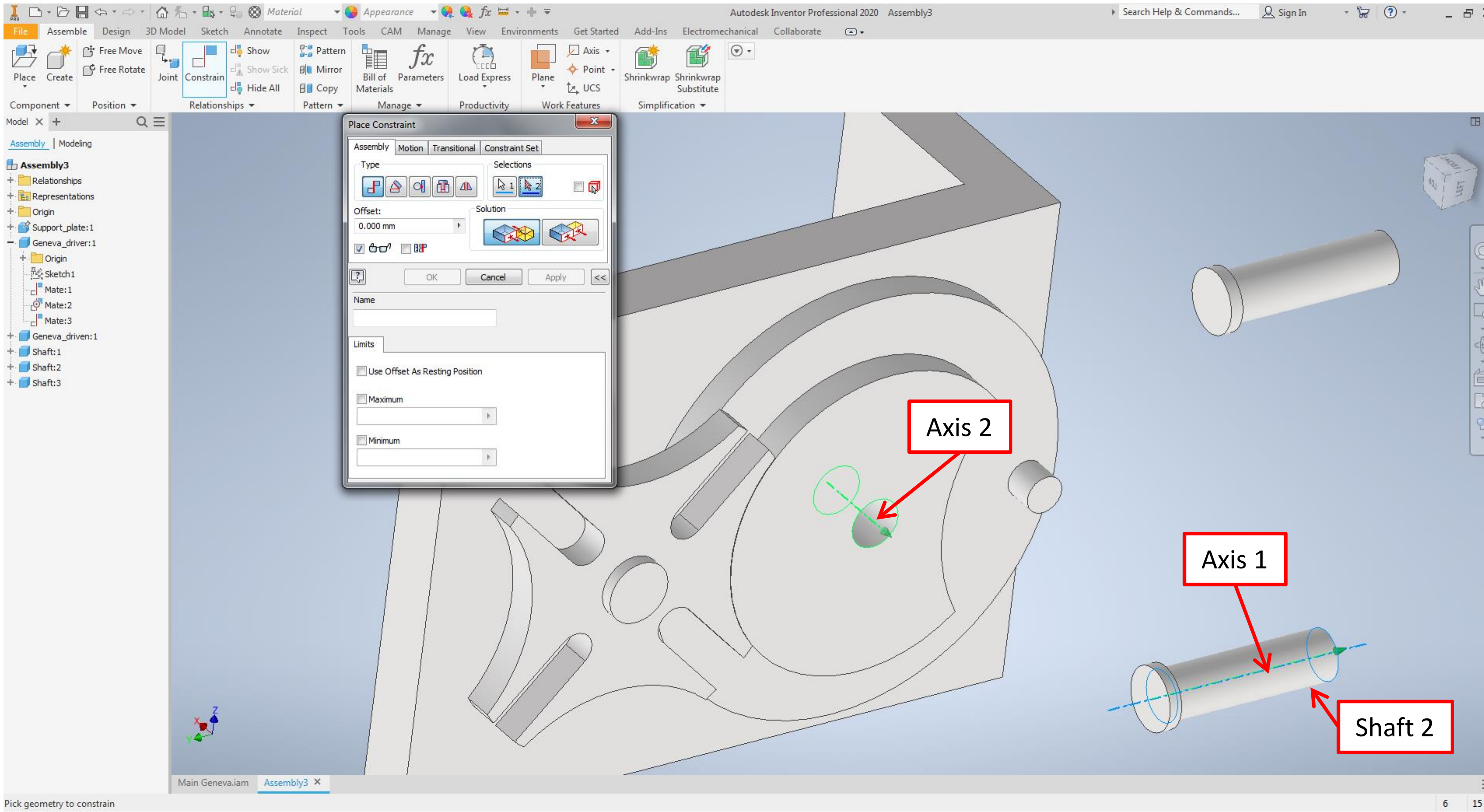


Click on “Constrain” and select the two faces as shown below
Click “OK”

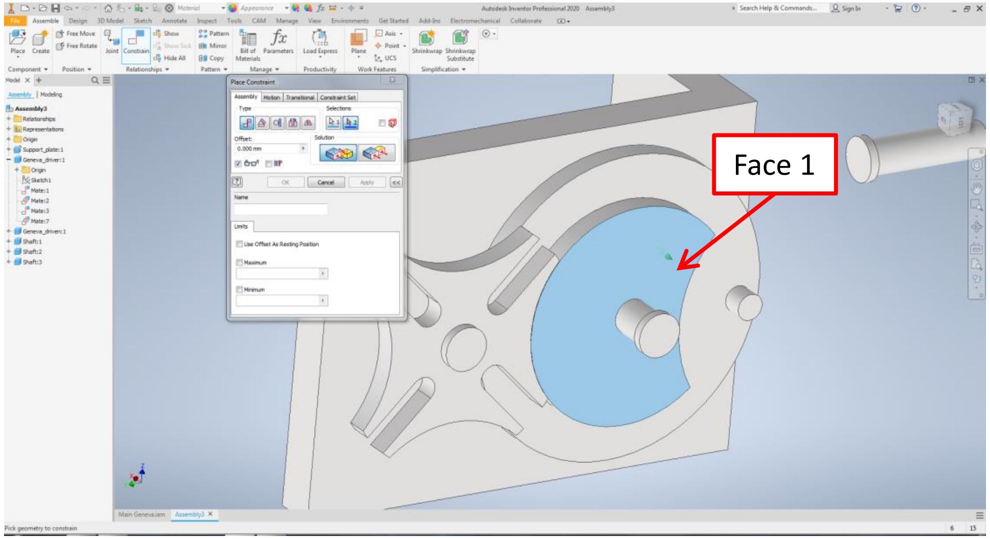


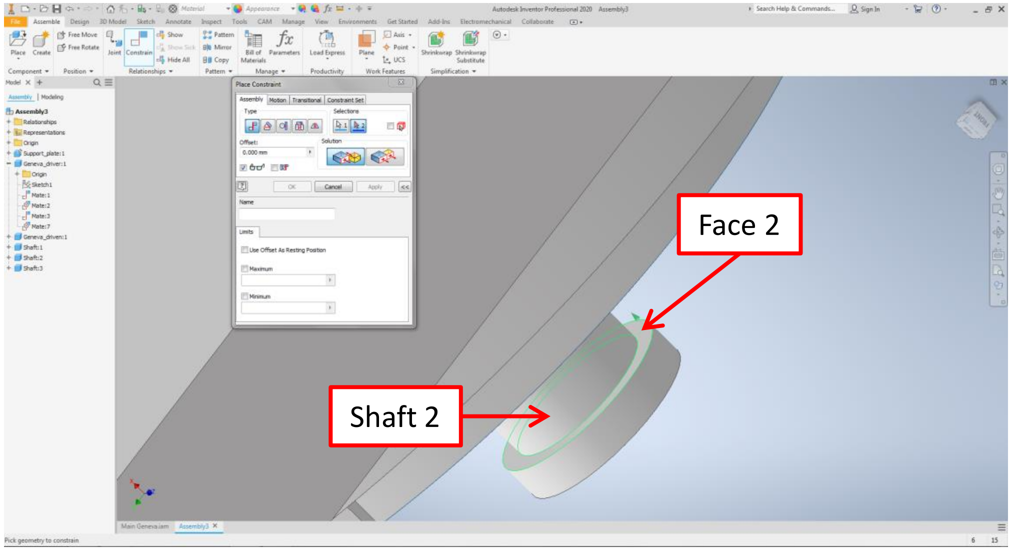


Click on “Constrain” and select the axis of Shaft 2 and axis of hole in the driver as shown
Click “OK”

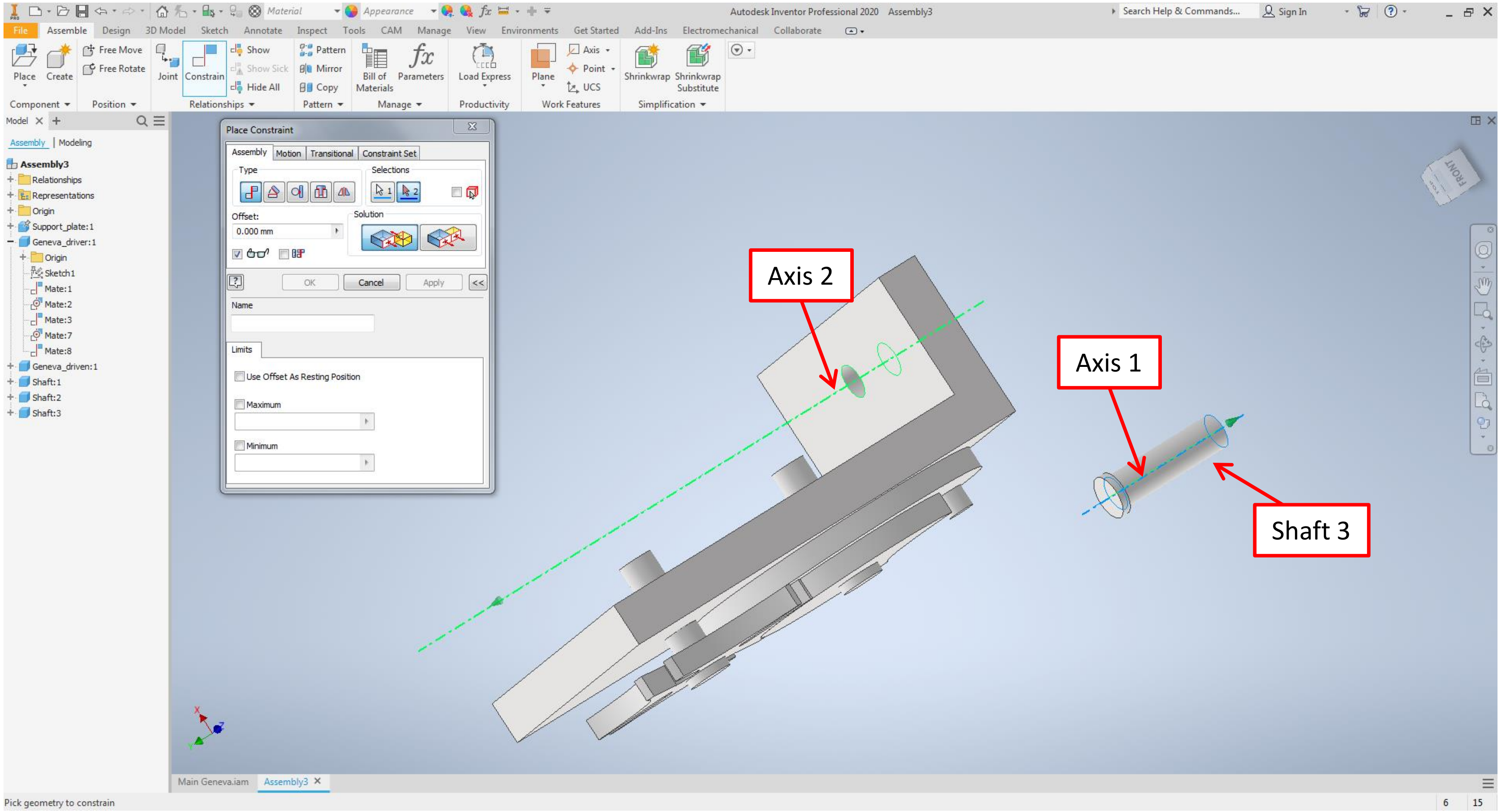


Click on “Constrain” and select the two faces as shown below
Click “OK”

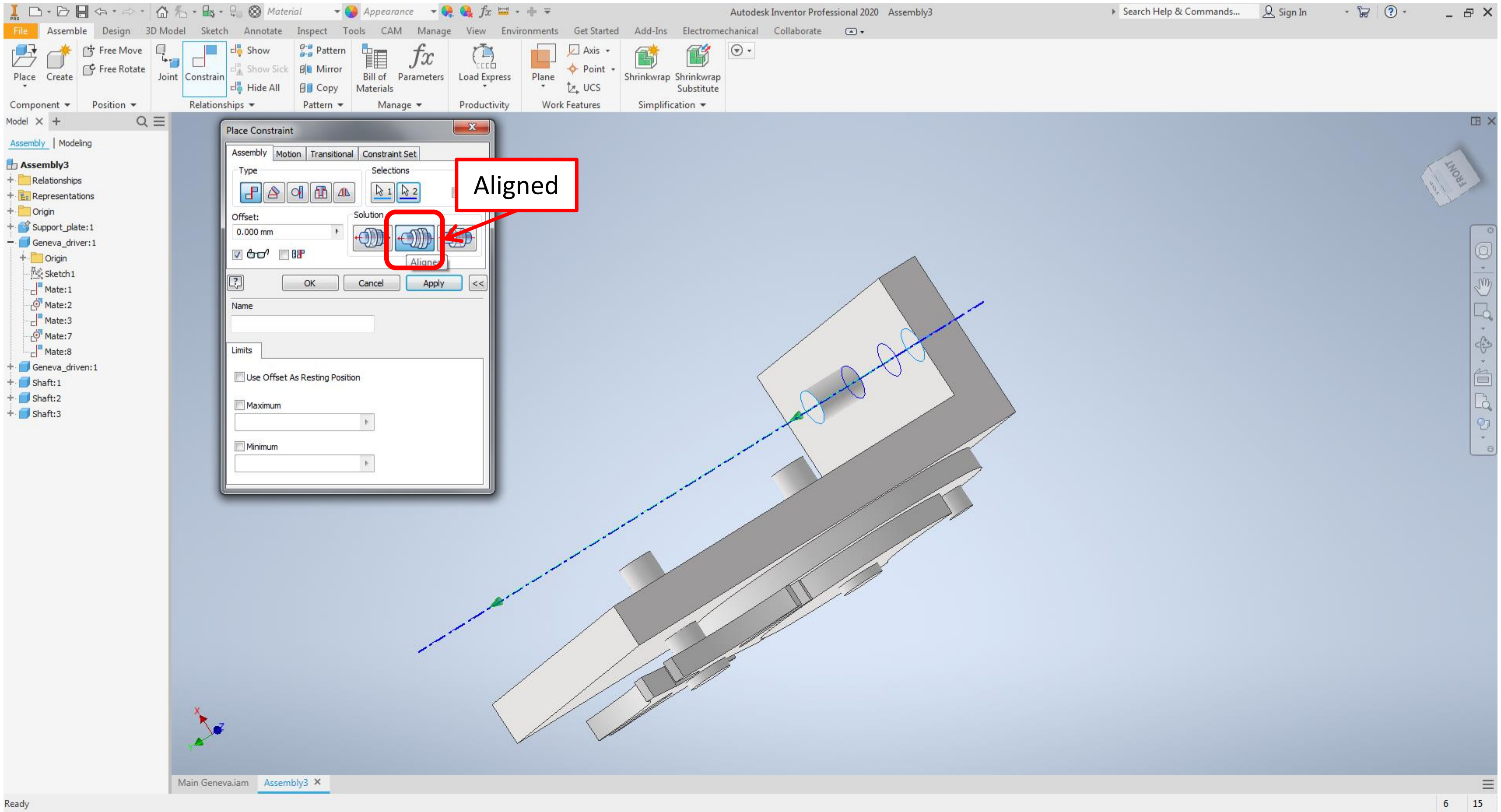




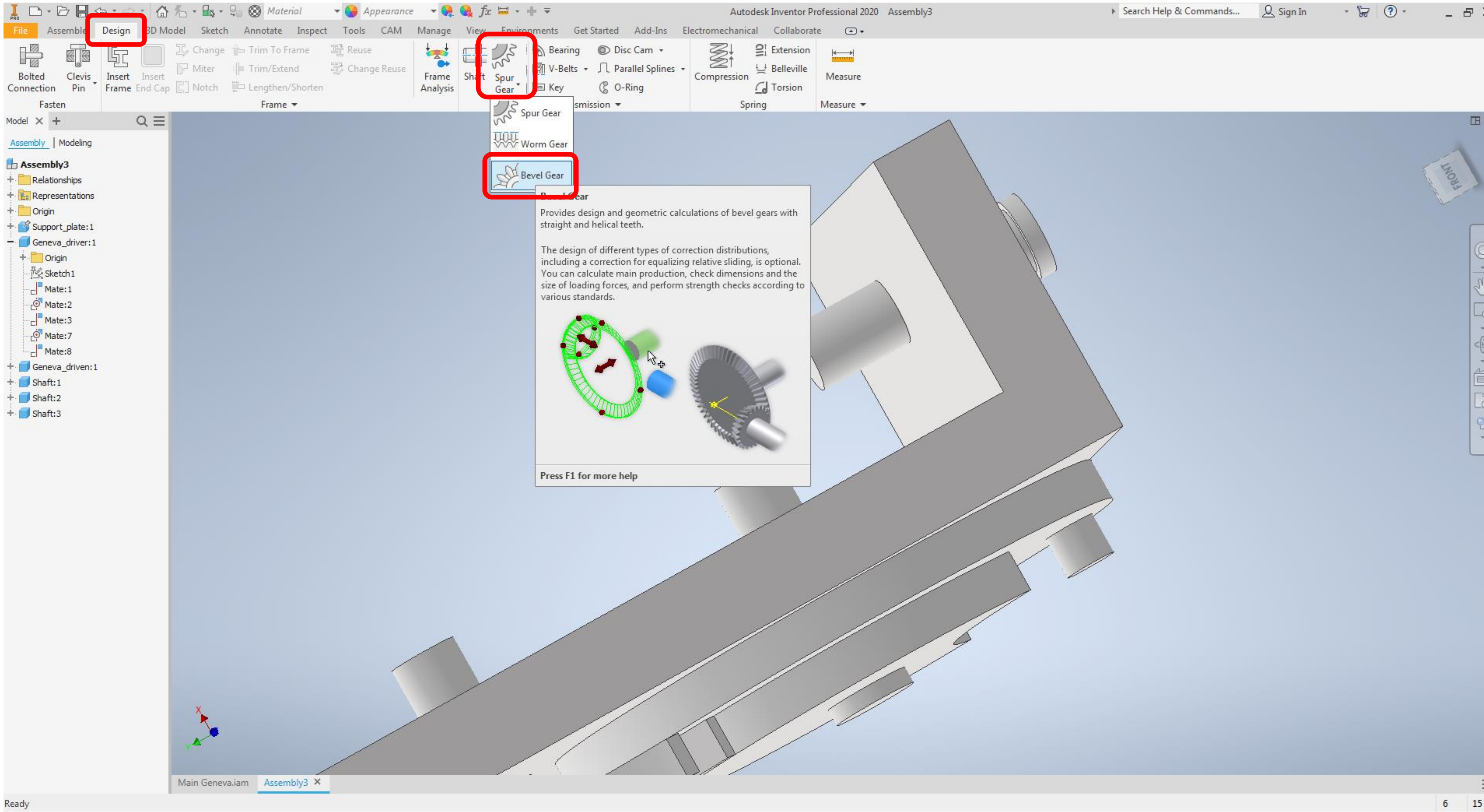
Click on “Constrain” select the axis of Shaft 3 and hole on the Support plate as shown below



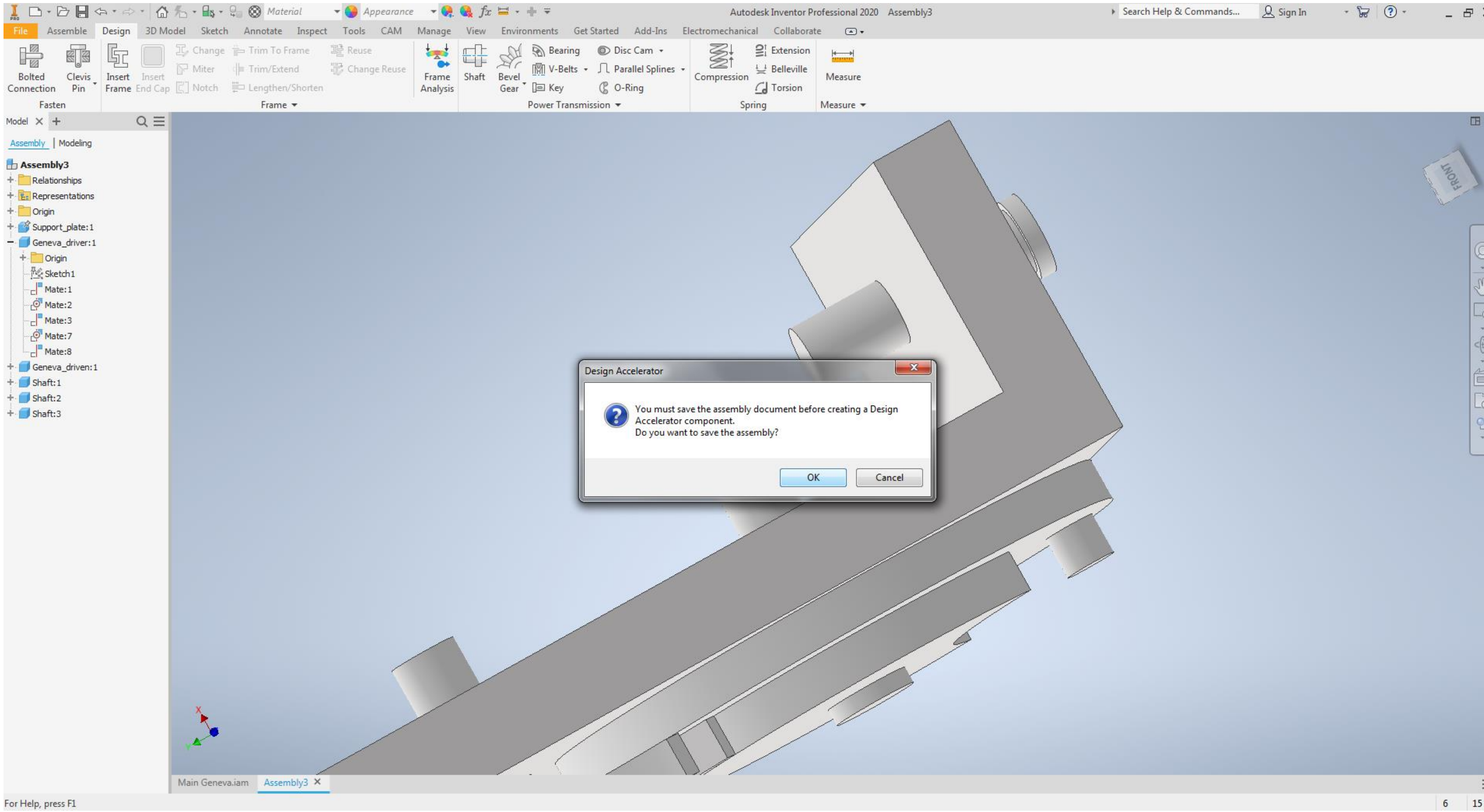
Click on “Aligned” to get the orientation right
Click “OK”



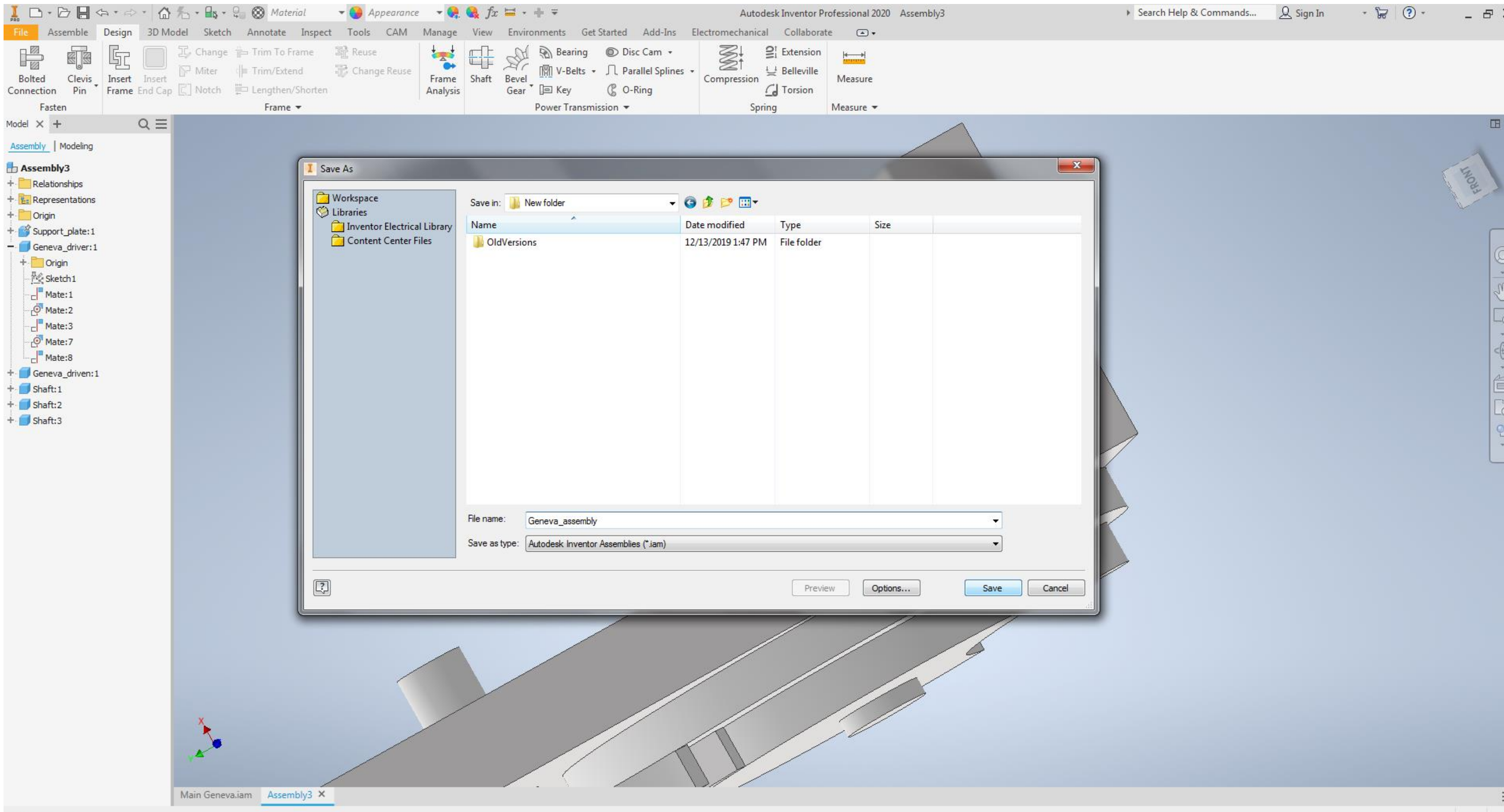
Click on “Design” and click on “Spur Gear” drop down
Select “Bevel Gear”



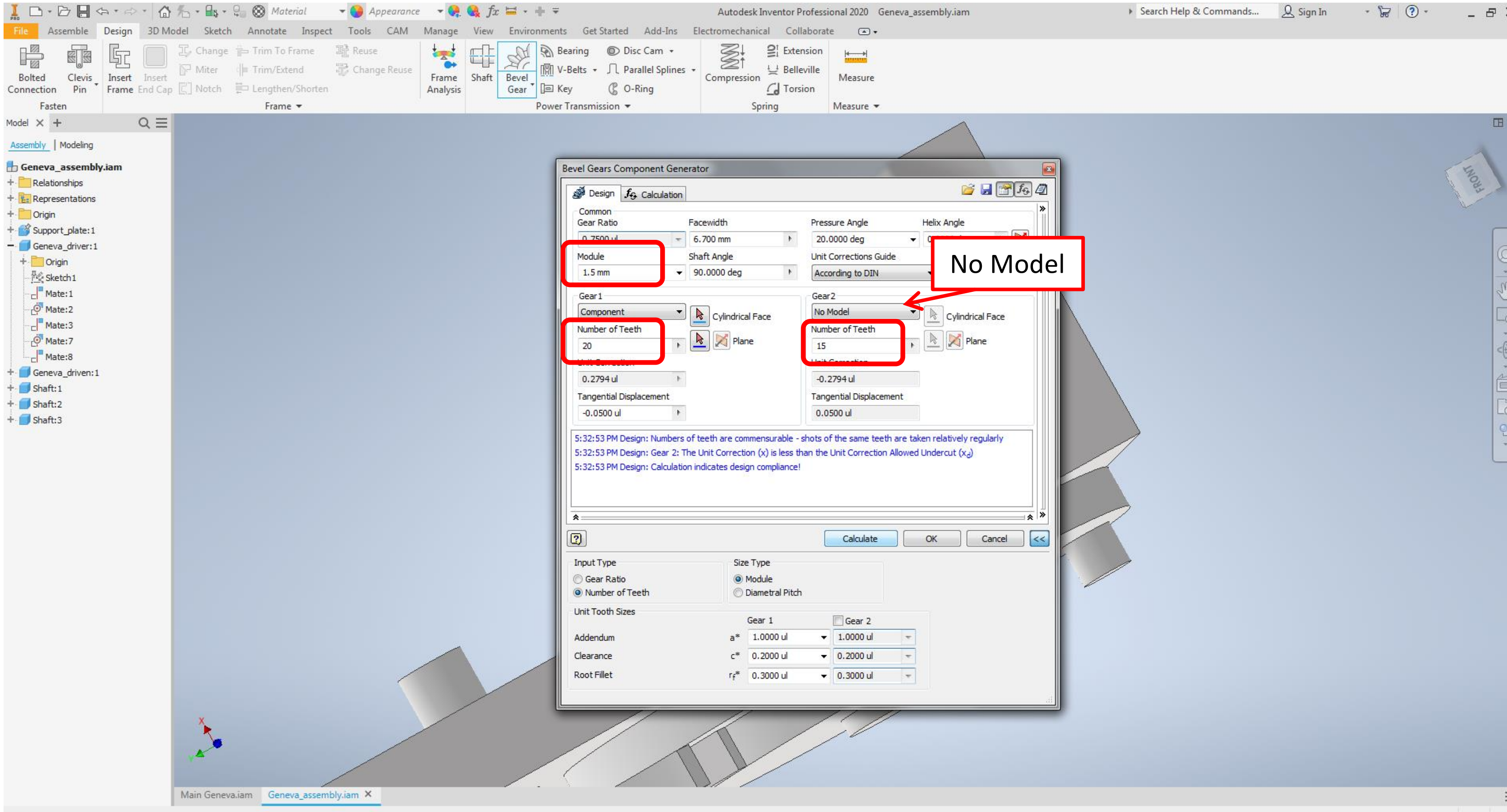
Click on “OK” and save the Assembly



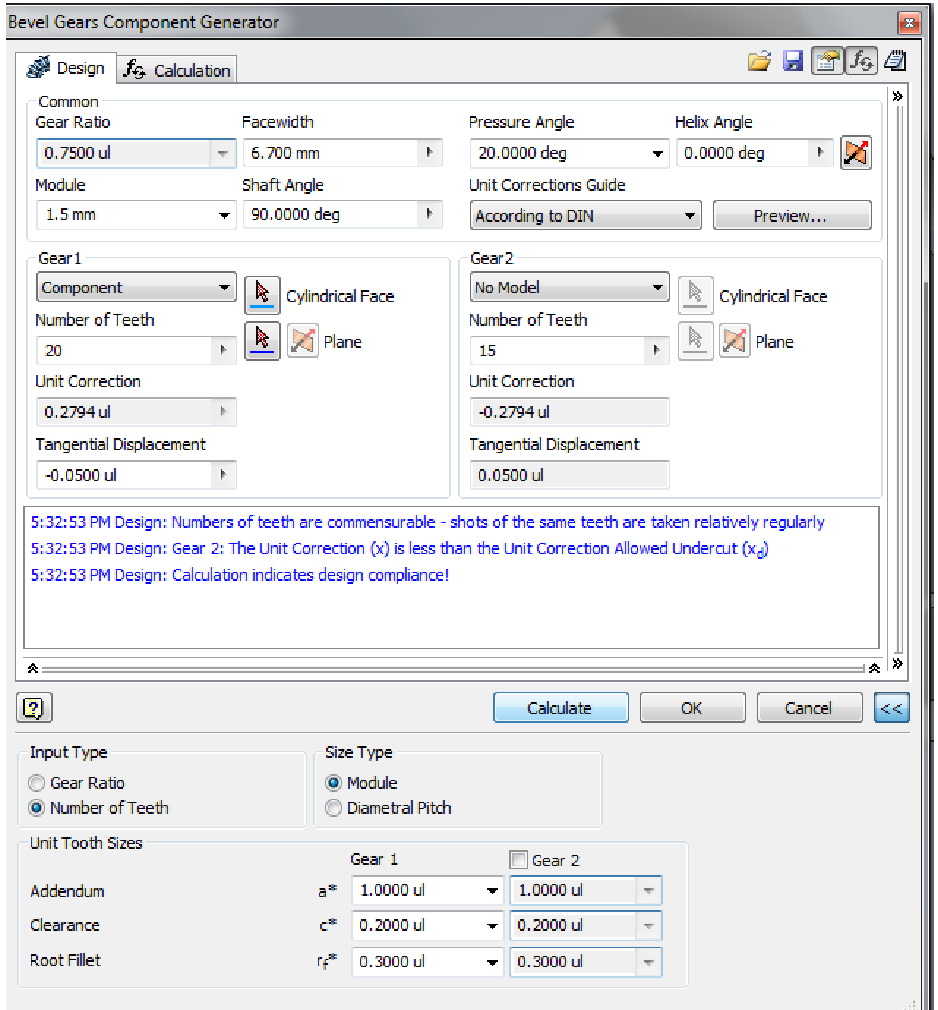
Enter the file name and click “Save”



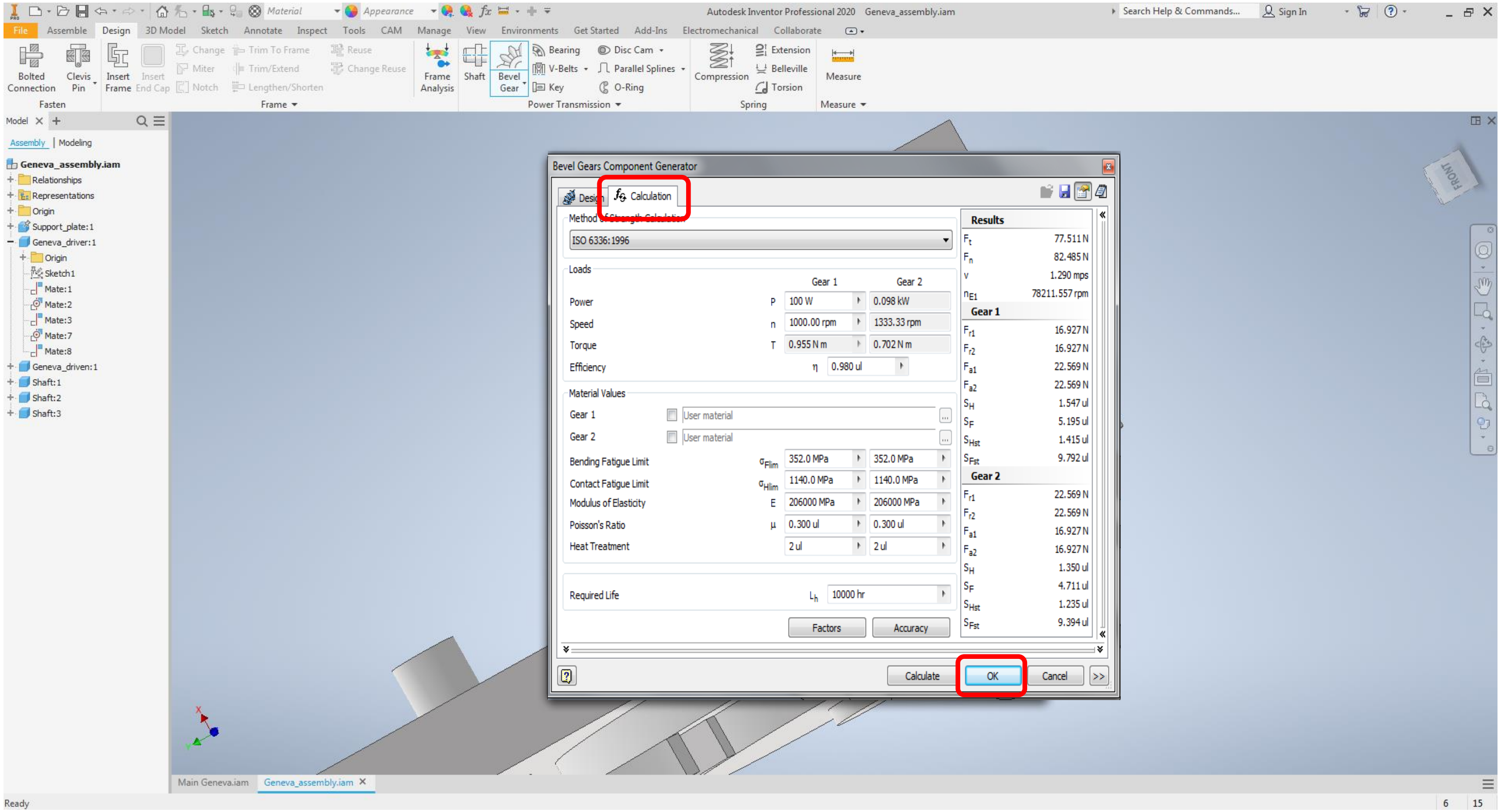
Enter the Module as 1.5mm
Enter Number of teeth as 20 for Gear 1 and 15 for Gear 2
Change Gear 2 to “No model”



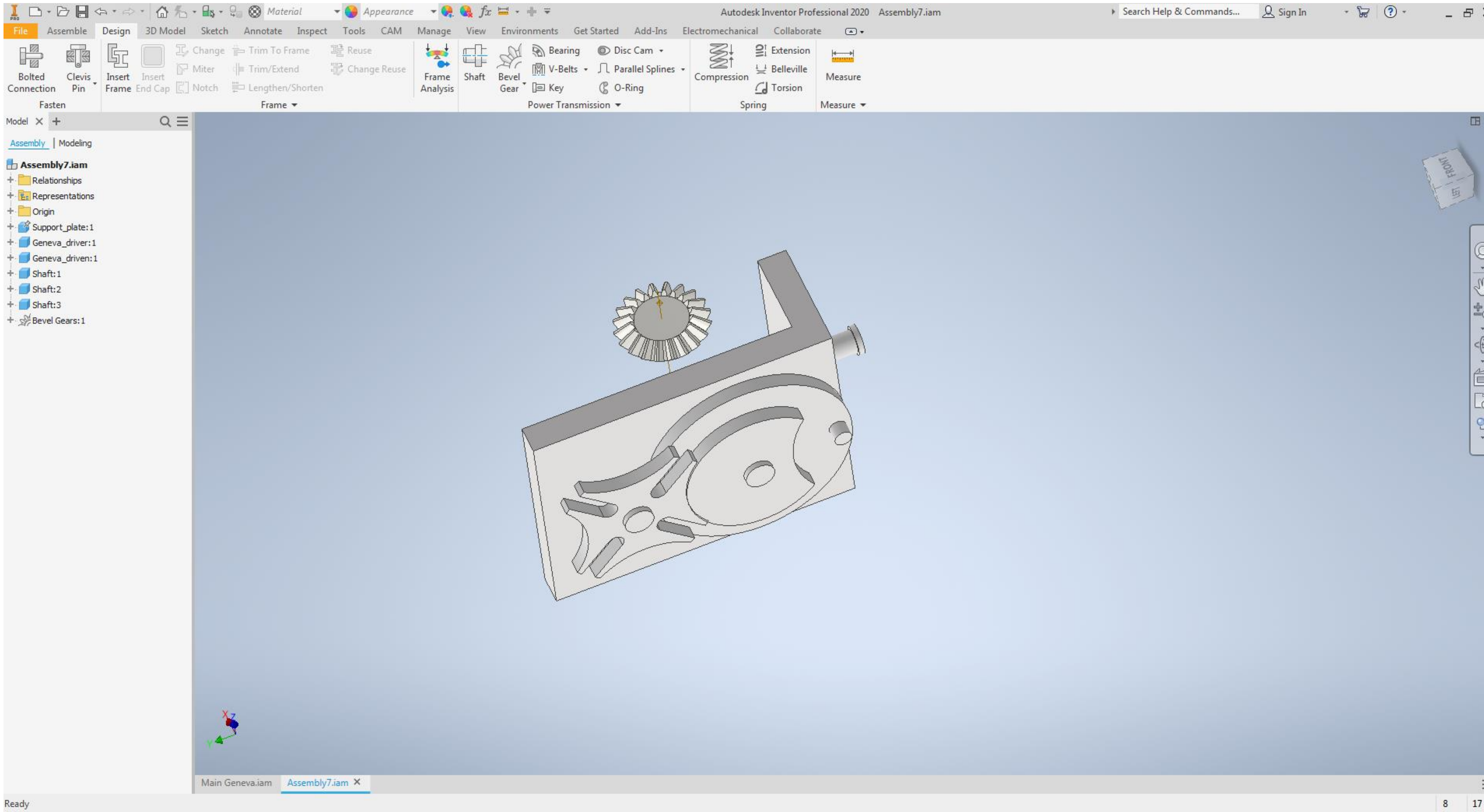
Make sure the other values are as shown below



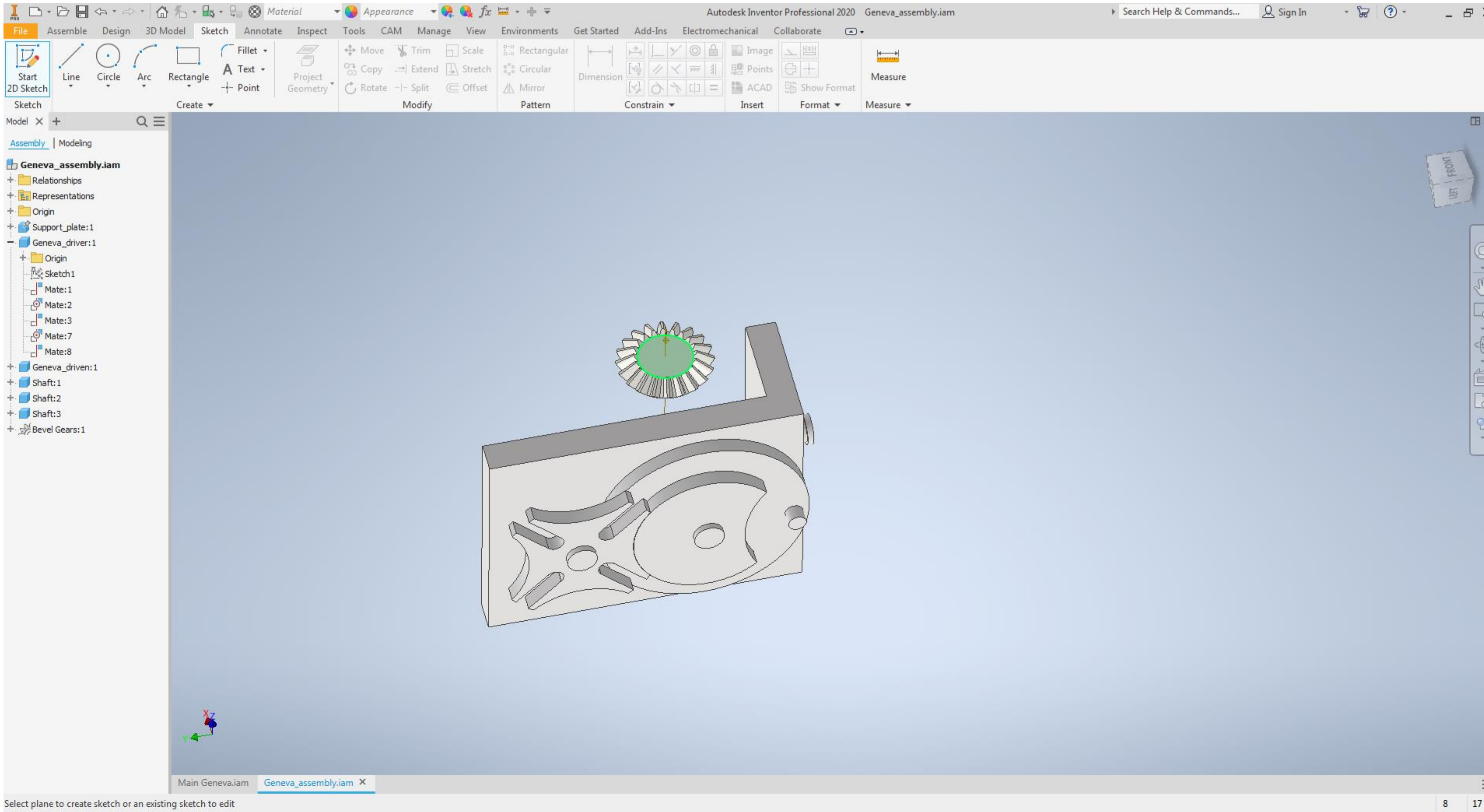
Click on “Calculation” and enter the values of Power as 100W and Speed as 1000rpm
Click “OK” and place the gear at required location by left clicking your mouse button



At this stage, your assembly should look like this

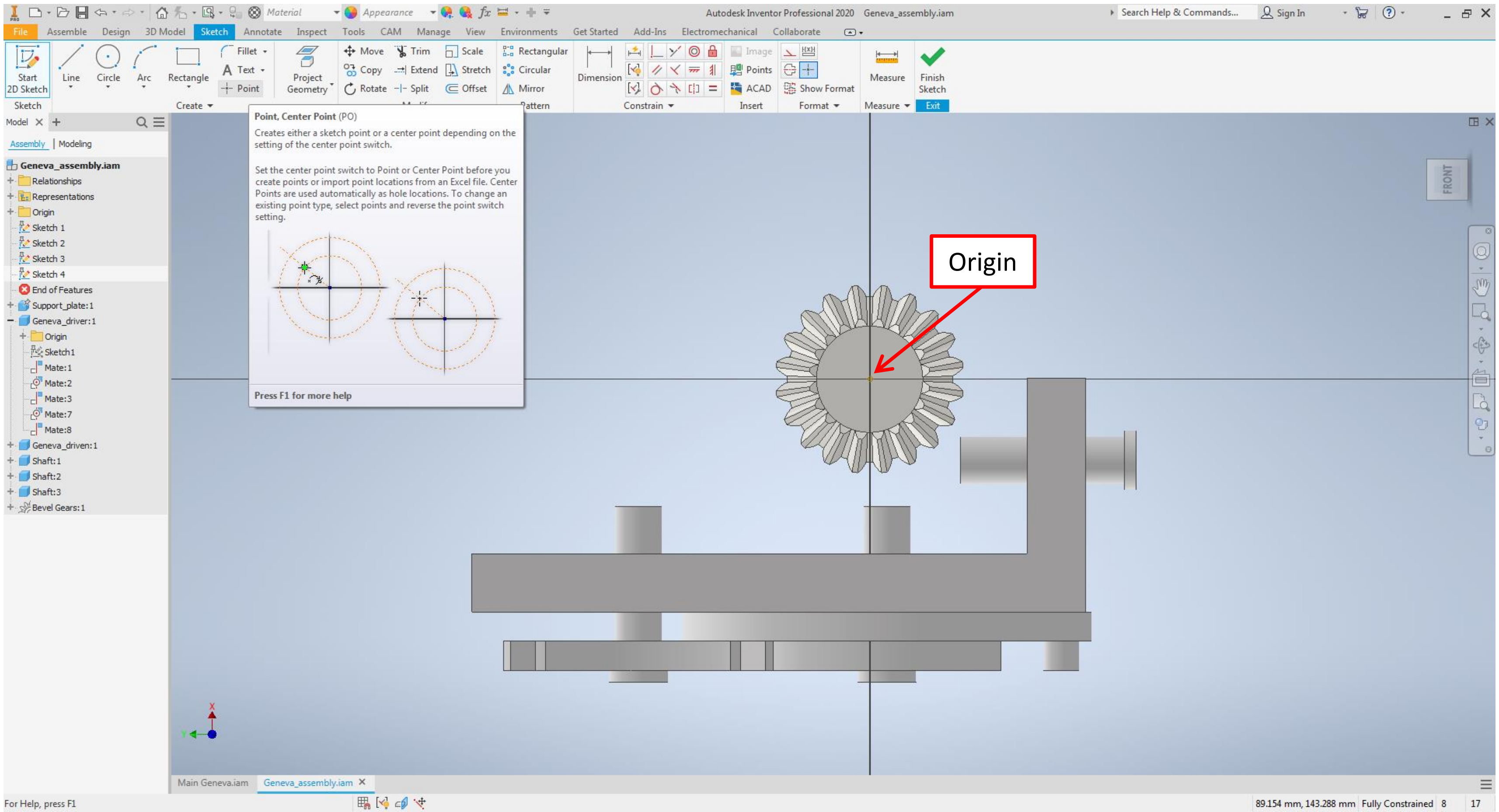


Click on “Start 2D Sketch” and select the top face of the Bevel gear(highlighted below)

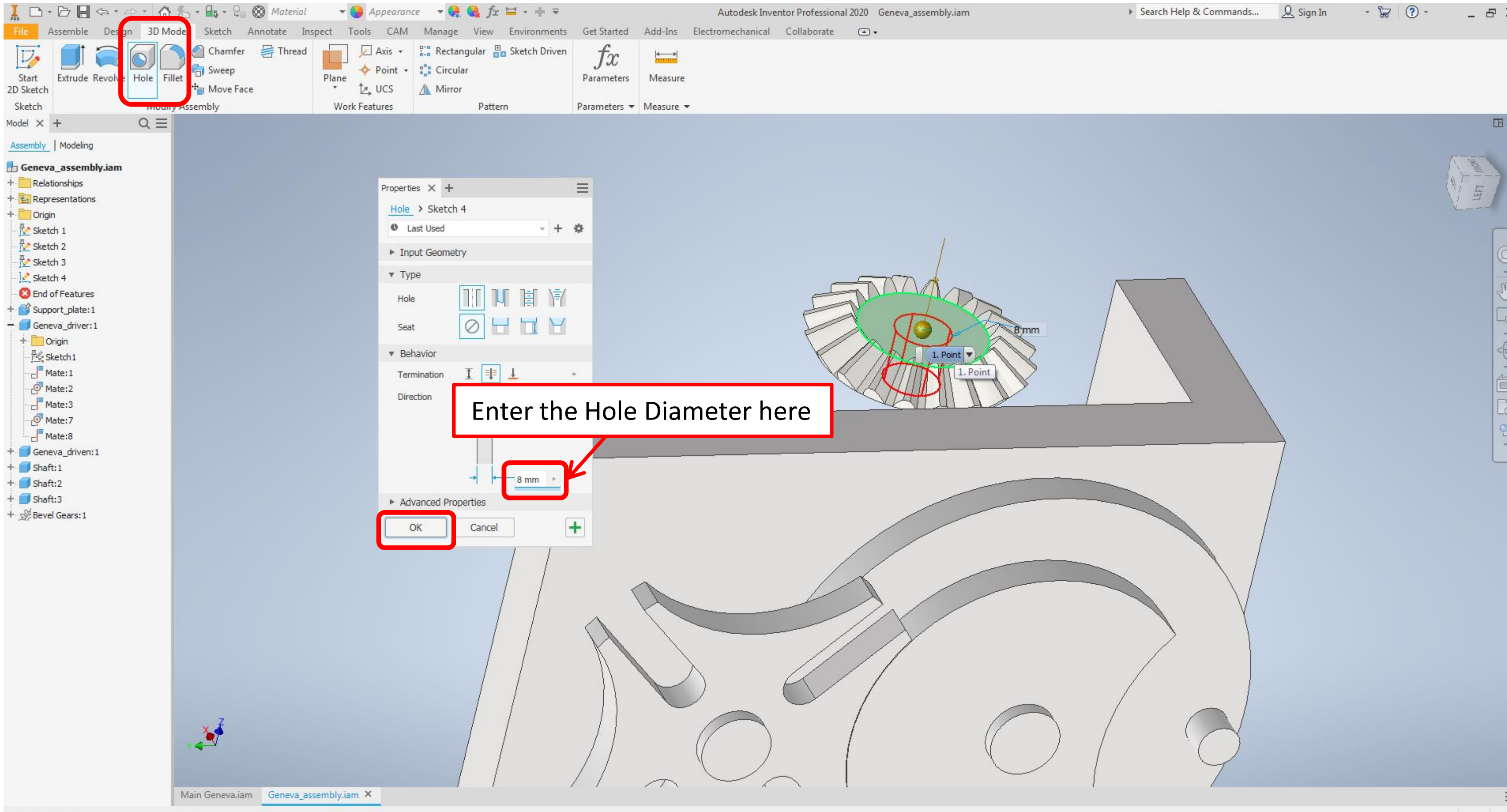


Click on “Point” and click on the origin as shown below to draw a center point on the face of bevel gear

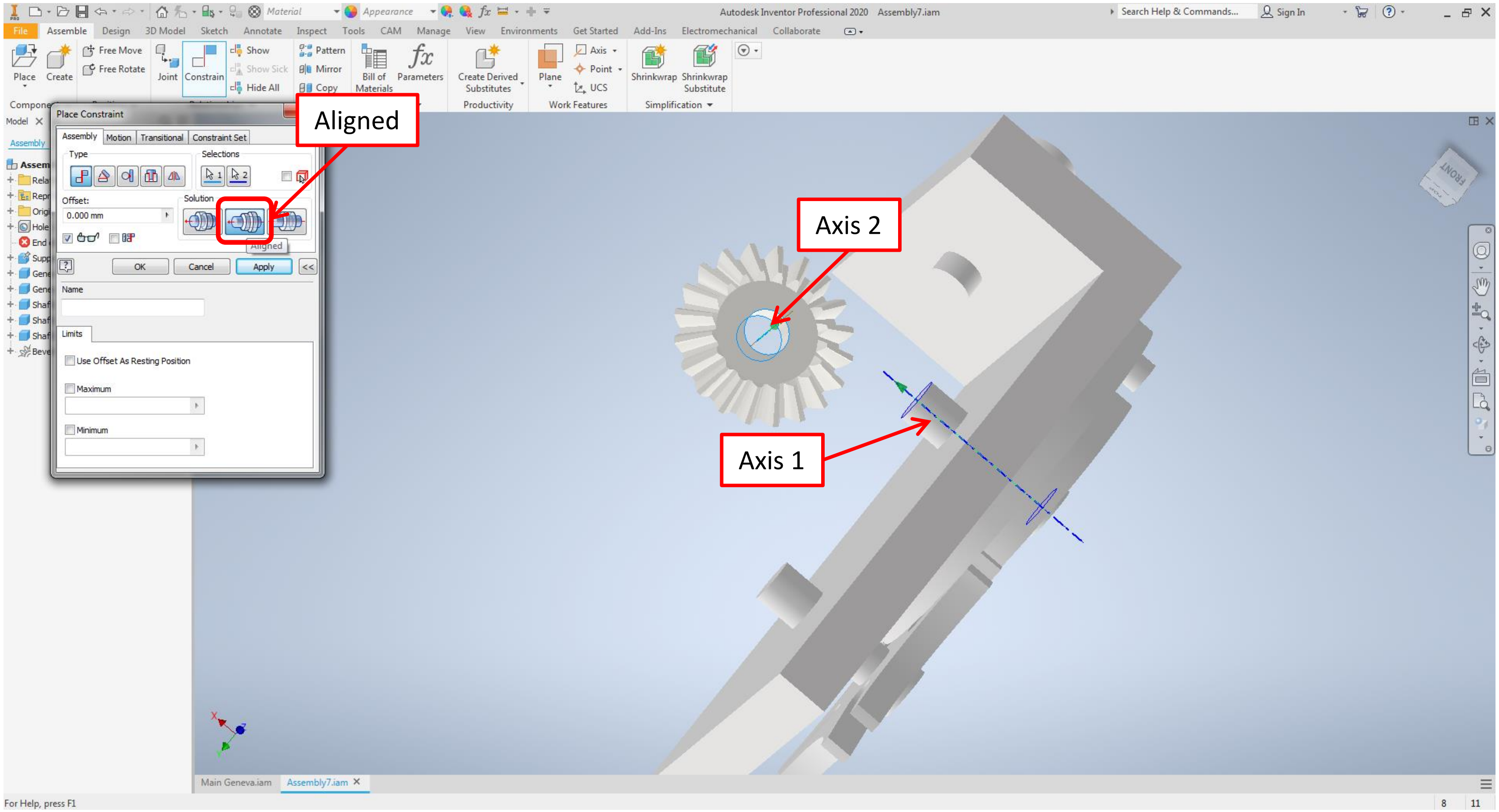
Click “Finish Sketch”



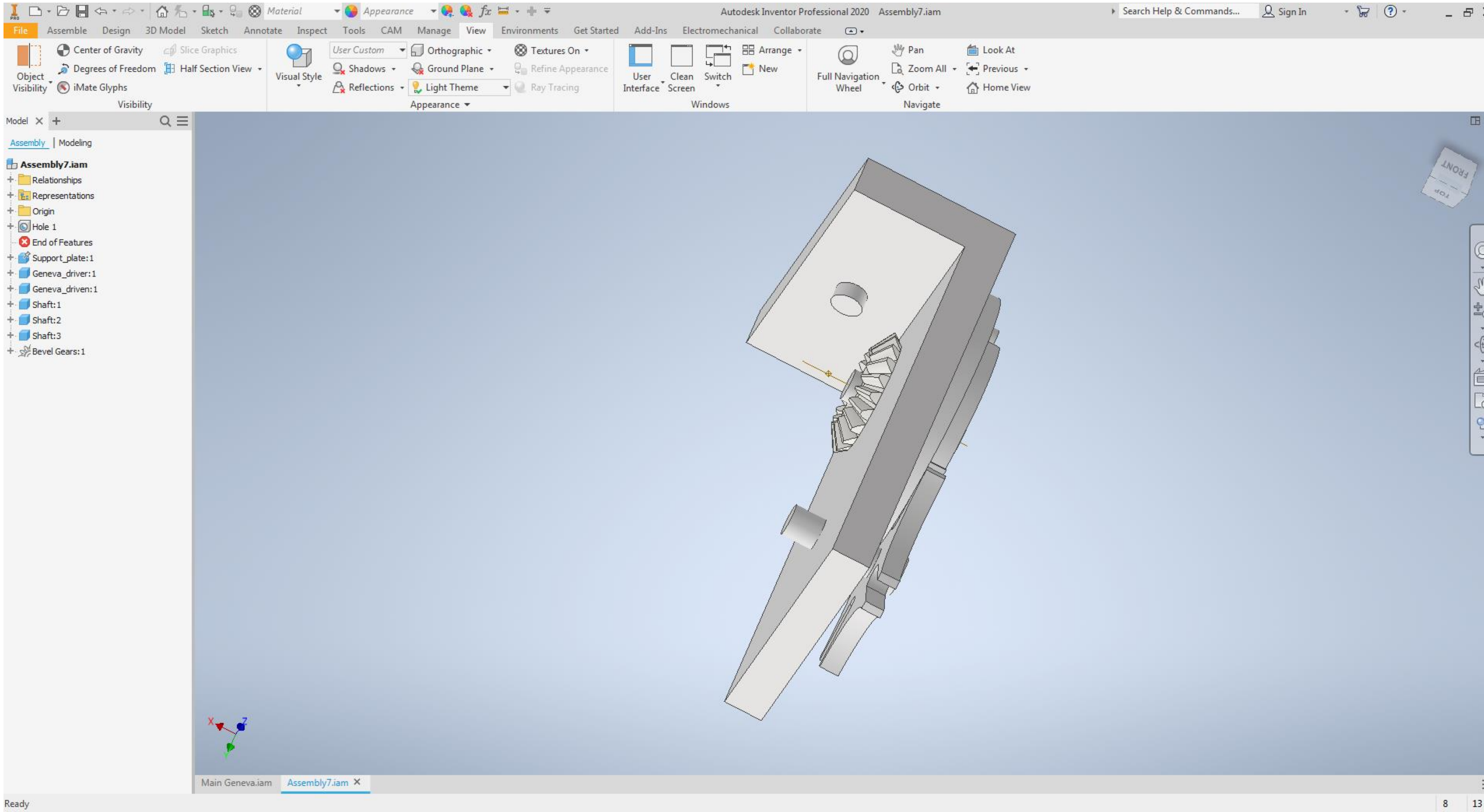
Under “3D Model” tab, click on “Hole” and enter the hole diameter as 8mm
Click “OK”



Click on “Constrain” and select the axis of Shaft 2 and hole axis of the Bevel Gear as shown
Click on “Aligned” to get the orientation right
Click “OK”



At this stage, your assembly should look like this



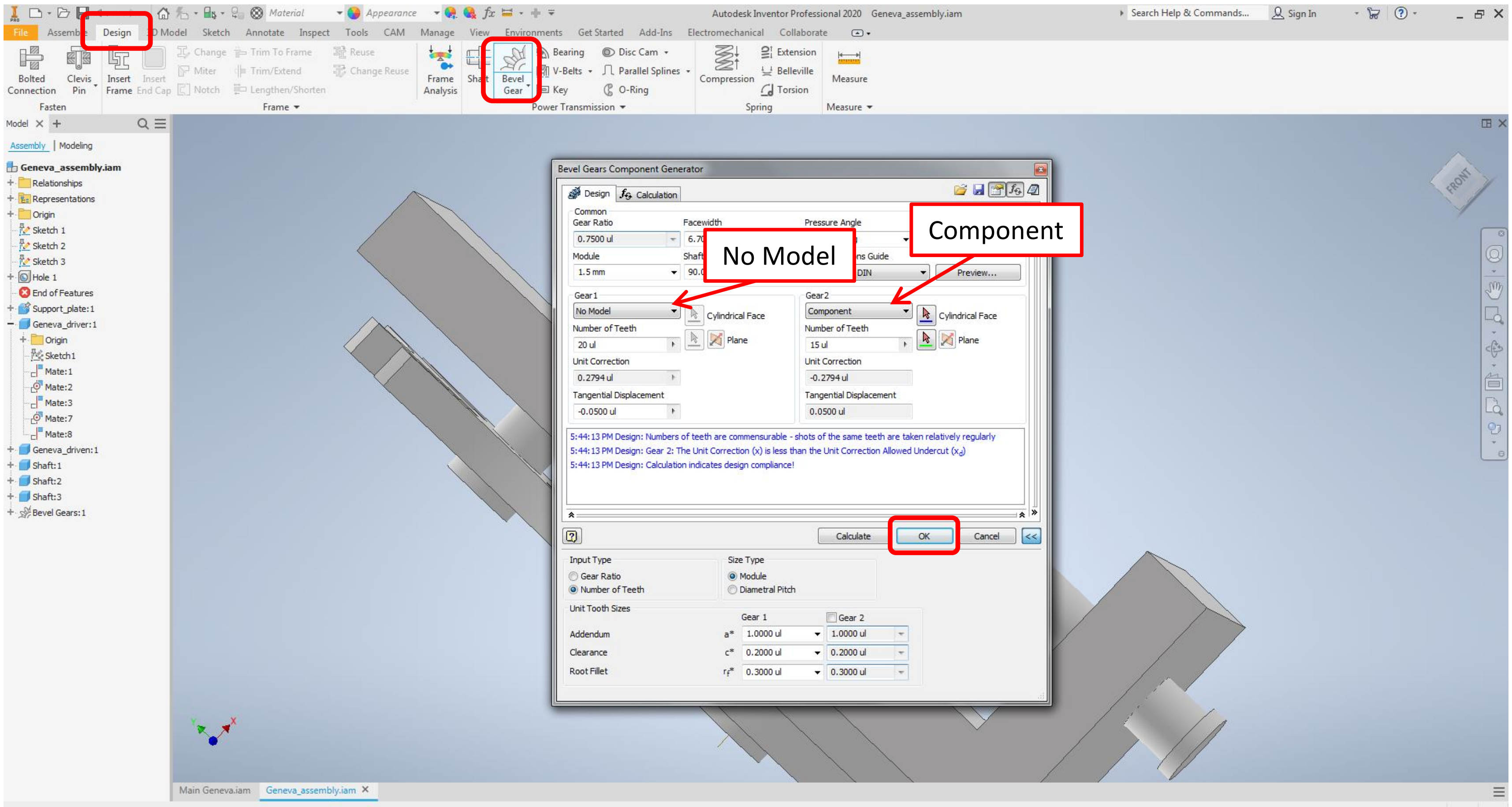
Click on “Design” and select “Bevel Gear”

Enter Module as 1.5mm

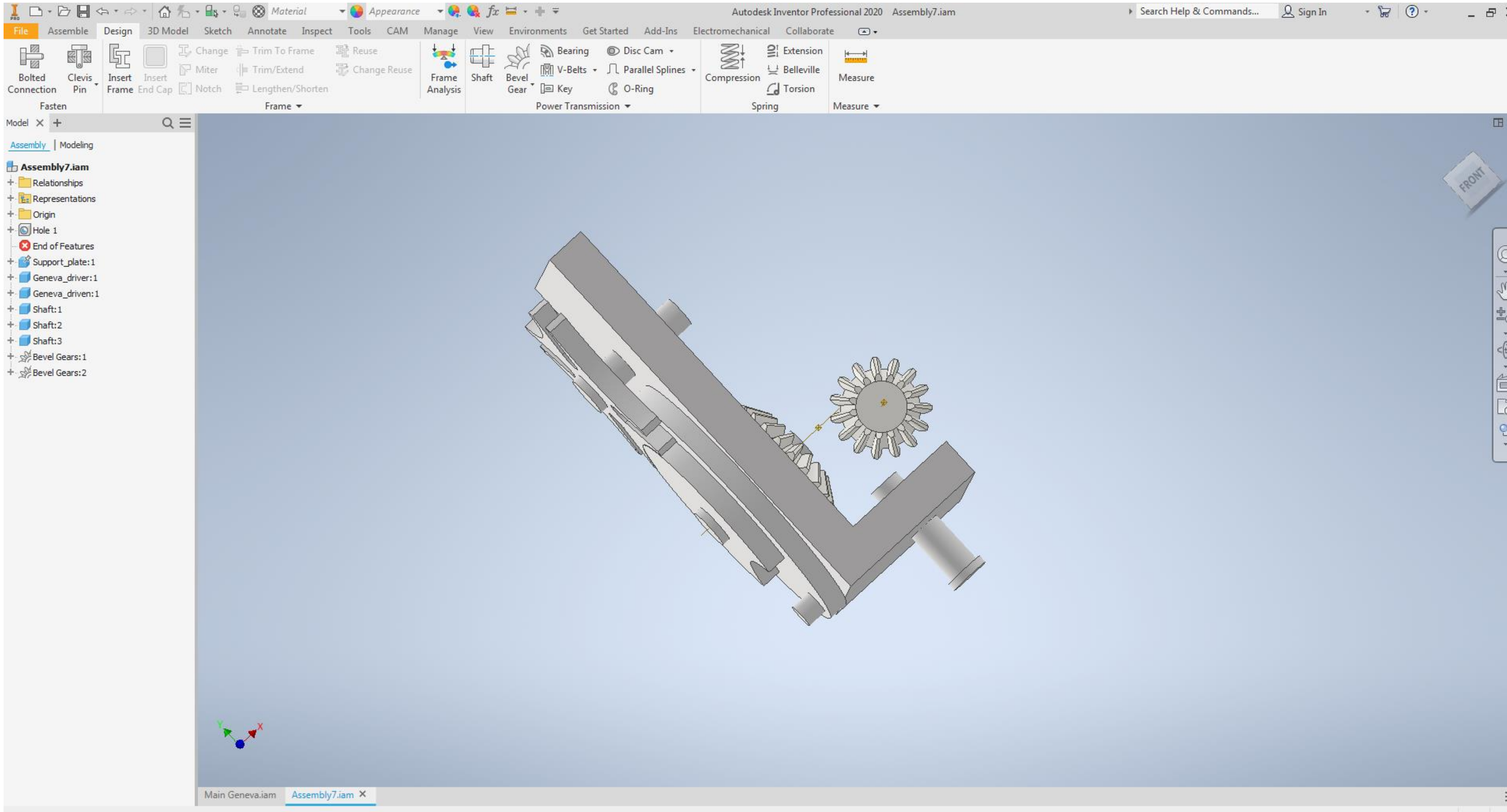
Enter Number of teeth in Gear 1 as 20 and Gear 2 as 15

Change Gear 1 to “No Model” and Gear 2 to “Component”

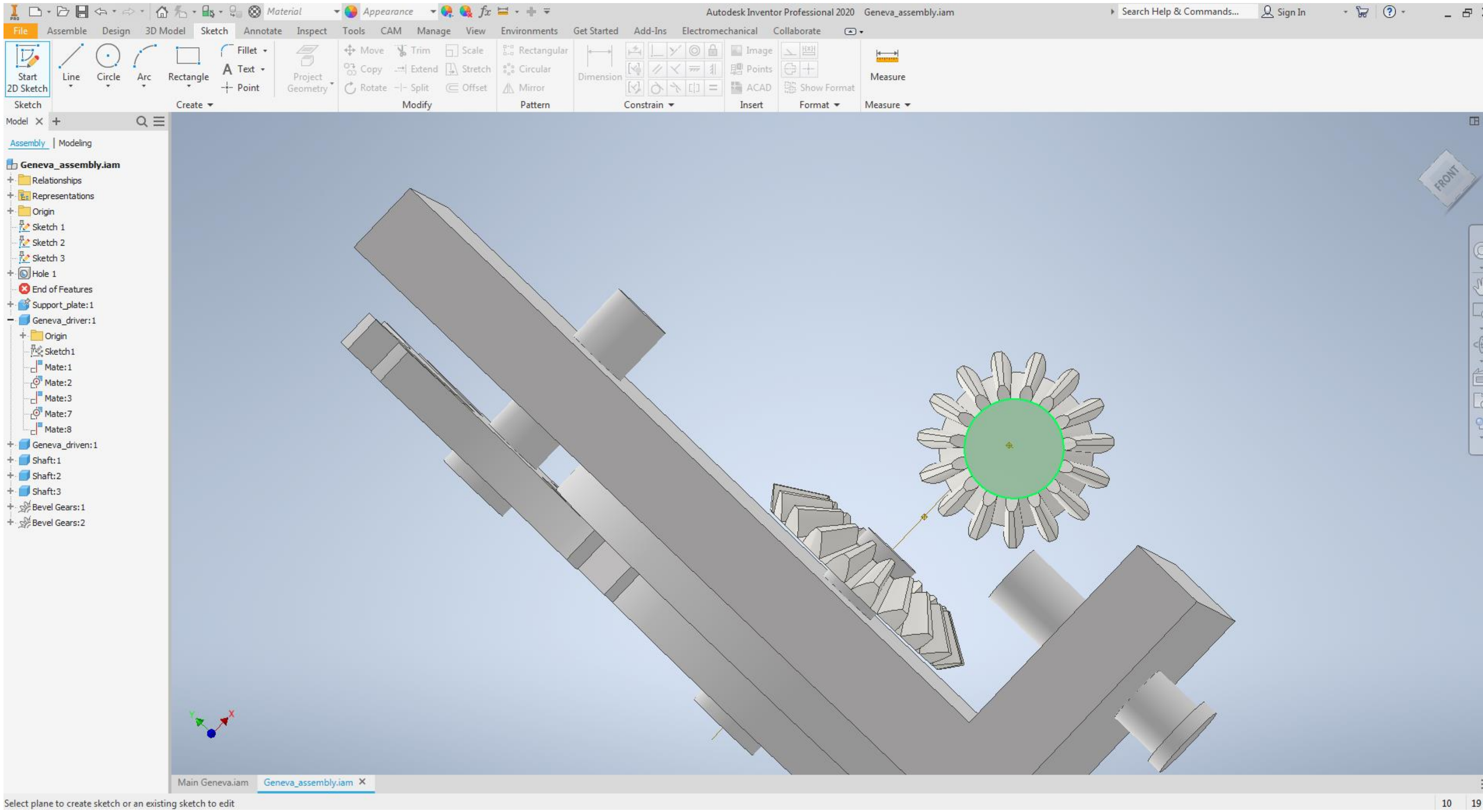
Click “OK” and place the gear at required location by left clicking your mouse button



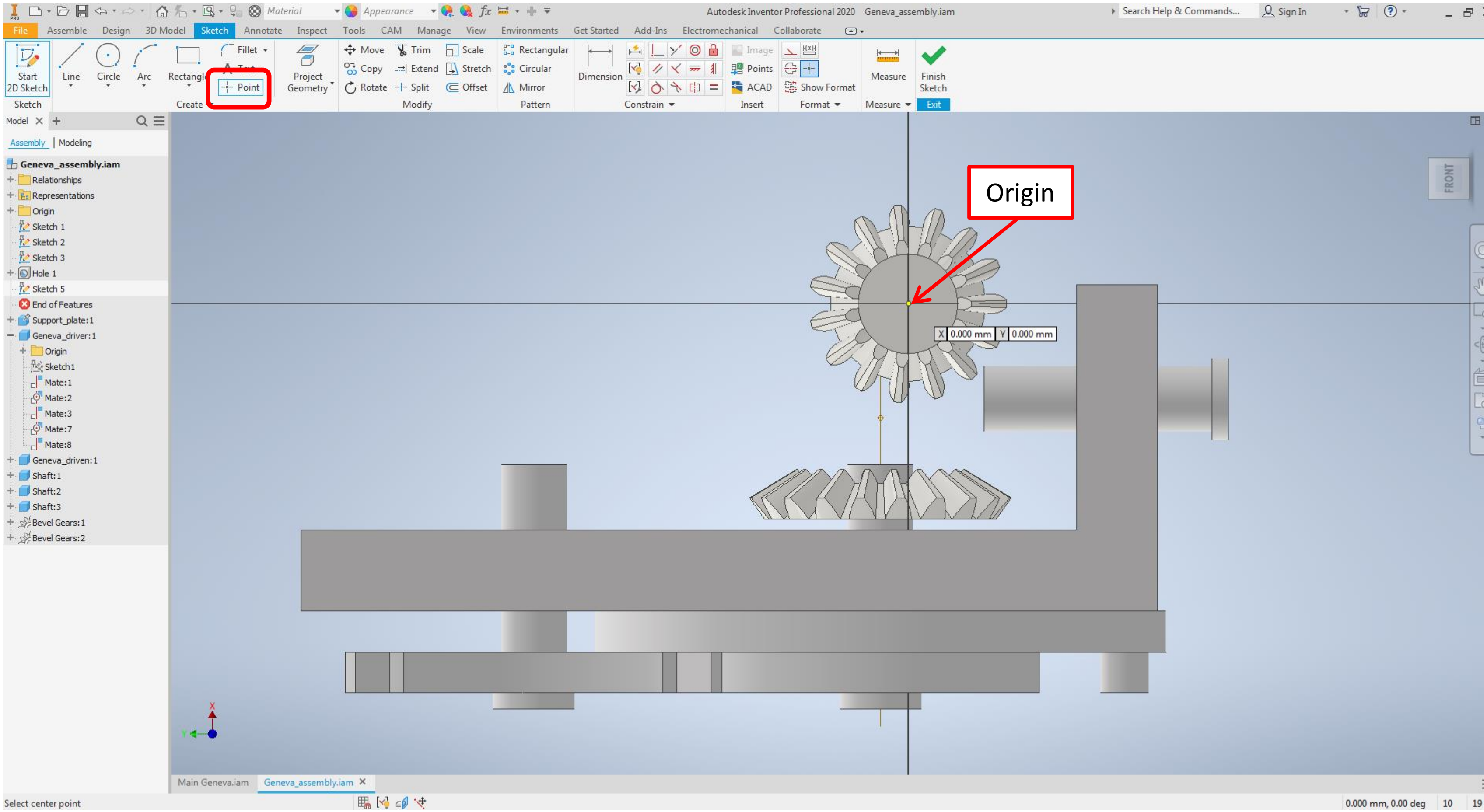
At this stage, your assembly should look like this



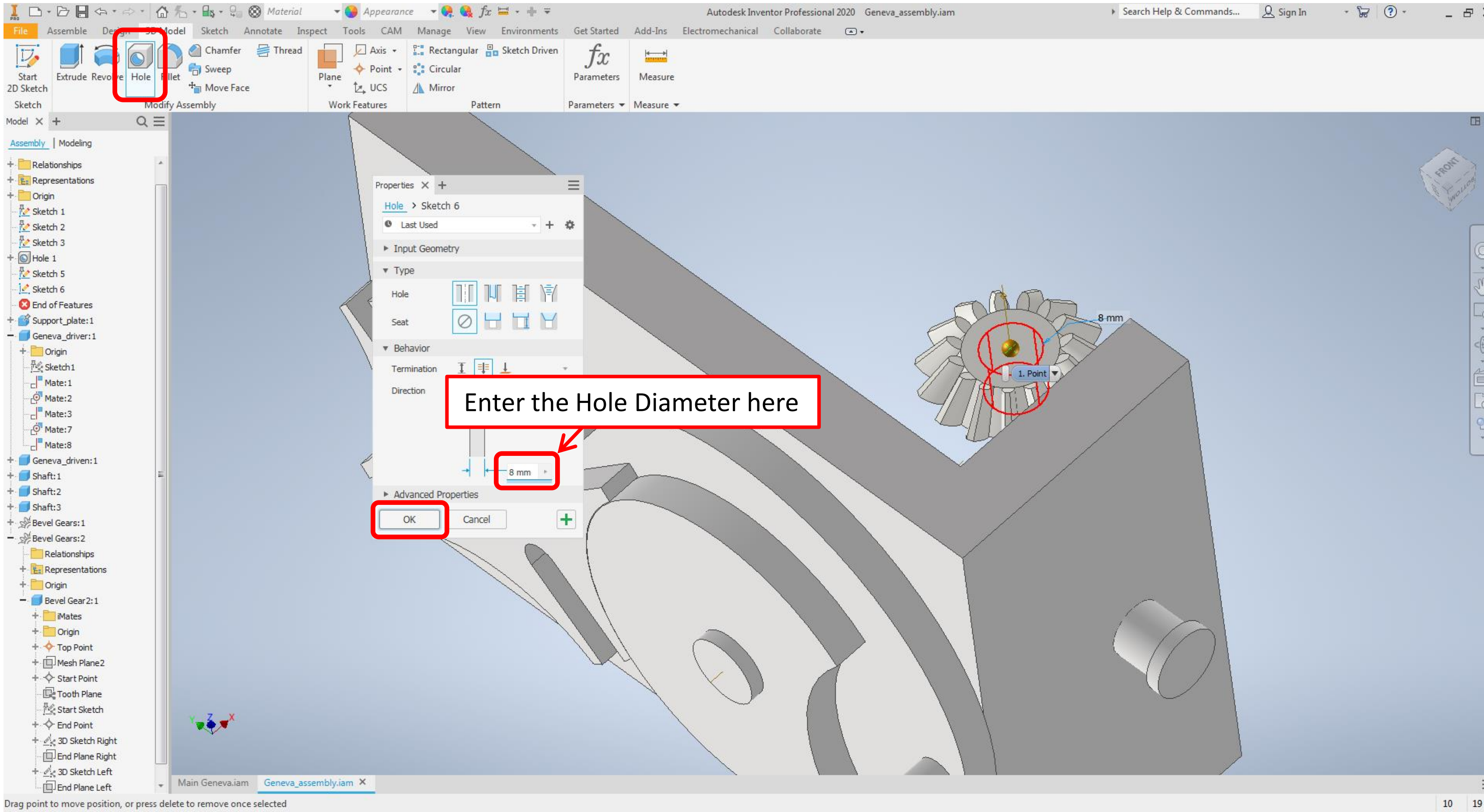
Click on “Start 2D Sketch” and select the top face of the Bevel Gear 2(highlighted below)



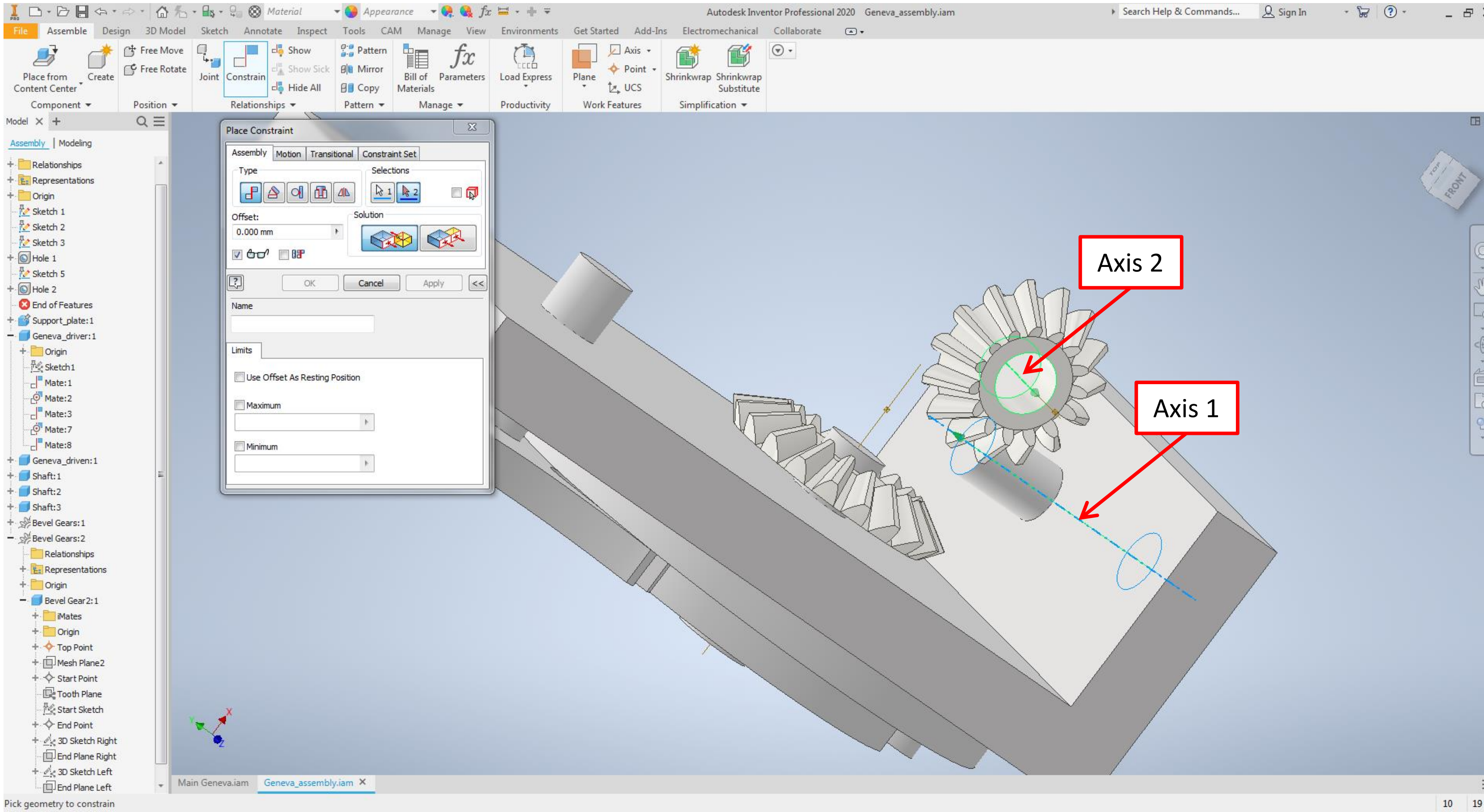
Click on “Point” and create a point on Bevel gear 2 at the origin as shown below
Click “Finish Sketch”



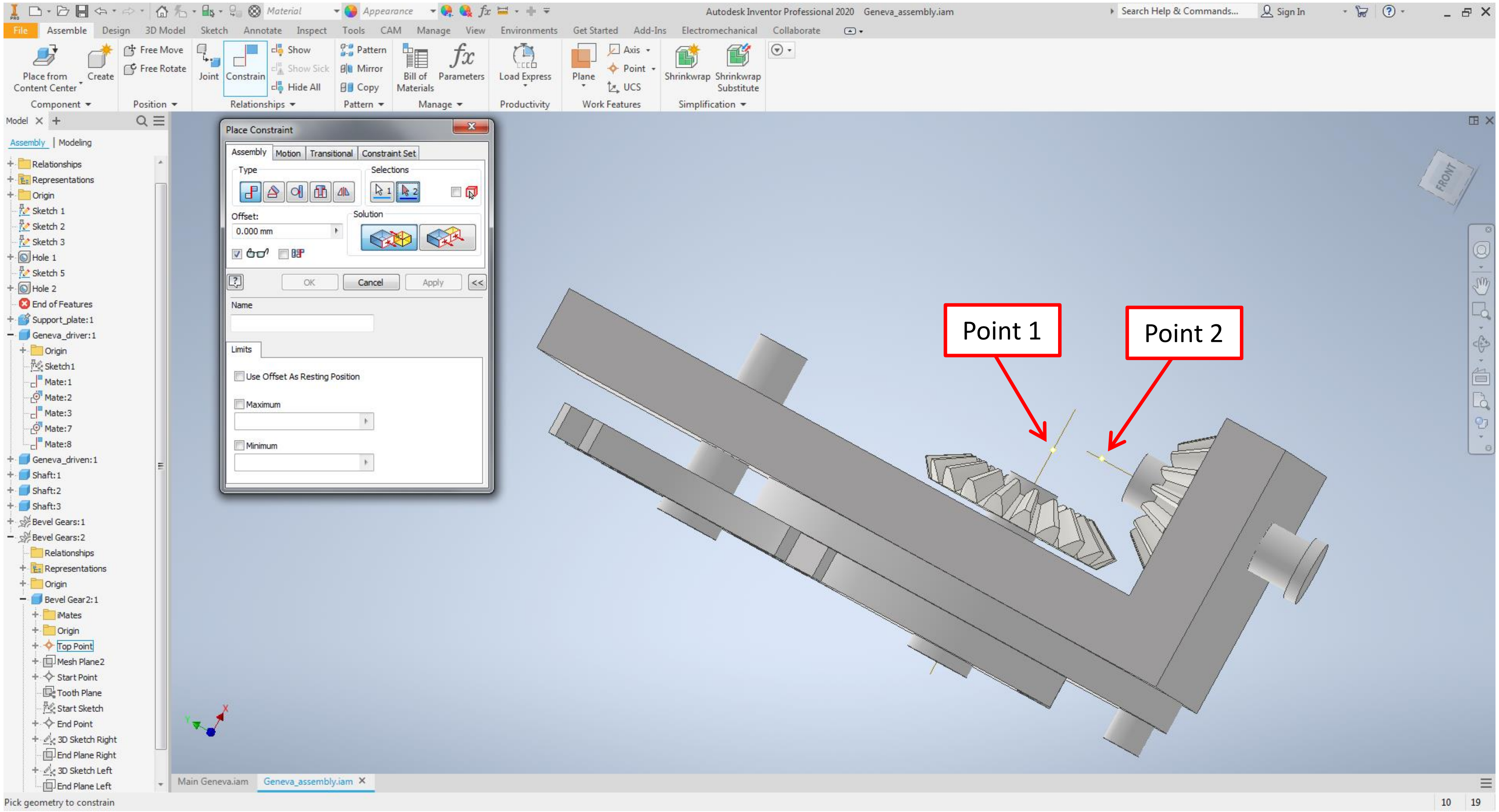
Under the “3D Model” tab. Click on “Hole” and enter the hole diameter as 8mm
Click “OK”



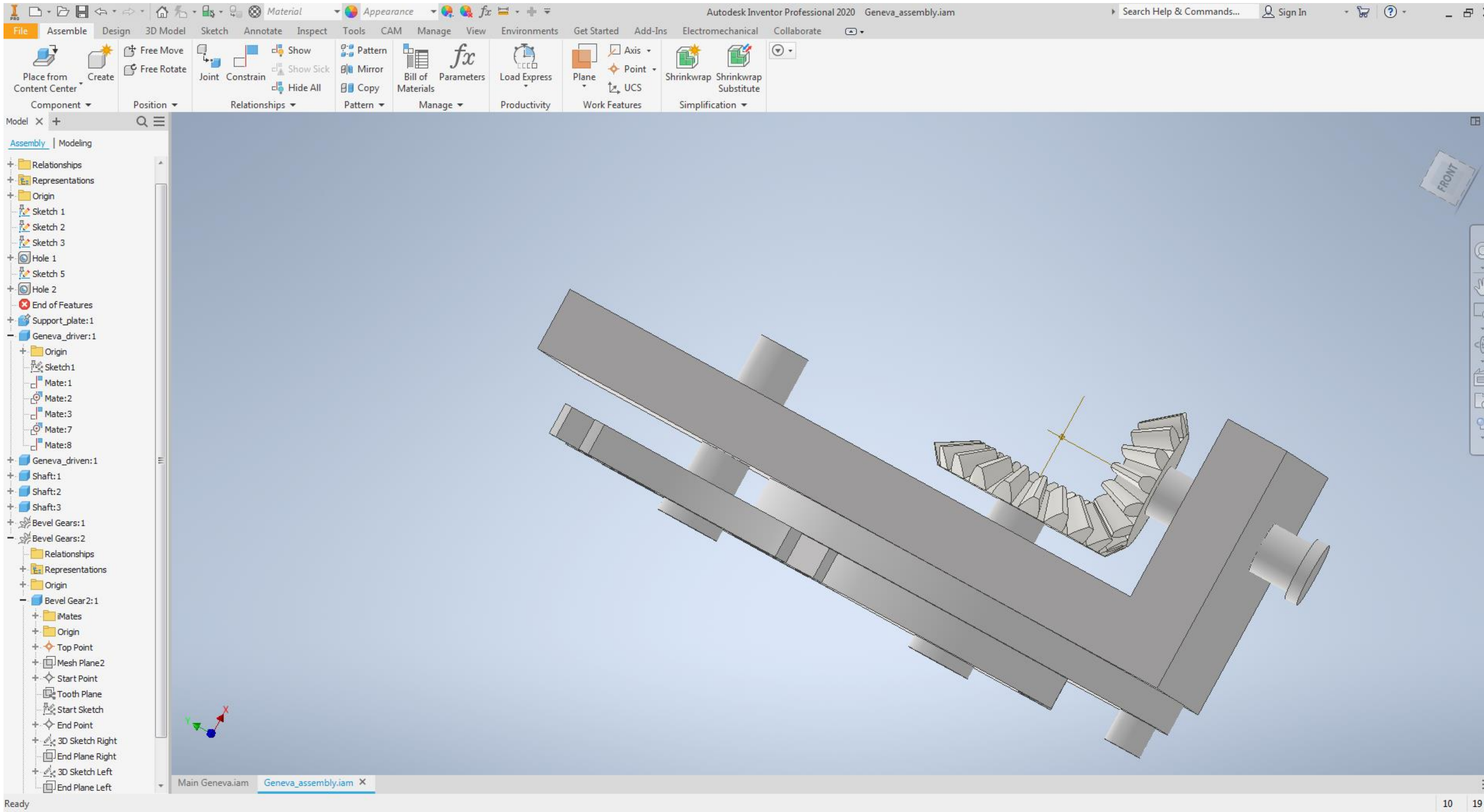
Click on “Constrain” and select the axis of Shaft 3 and Bevel Gear 2 as shown below
Click “OK”



Click on “Constrain” and select the two points of the bevel gears as shown below
Click “Ok”

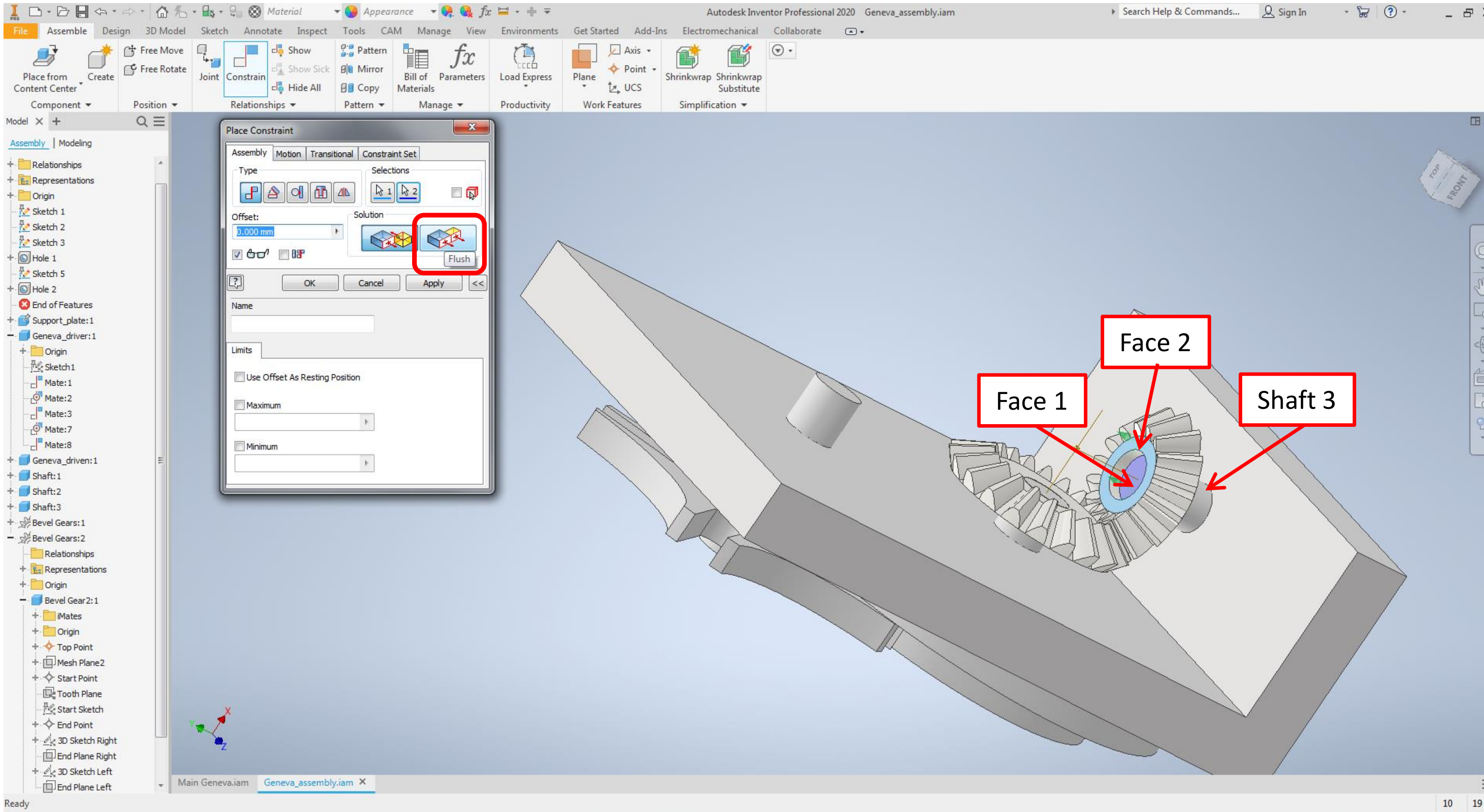


At this stage, your Assembly should look like this



Click on “Constrain” and select the face of Shaft 3 and the top face of Bevel Gear 2 and as shown below

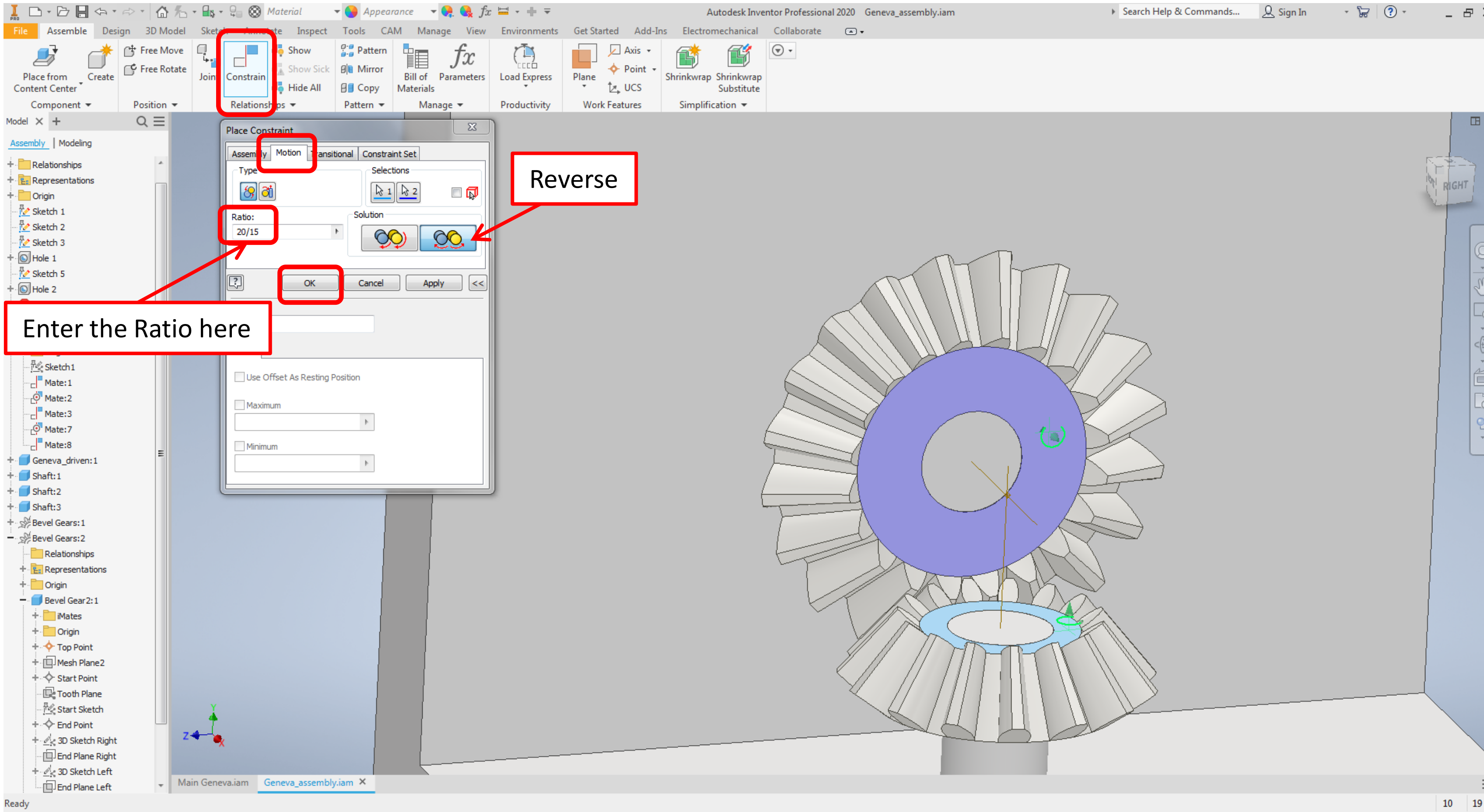
Click on “Flush” and click “Ok”



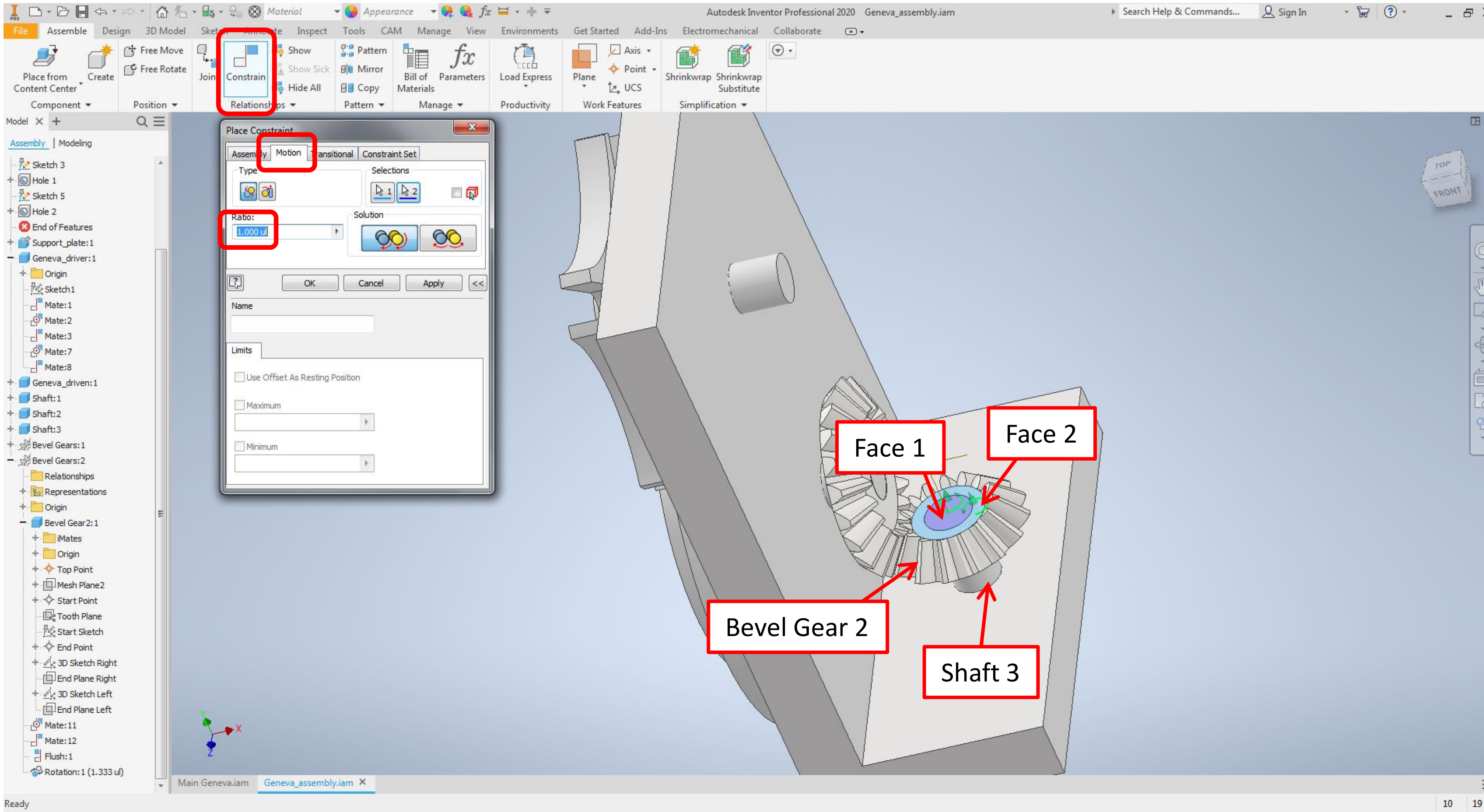
Click on “Constrain” and select “Motion

Enter the Ratio as 20/15 and click on “Reverse”

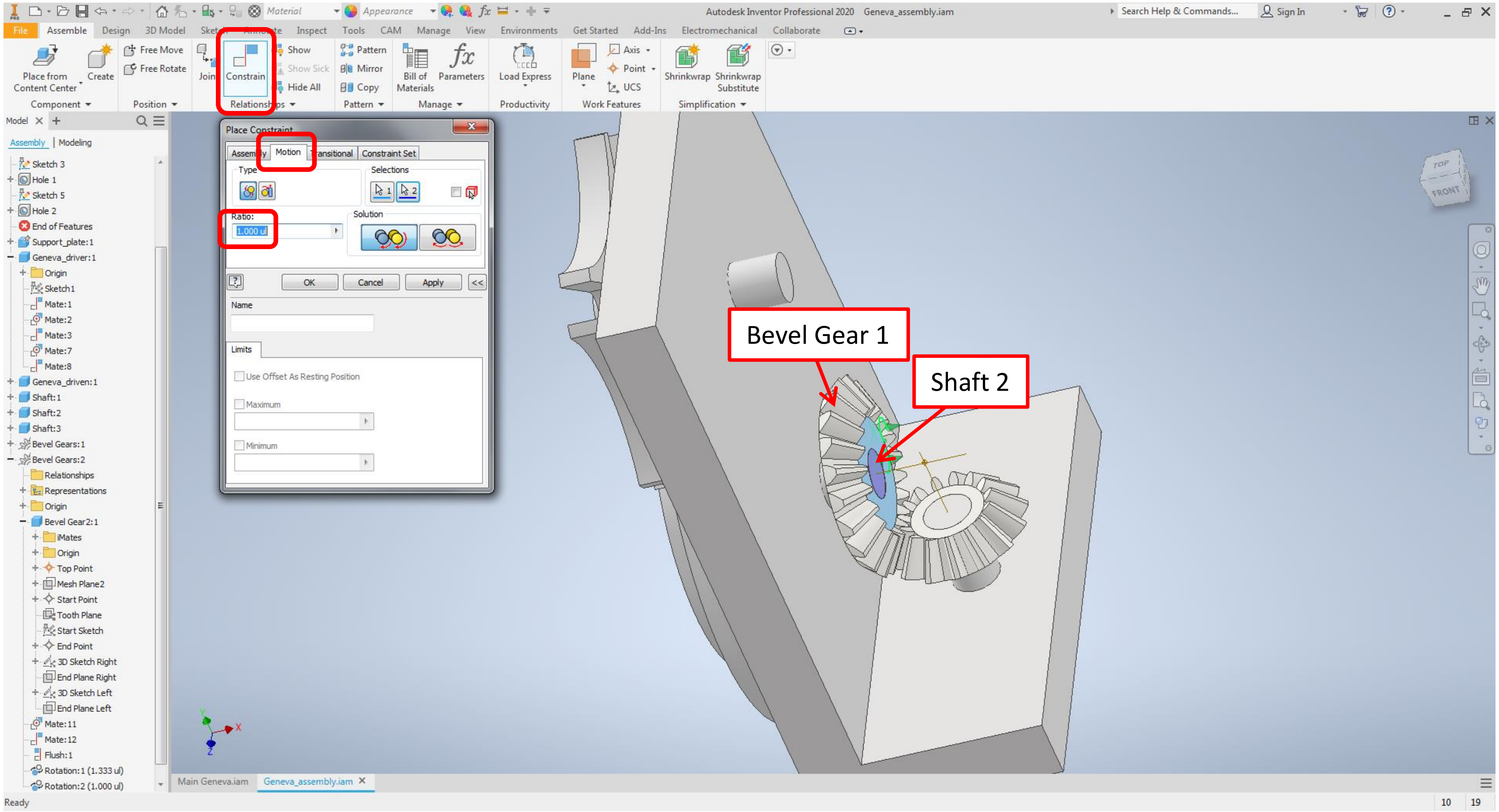
Select the faces of two Bevel Gears as shown below and Click “OK”



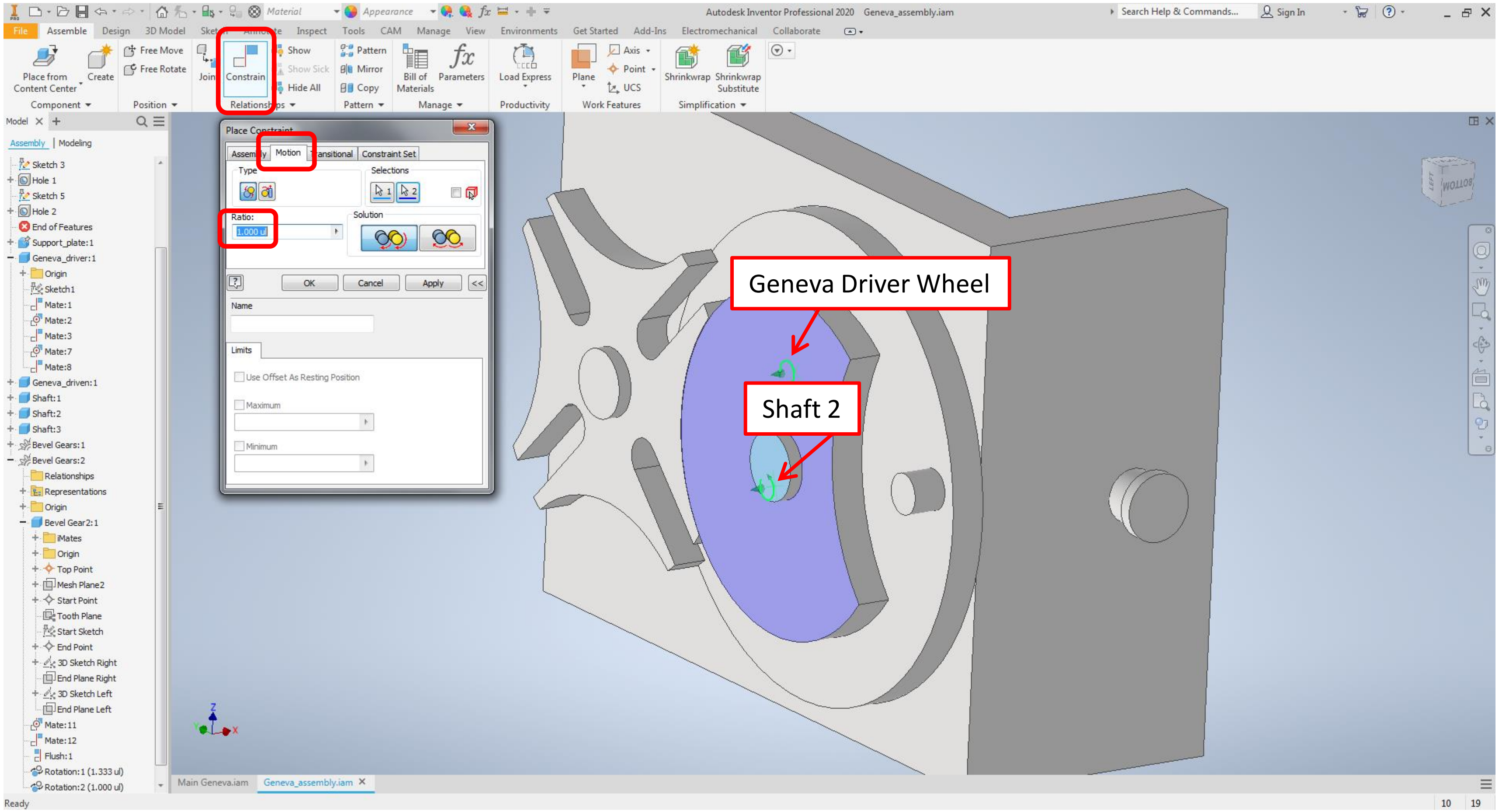
Click on “Constrain” and select “Motion”
Enter the Ratio as 1
Select the faces of Shaft 3 and Bevel Gear 2 as shown below and click “OK”



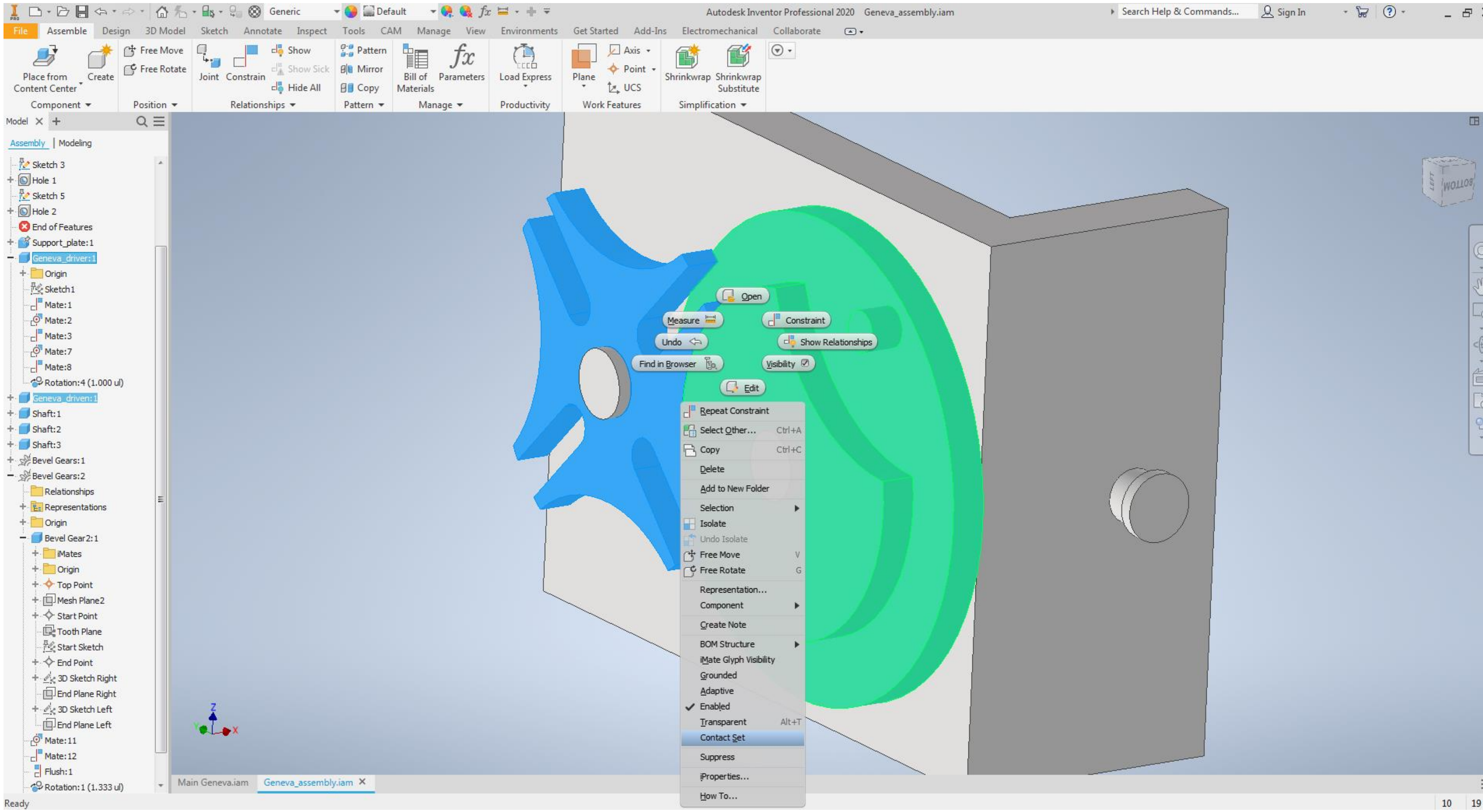
Follow the exact same steps as given in the previous slide and give motion constrain to Shaft 2 and Bevel Gear 1 keeping the Ratio as 1



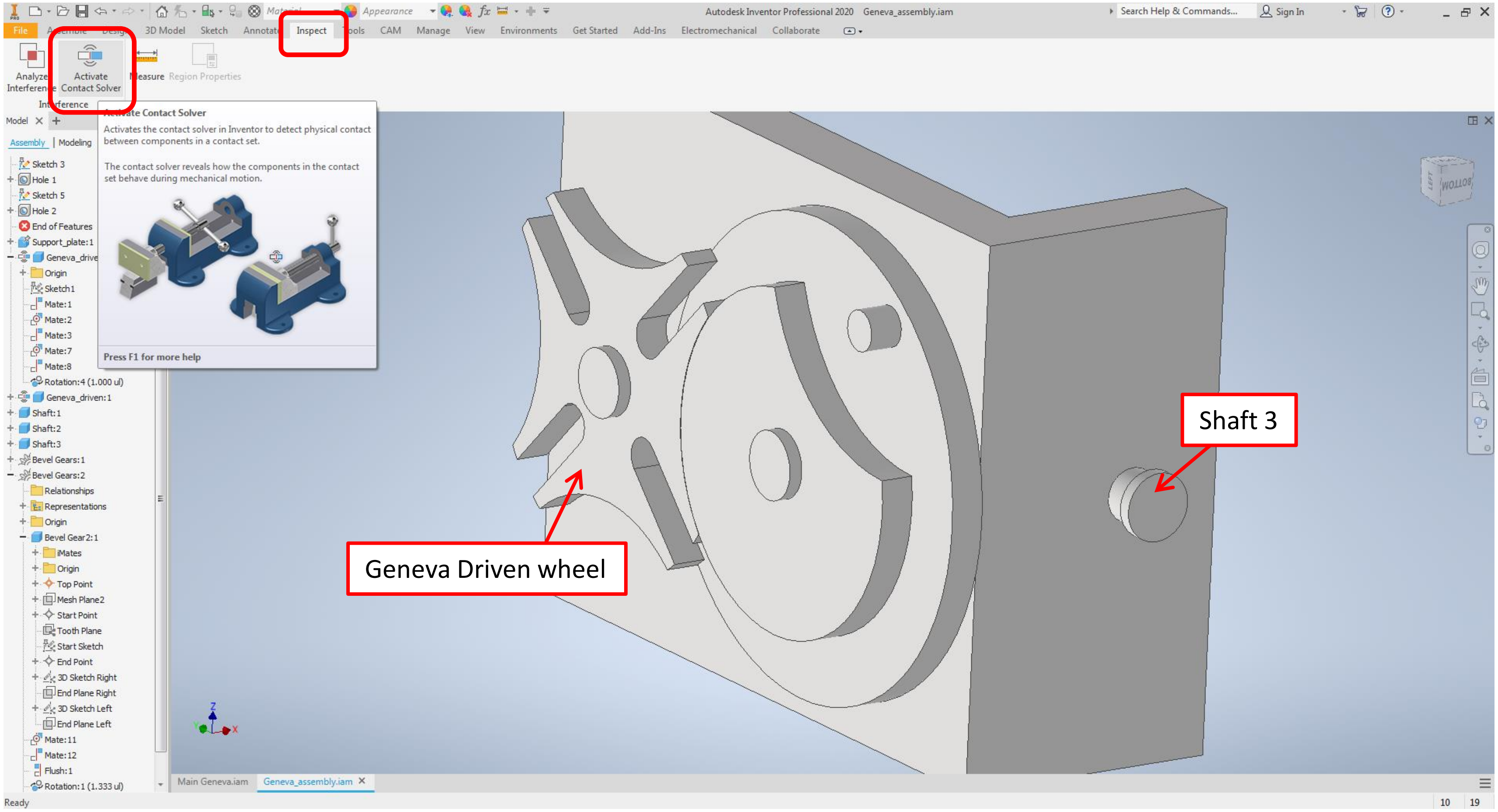
Give motion constrain to Shaft 2 and Geneva Driver Wheel keeping the Ratio as 1



Holding the control key, select both Geneva driver and the driven wheel
Right click and select “Contact Set



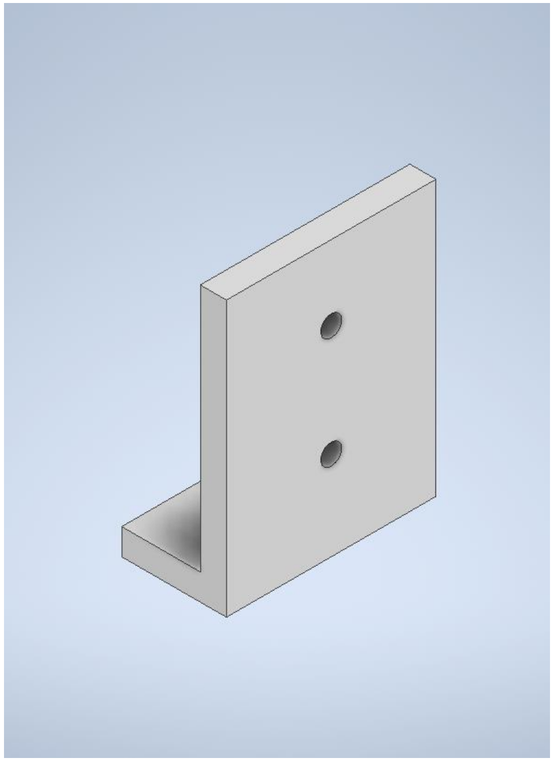
Click on “Inspect” and select “Activate Contact Solver”
Now the Geneva Driven Wheel should move when you rotate Shaft 3
*Finally, save your assembly

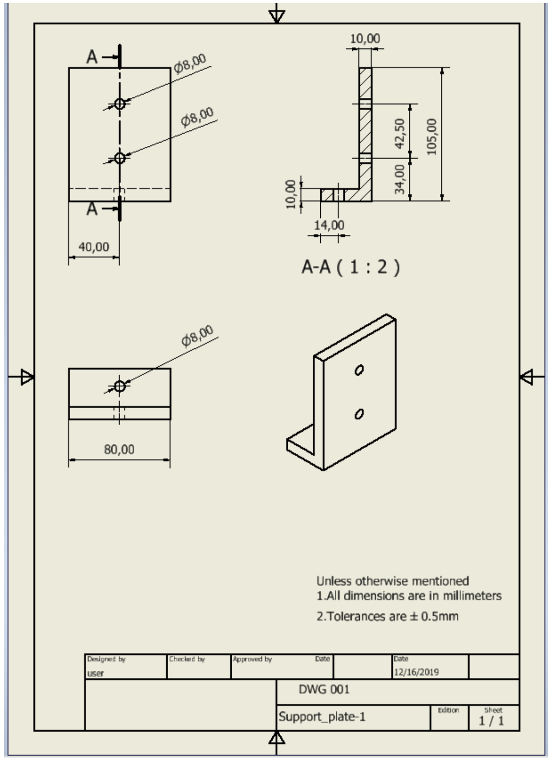


This ends your task 5. Please show your progress to your guide/TA/Tutor. Proceed after.

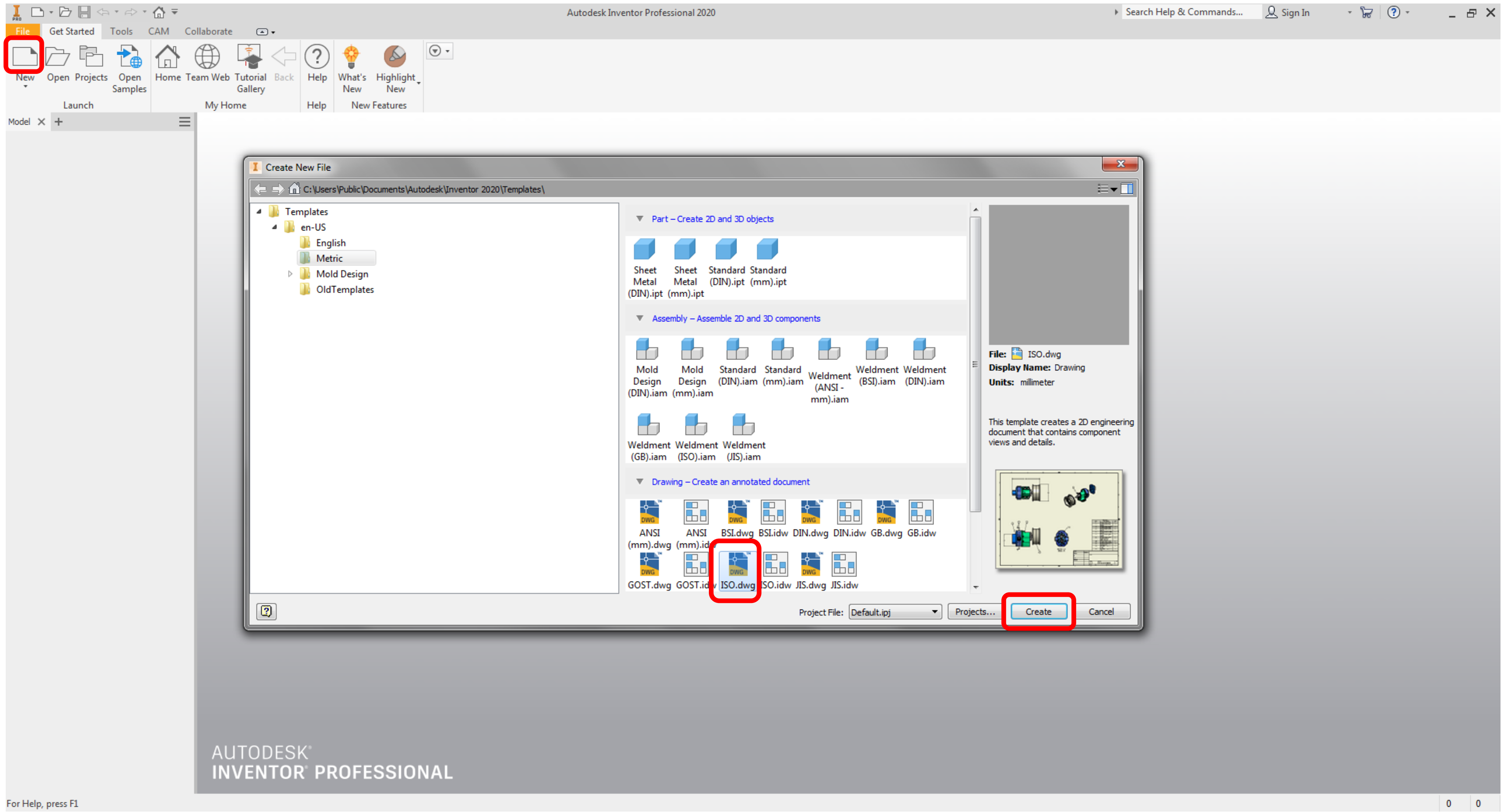
Task 6

To make a 2D manufacturing drawing from a 3D part

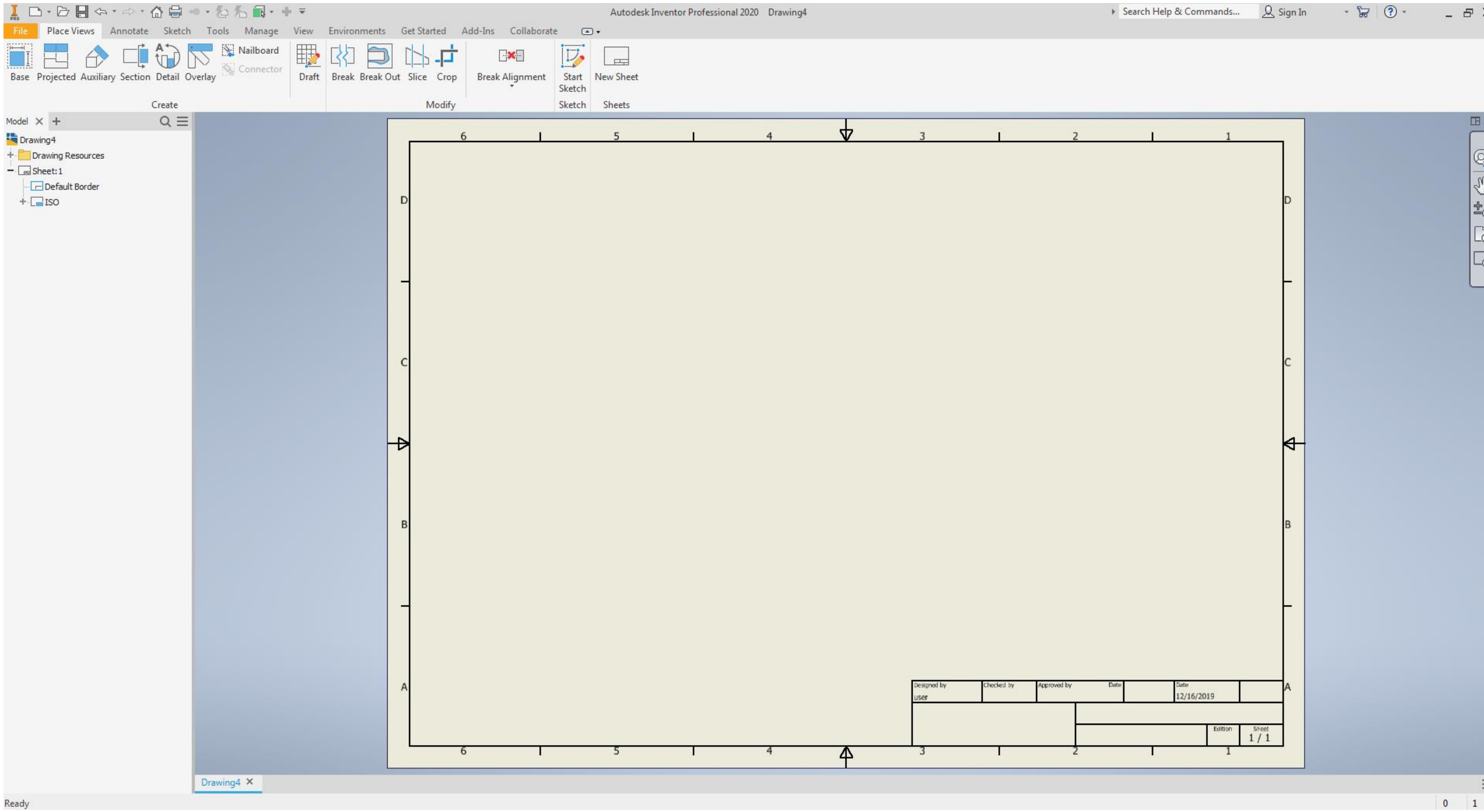




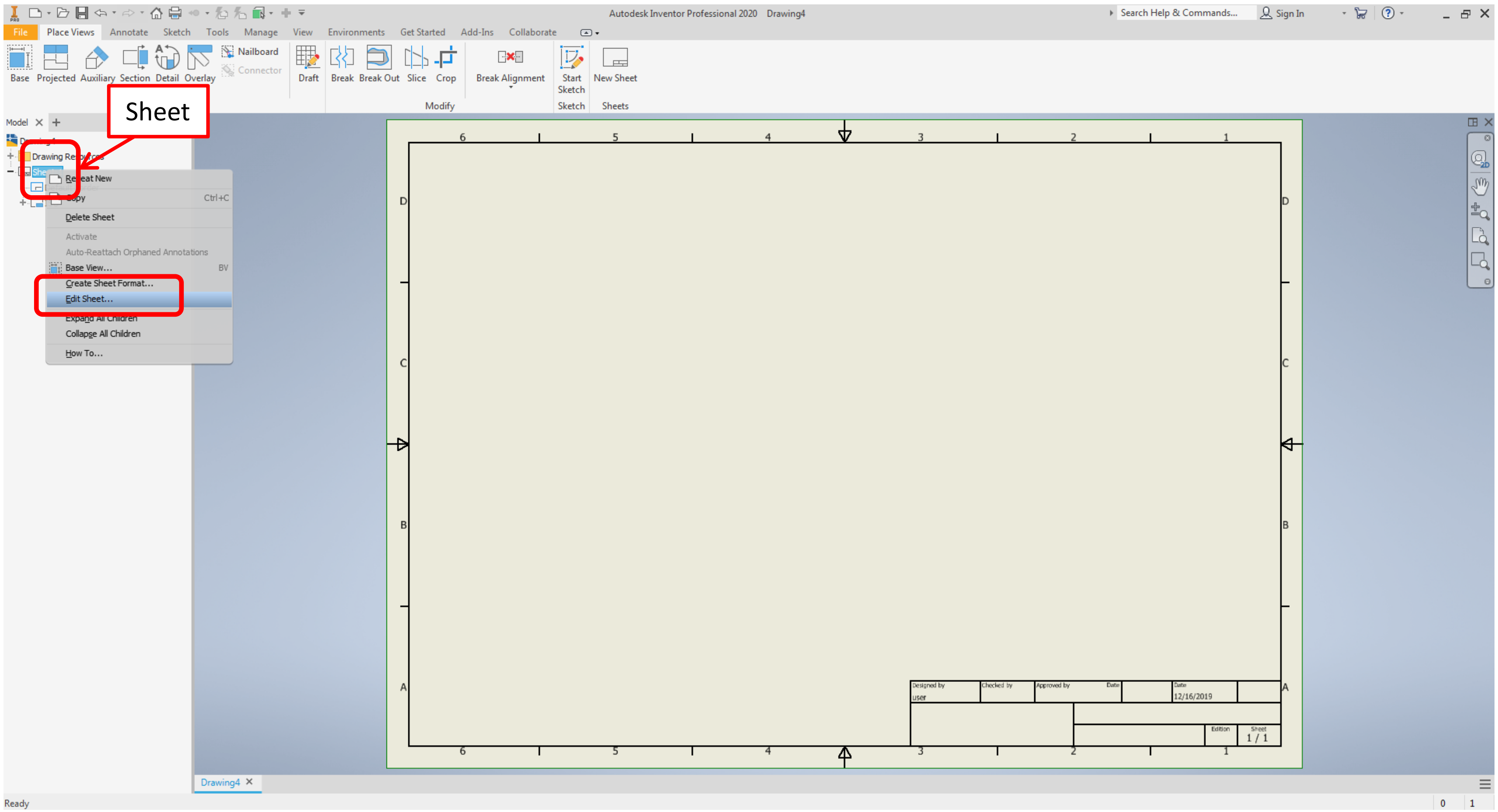
Click on “New”, select “ISO.dwg” and click “Create”



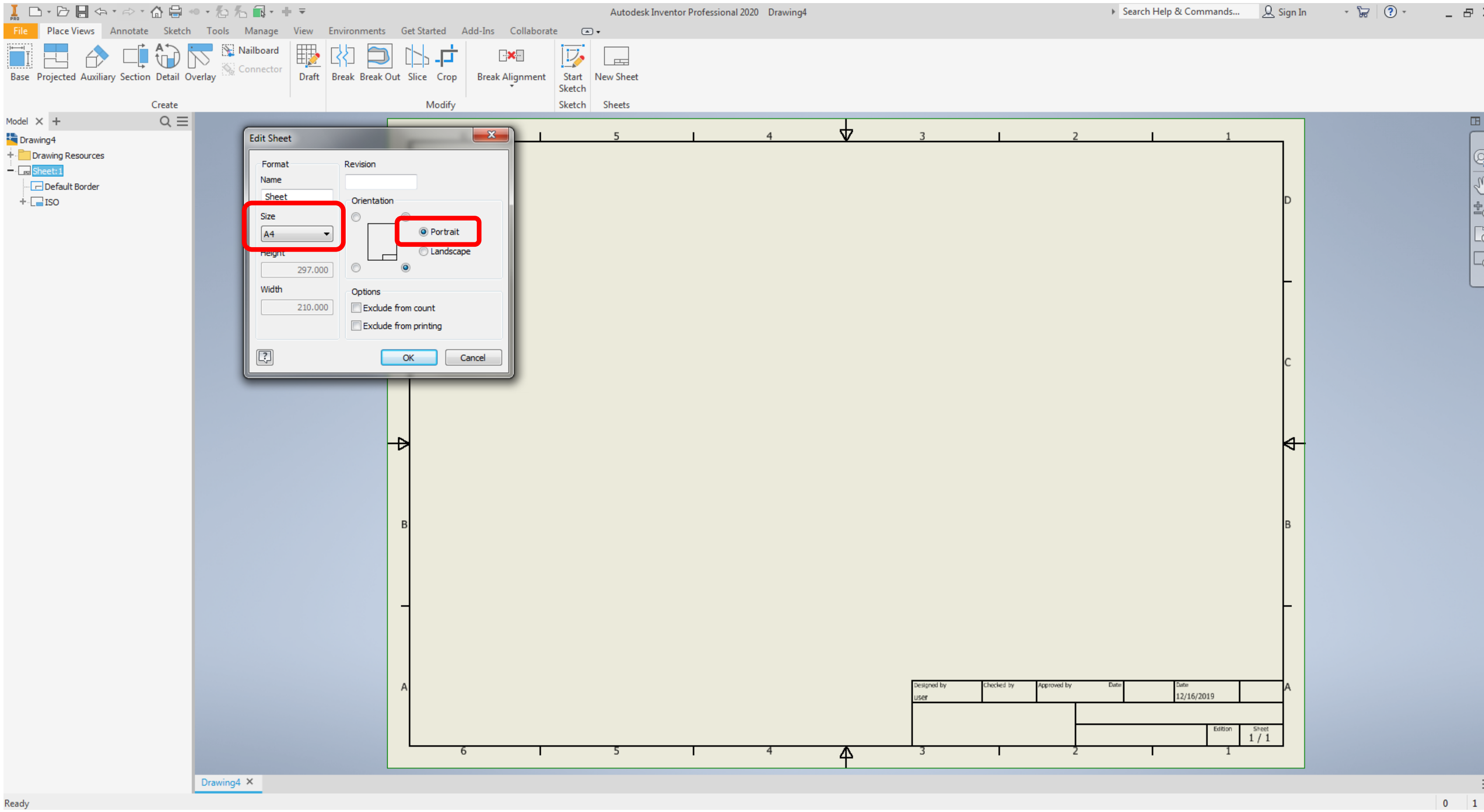
This is what you should see. The default layout is an A3 sheet, which we want to make into an A4 sheet.



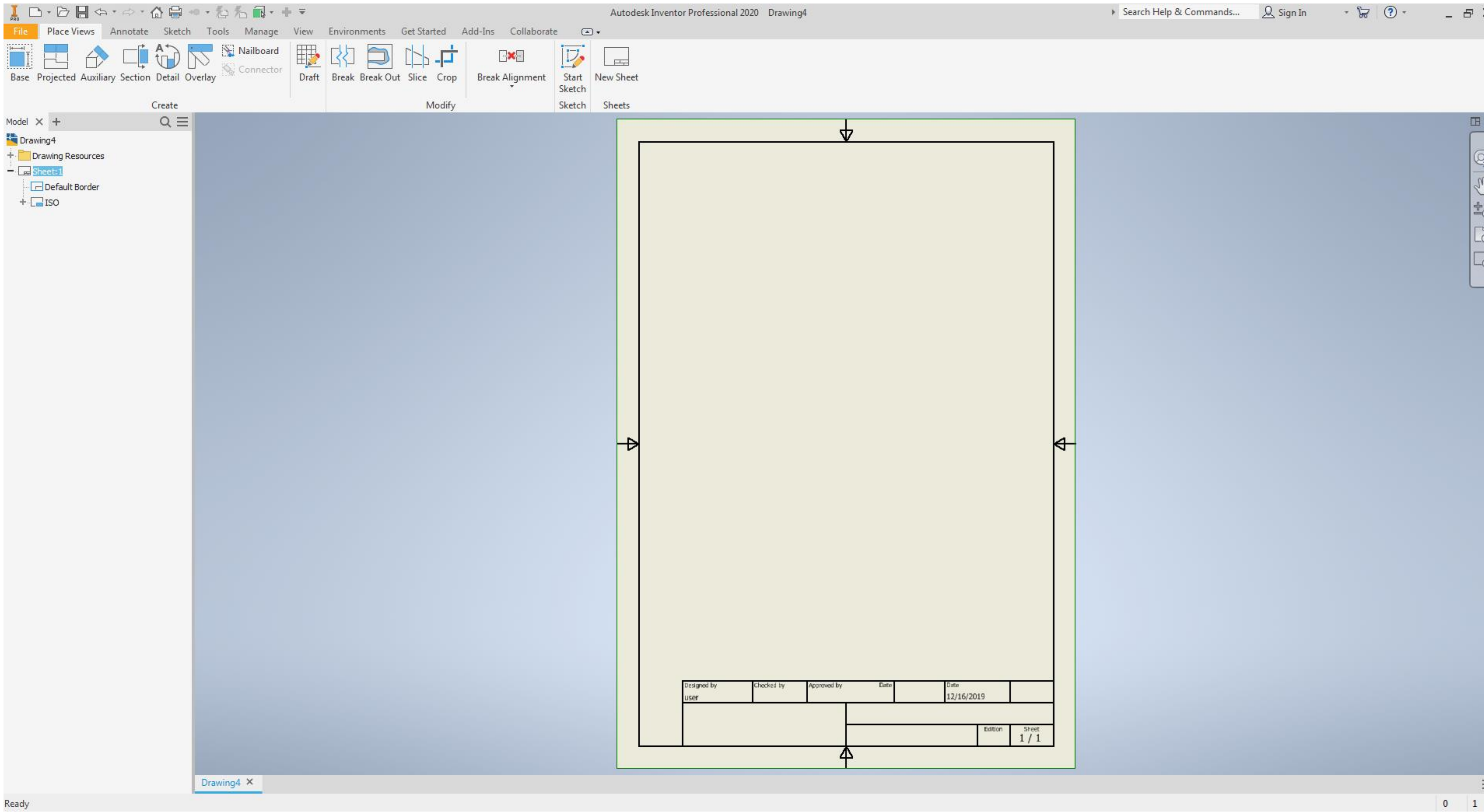
Right click on “Sheet” and select “Edit Sheet”



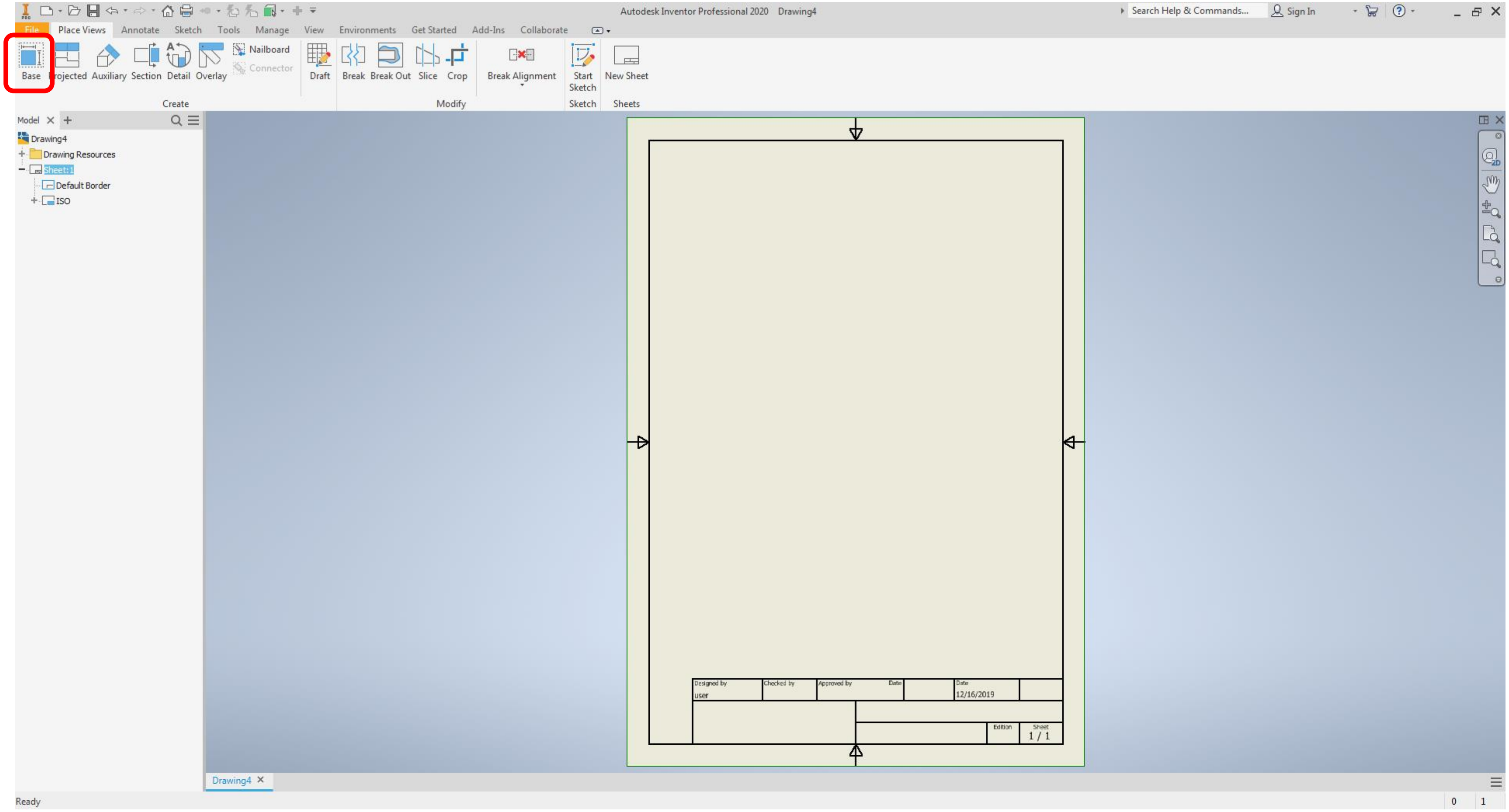
Select the required Sheet Size (A4) and Orientation (portrait)
Click “OK”



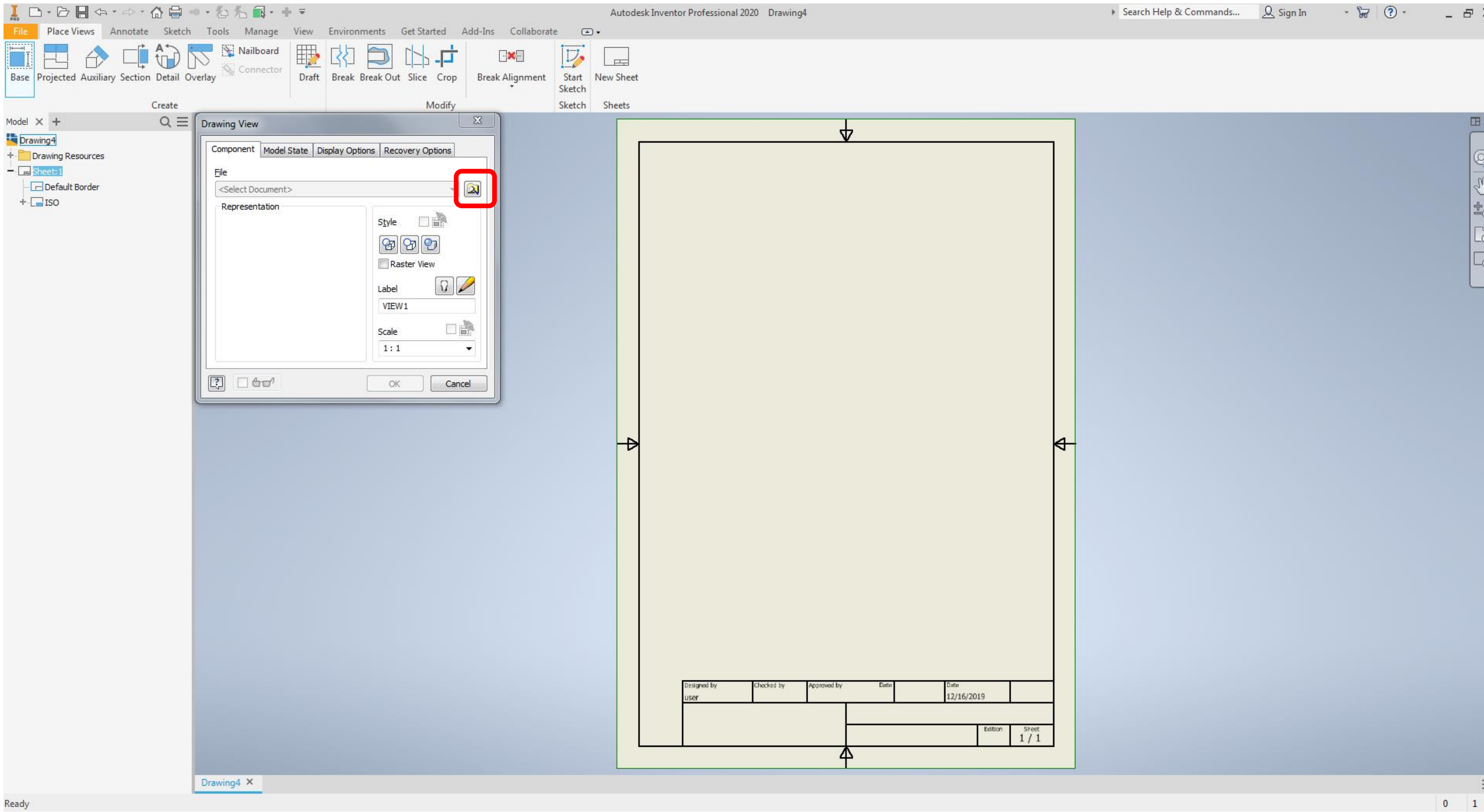
This is what you should now see. We will now bring in the “Support Plate” to make manufacturing drawing for it.



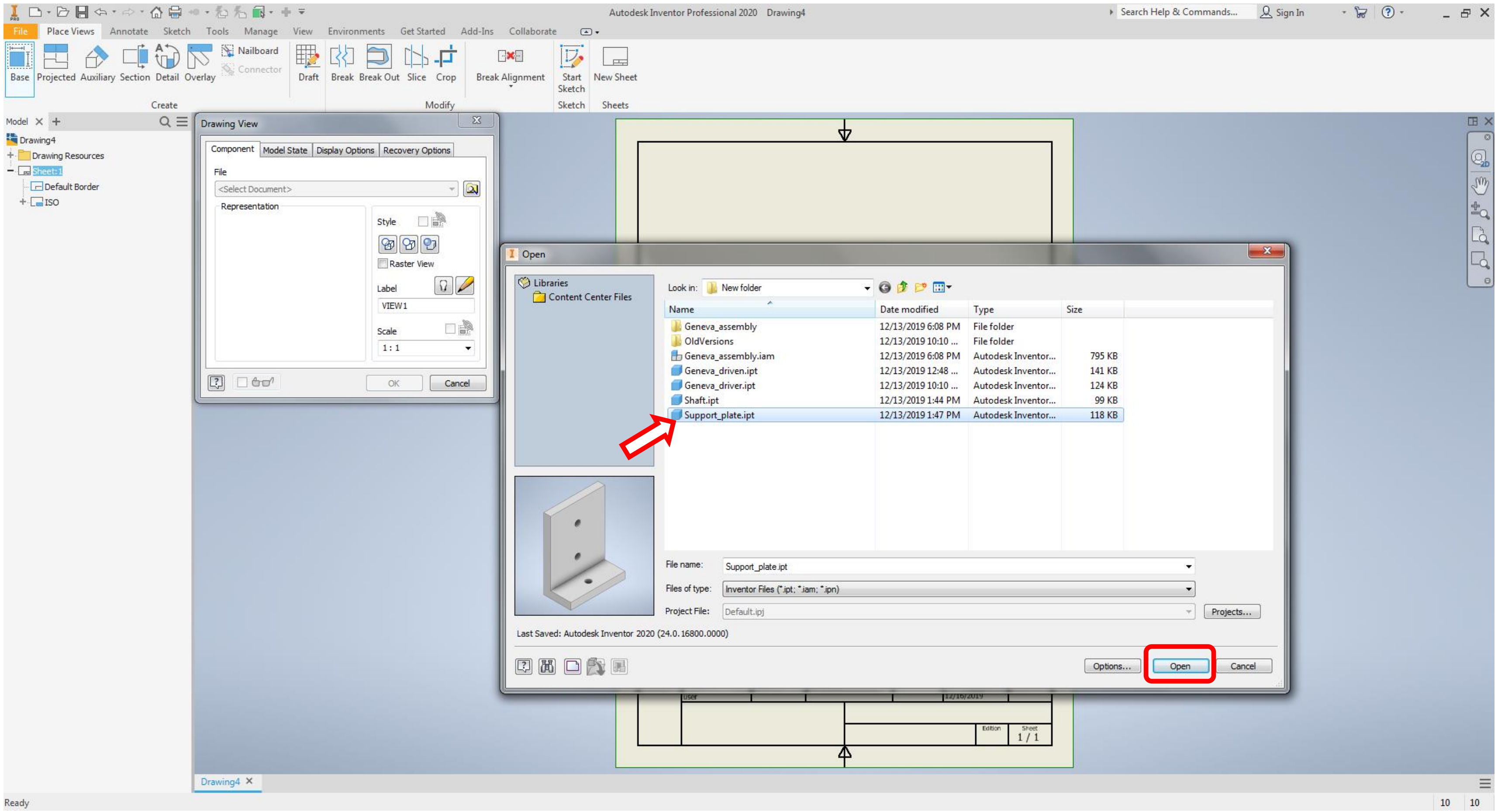
Click on “Base”



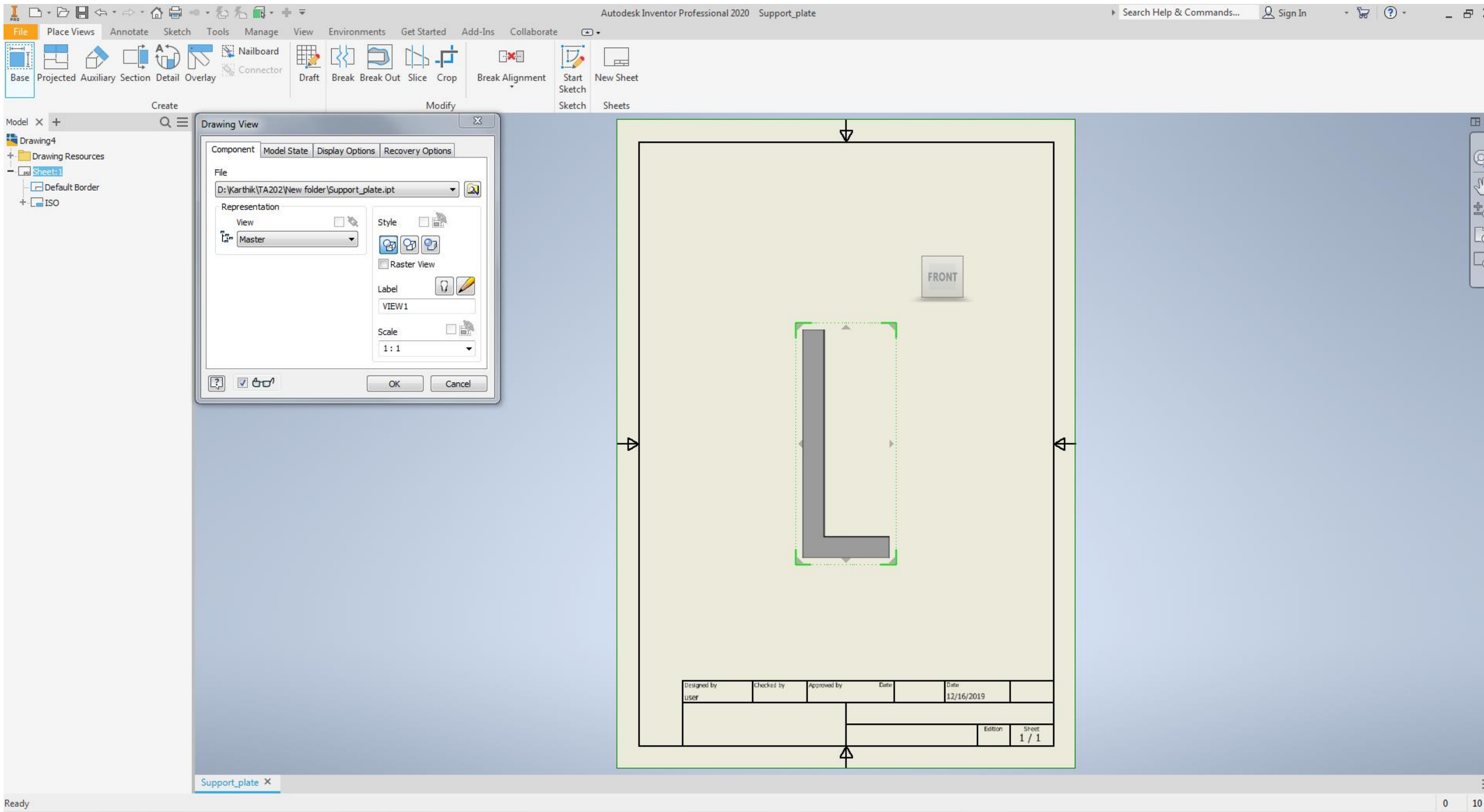
Click on “Browse” icon



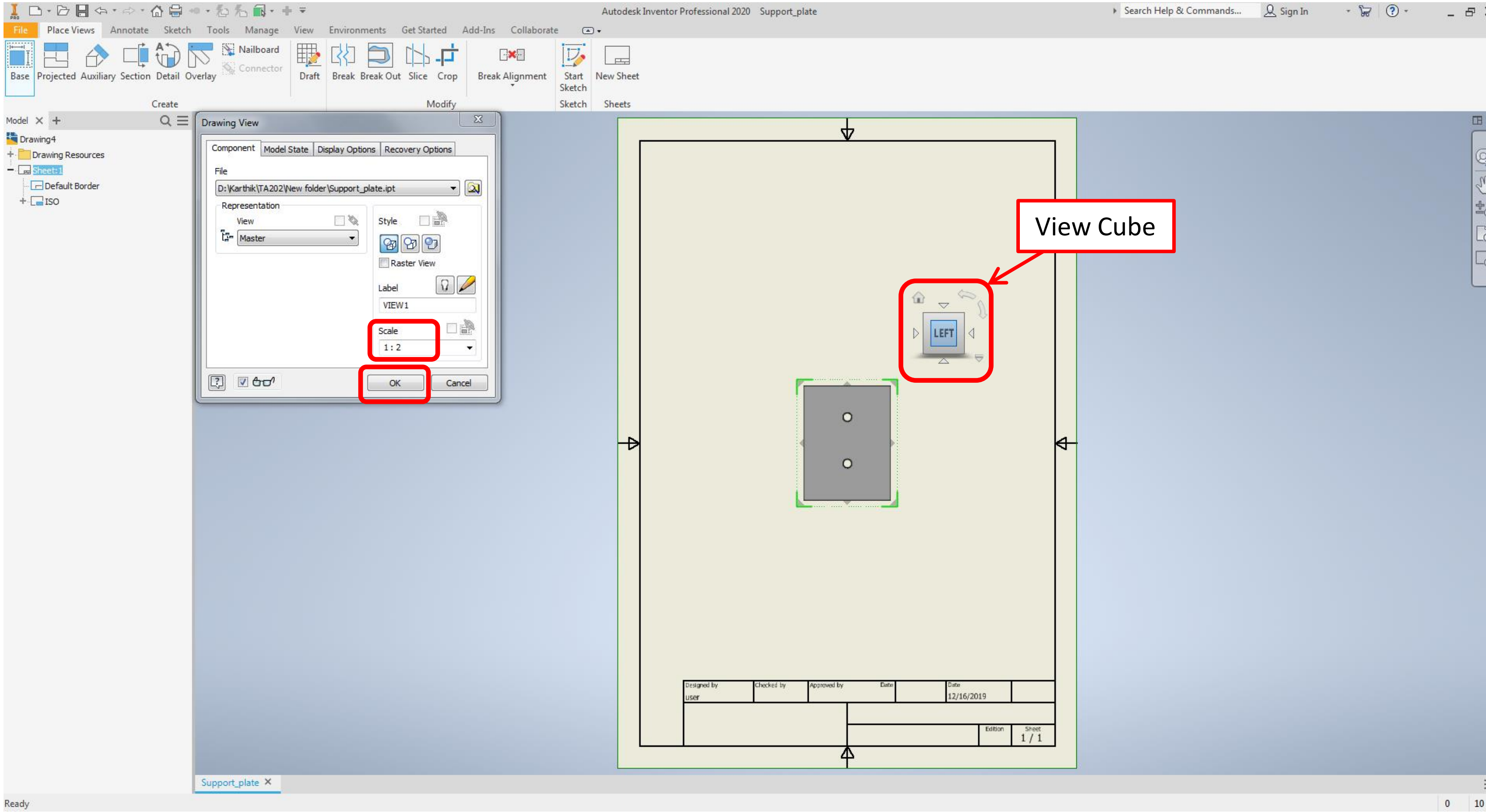
Select the required file and click “Open”



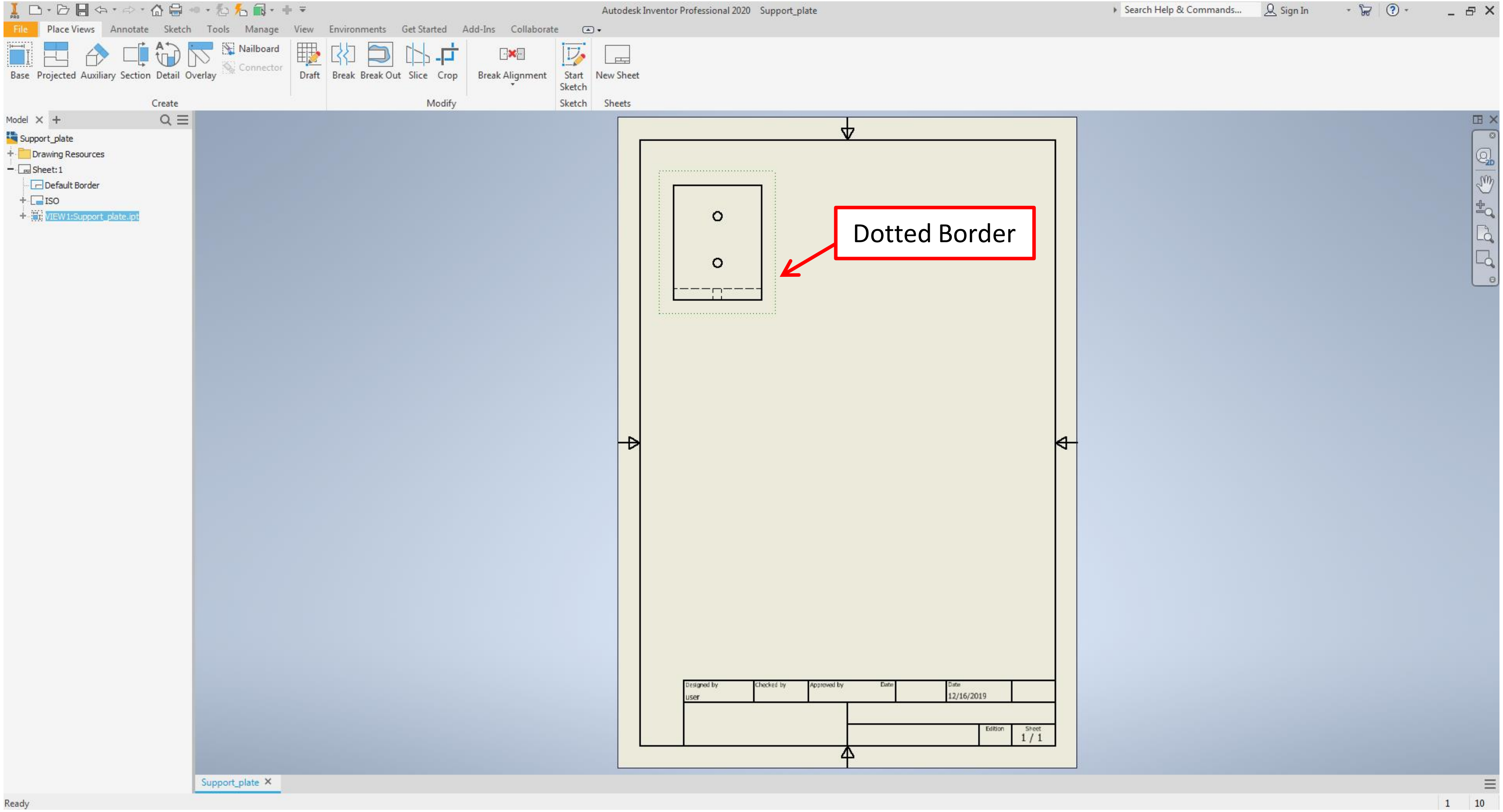
This is what you should see



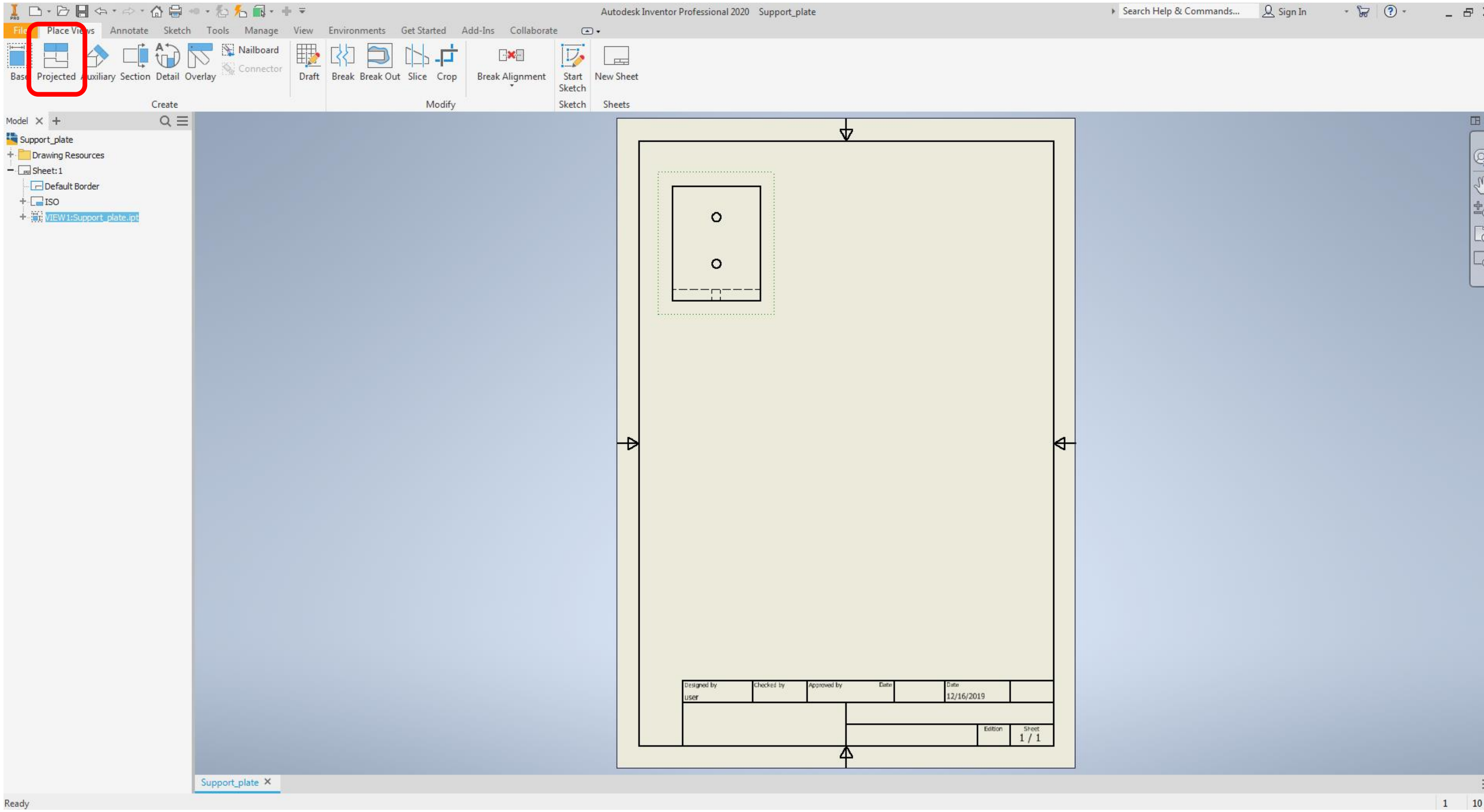
Select the required Base View with the help of "View Cube", set the scale(1:2 in this case)
Click "OK"
Note: 1:1, 1:2, 1:5 and 1:10 are standard scales generally used in Engineering drawings



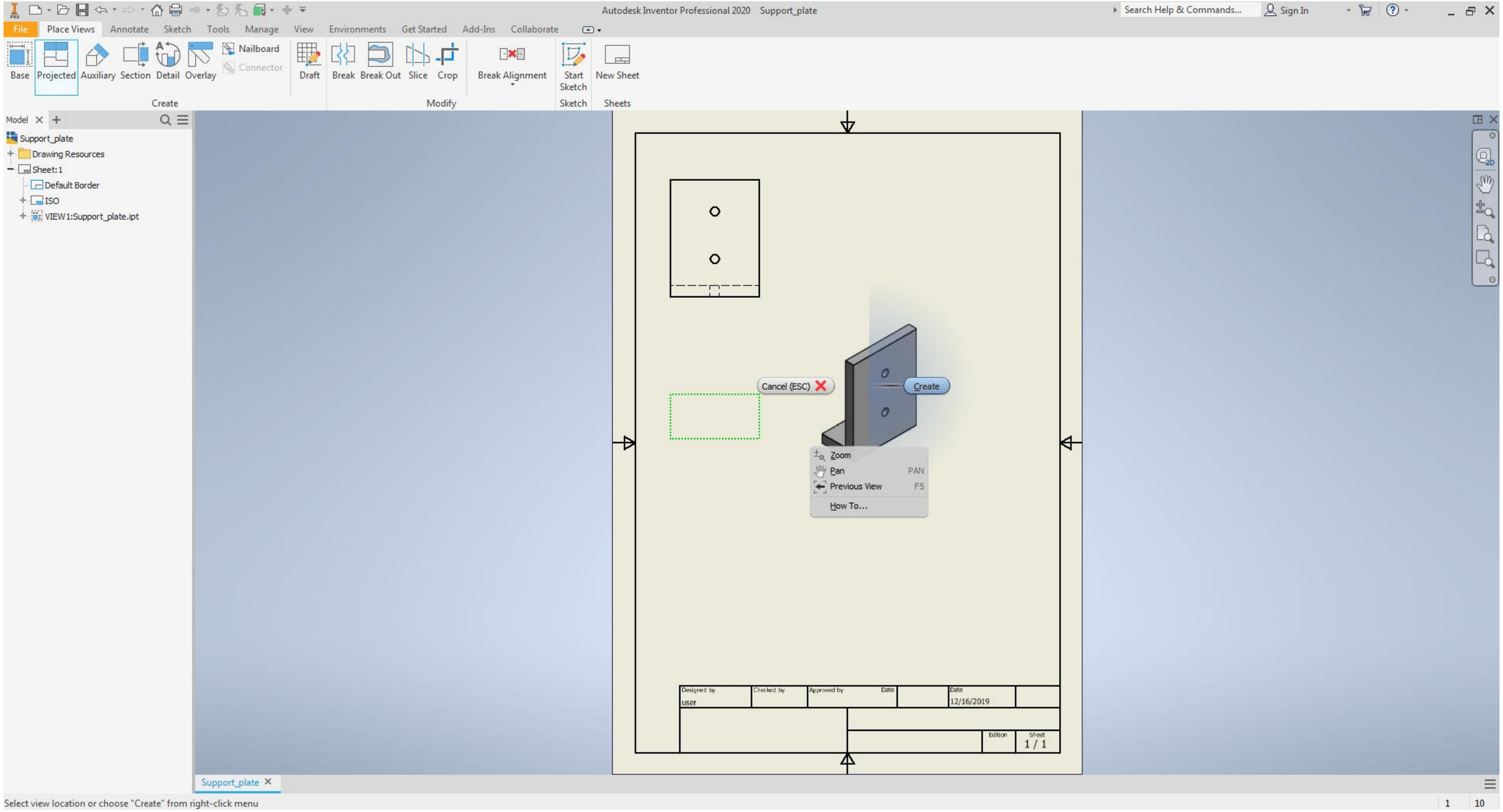
Click on the dotted border around the base view and drag it to the required location in the sheet



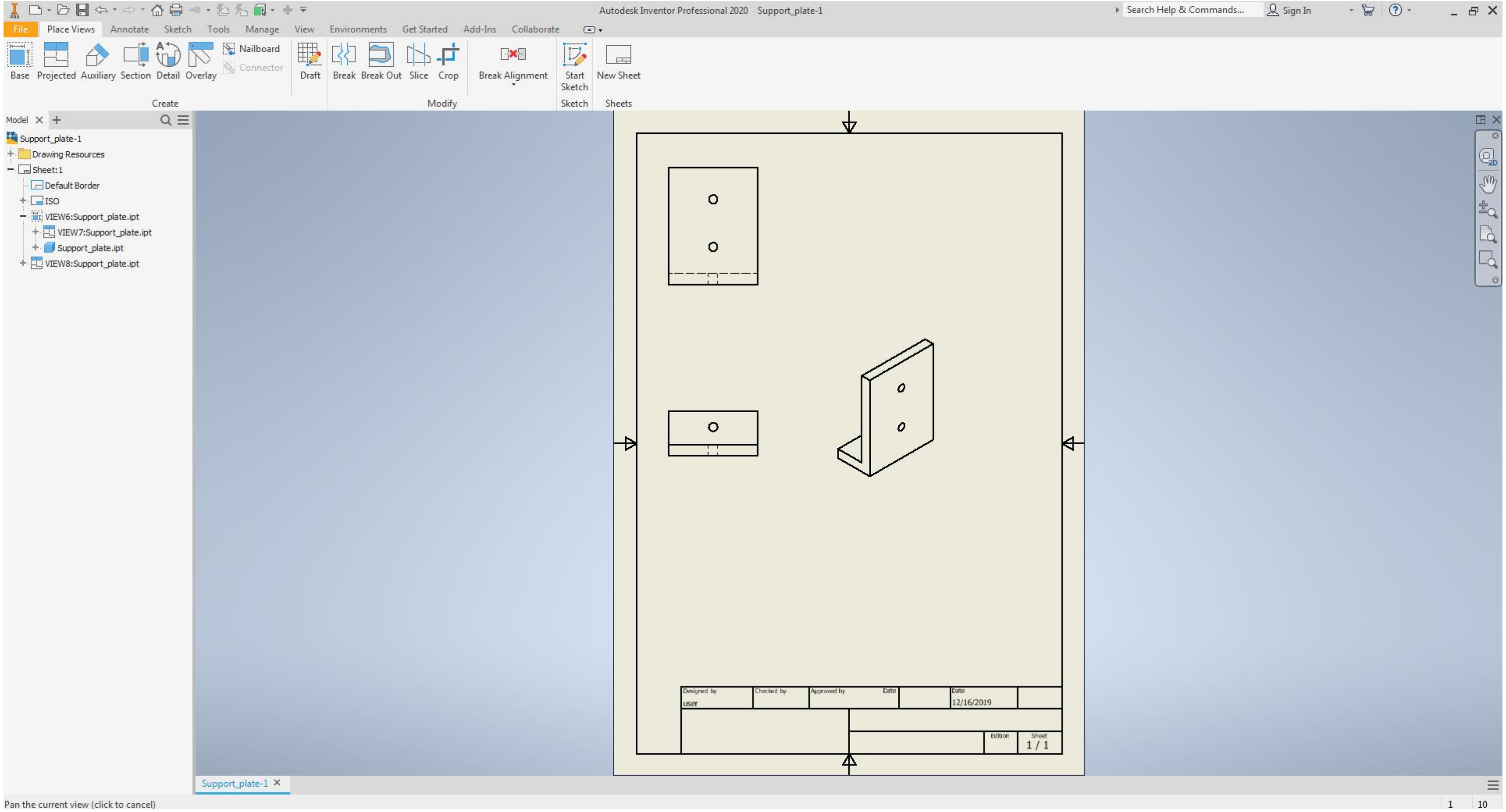
Click on “Projected”



Place the required views
Right click and select “Create”.

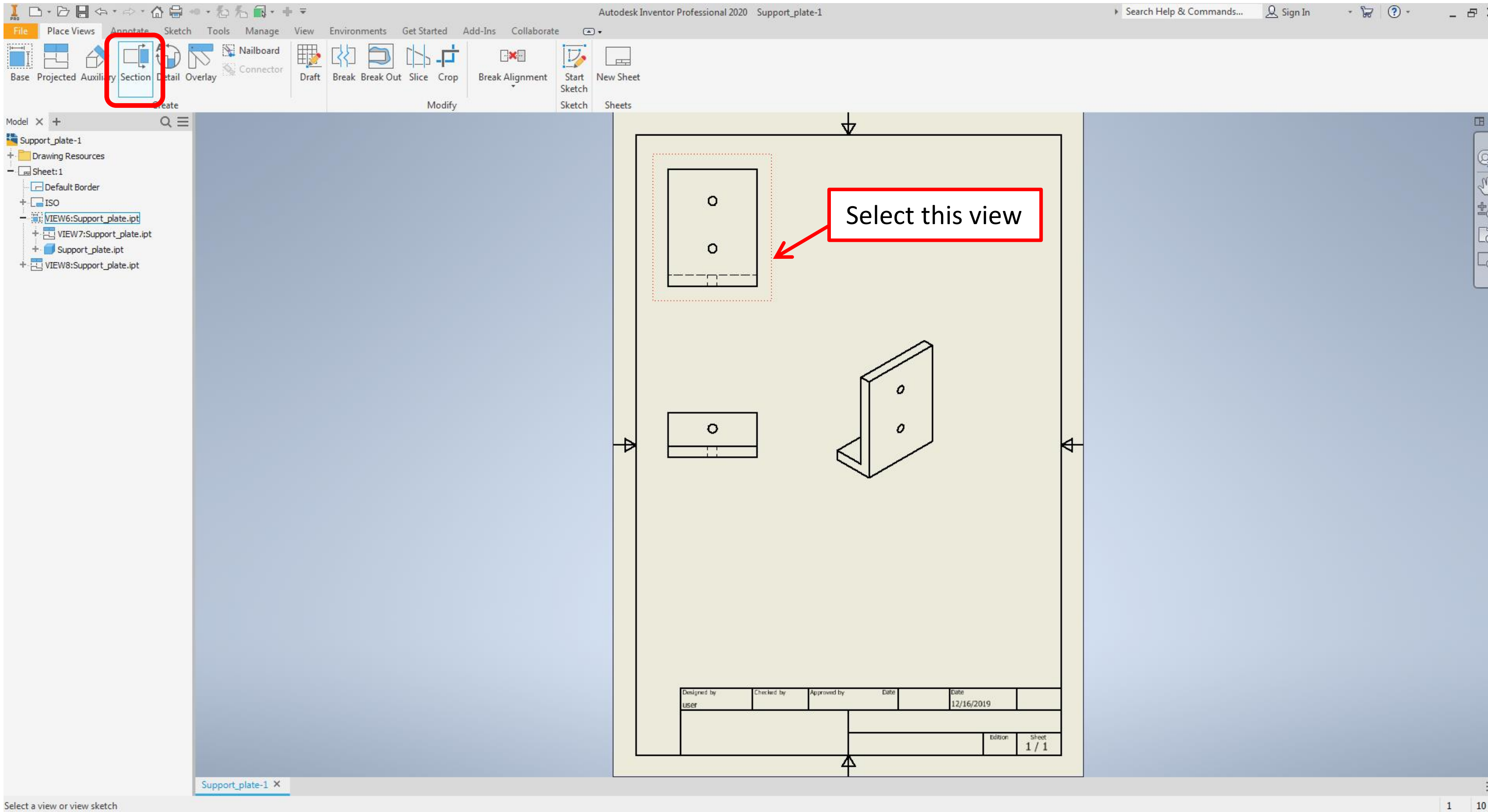


You may end up with something like this



To create Section View:

Click on “Section” and select the view from which you need the Sectional View

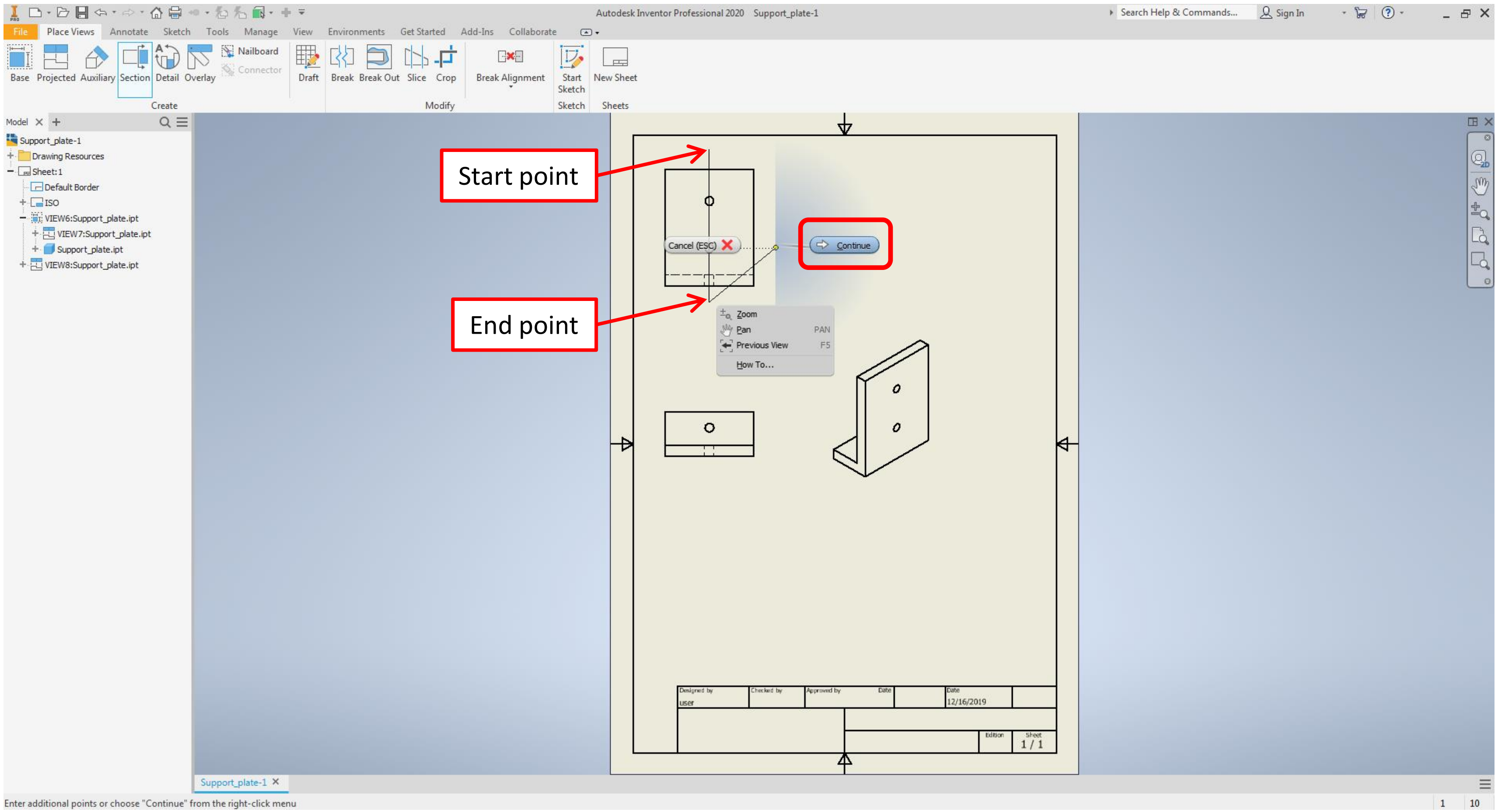


Draw a line along which you need sectional view

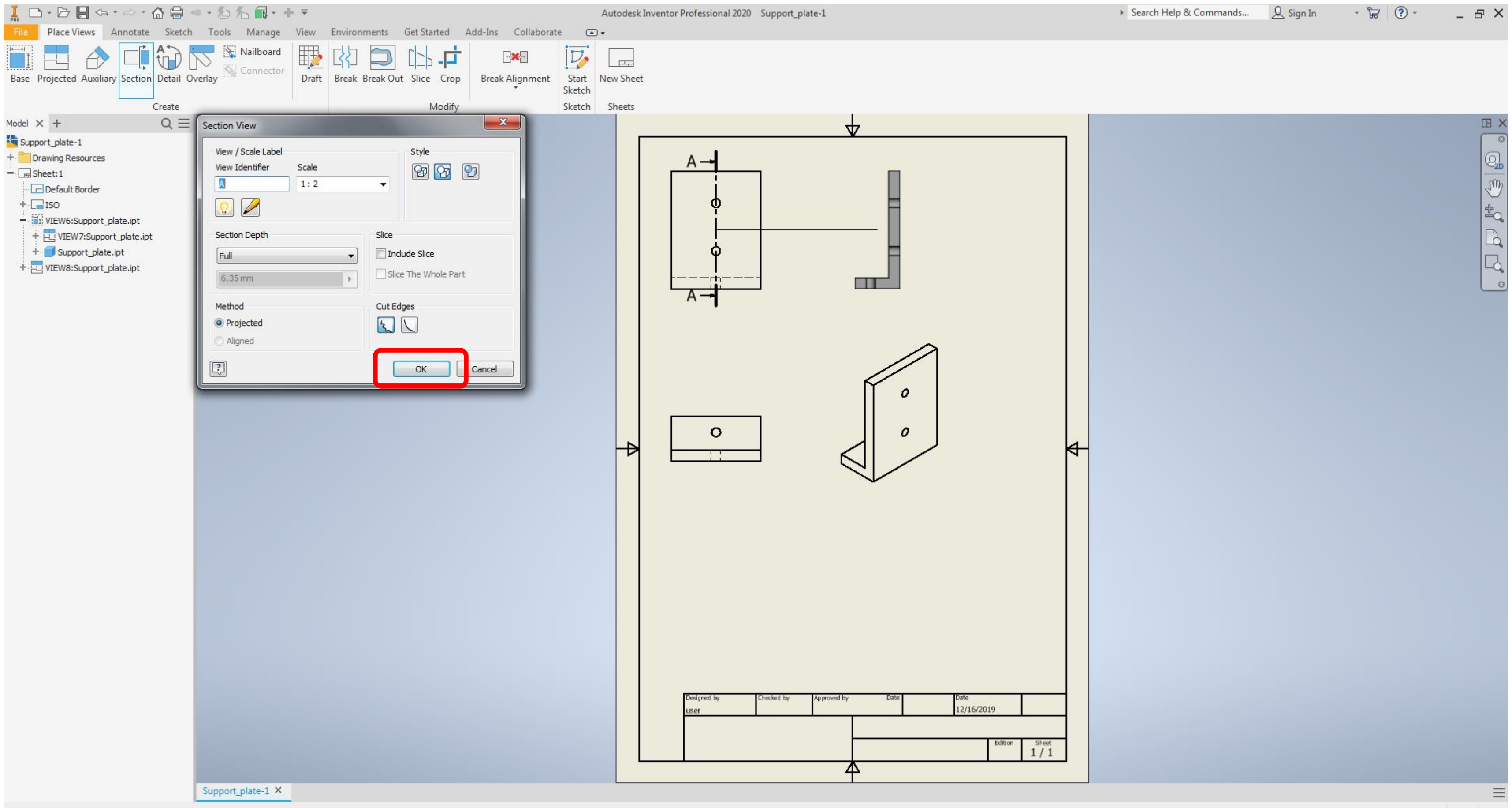
*Make sure that the start point and the end point of the sectional line are outside the part

*Make sure that the sectional line passes through the center of the hole

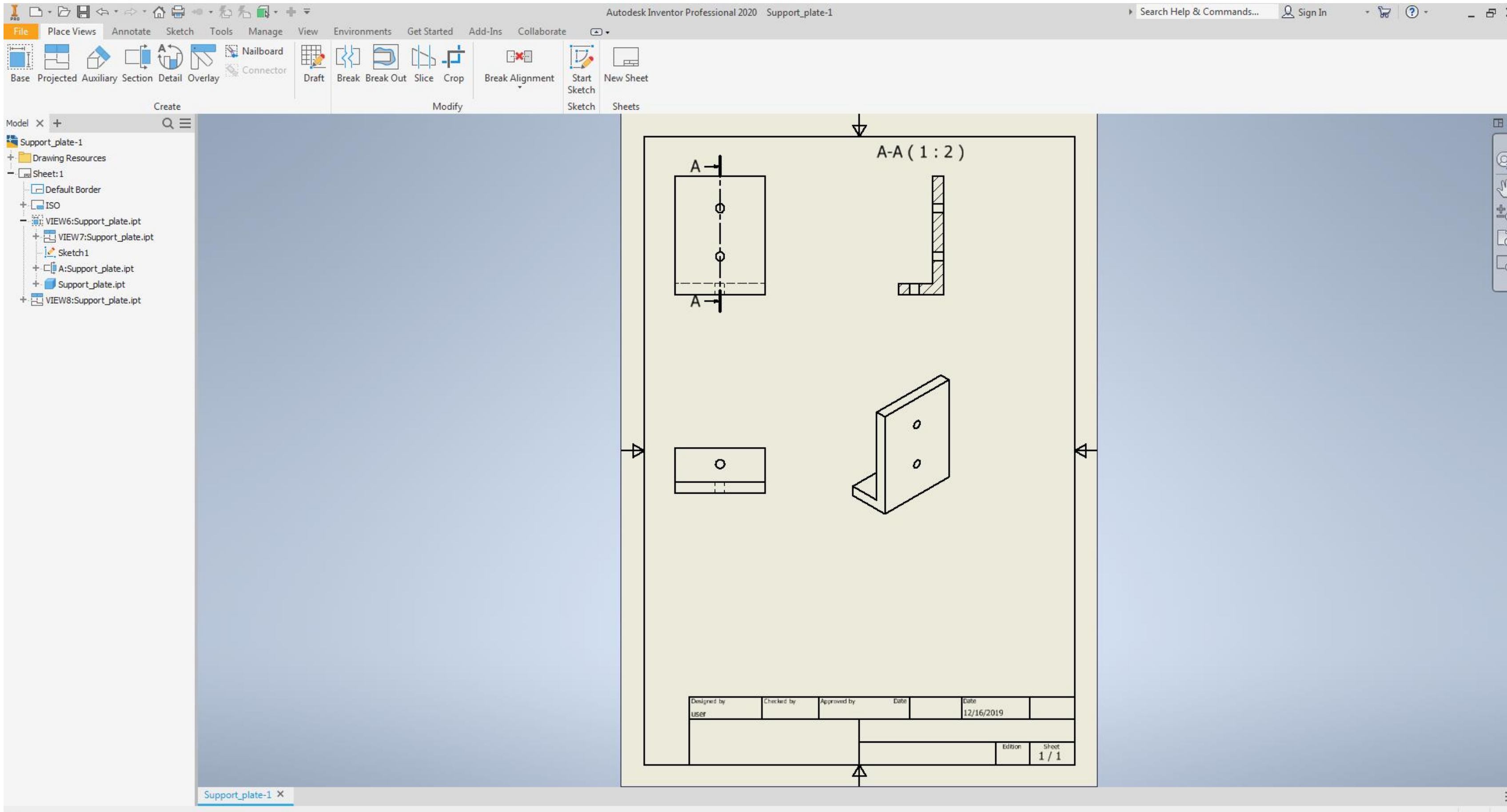
Right click and select “Continue”



Place the view and click “OK”



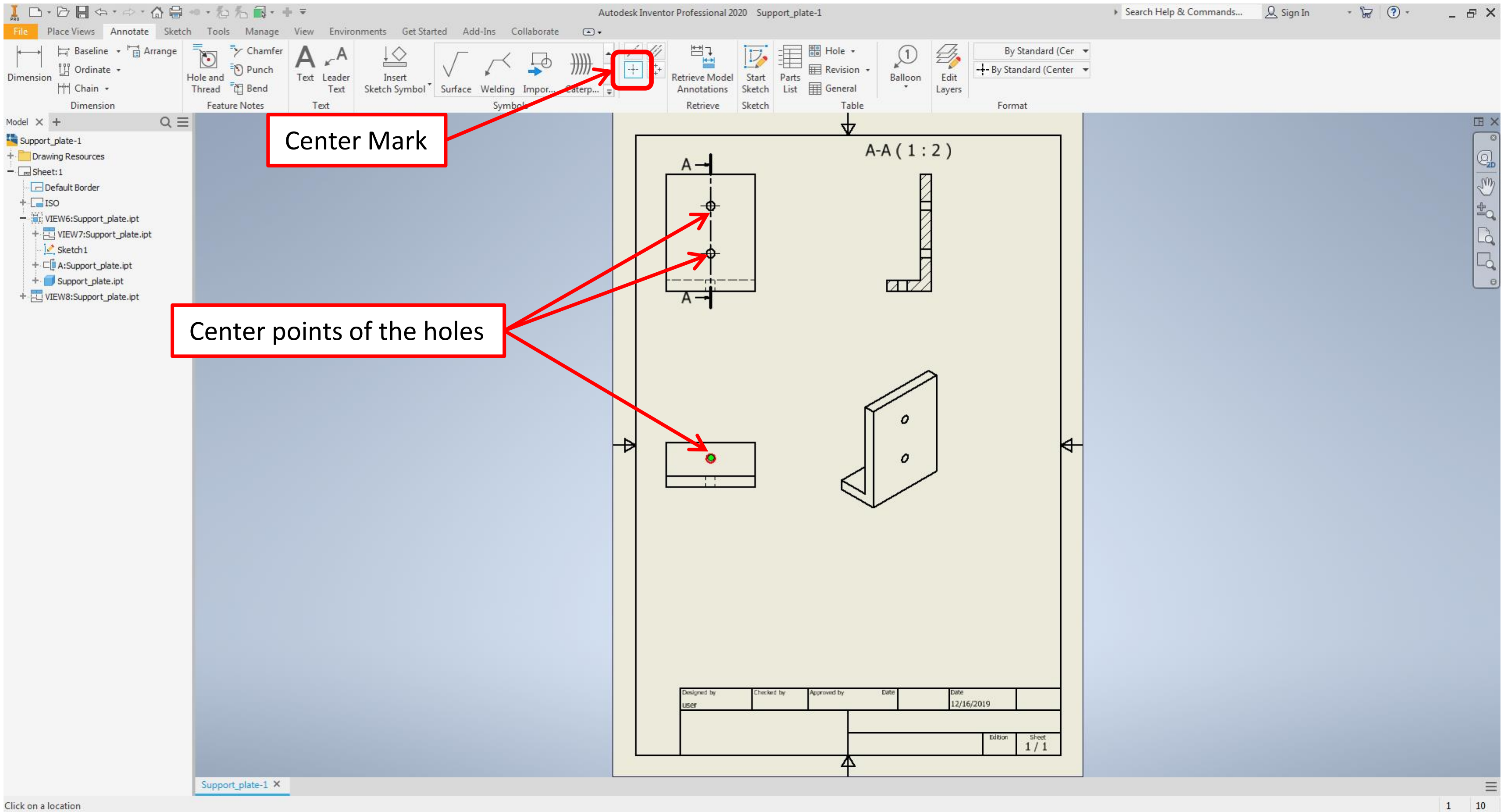
You may end up with something like this



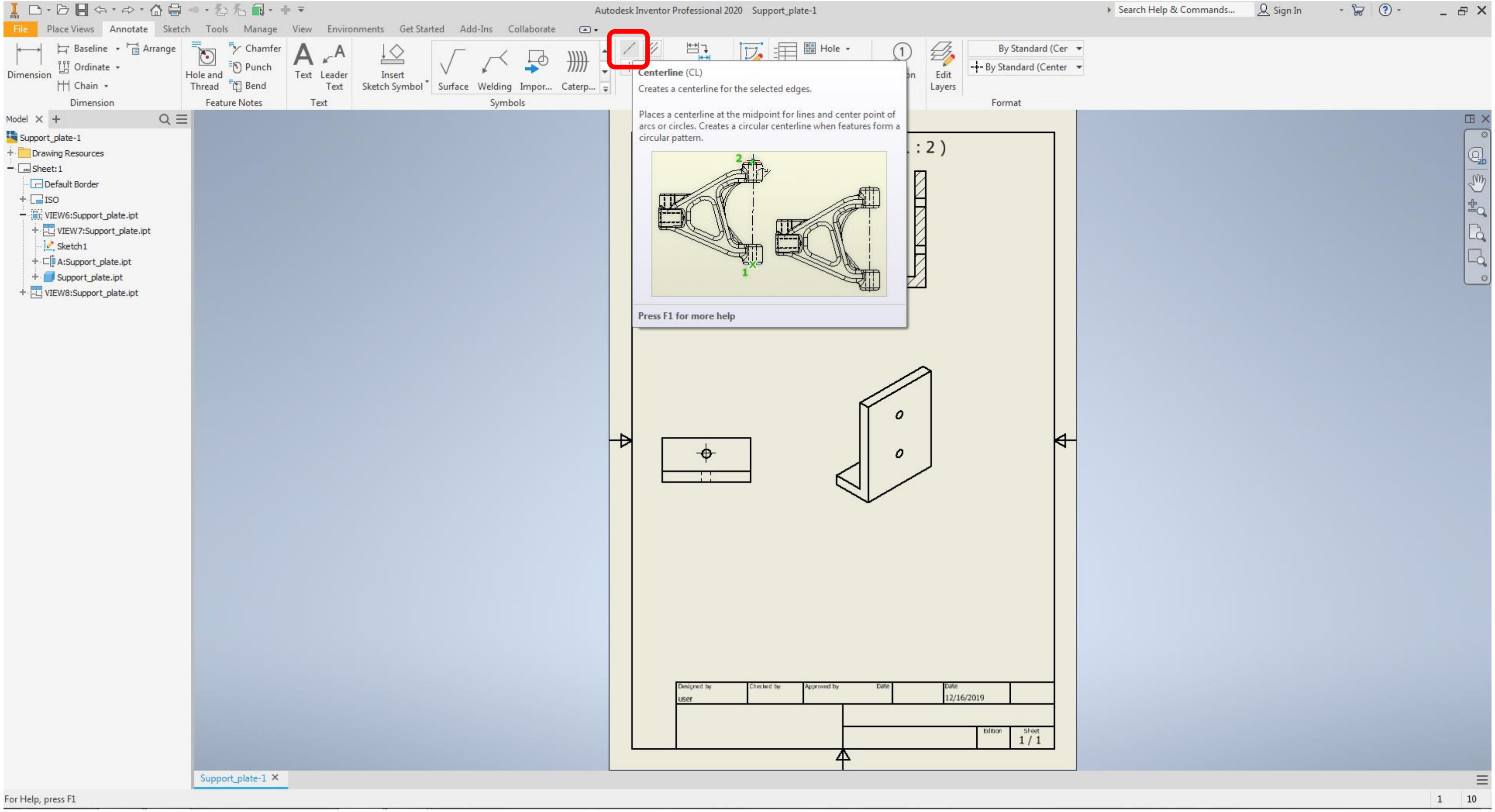
To add Center Mark for the holes

Click on “Center Mark” and click on center points of the holes

*You should see a green dot when you hover your mouse around the hole indicating center of the hole



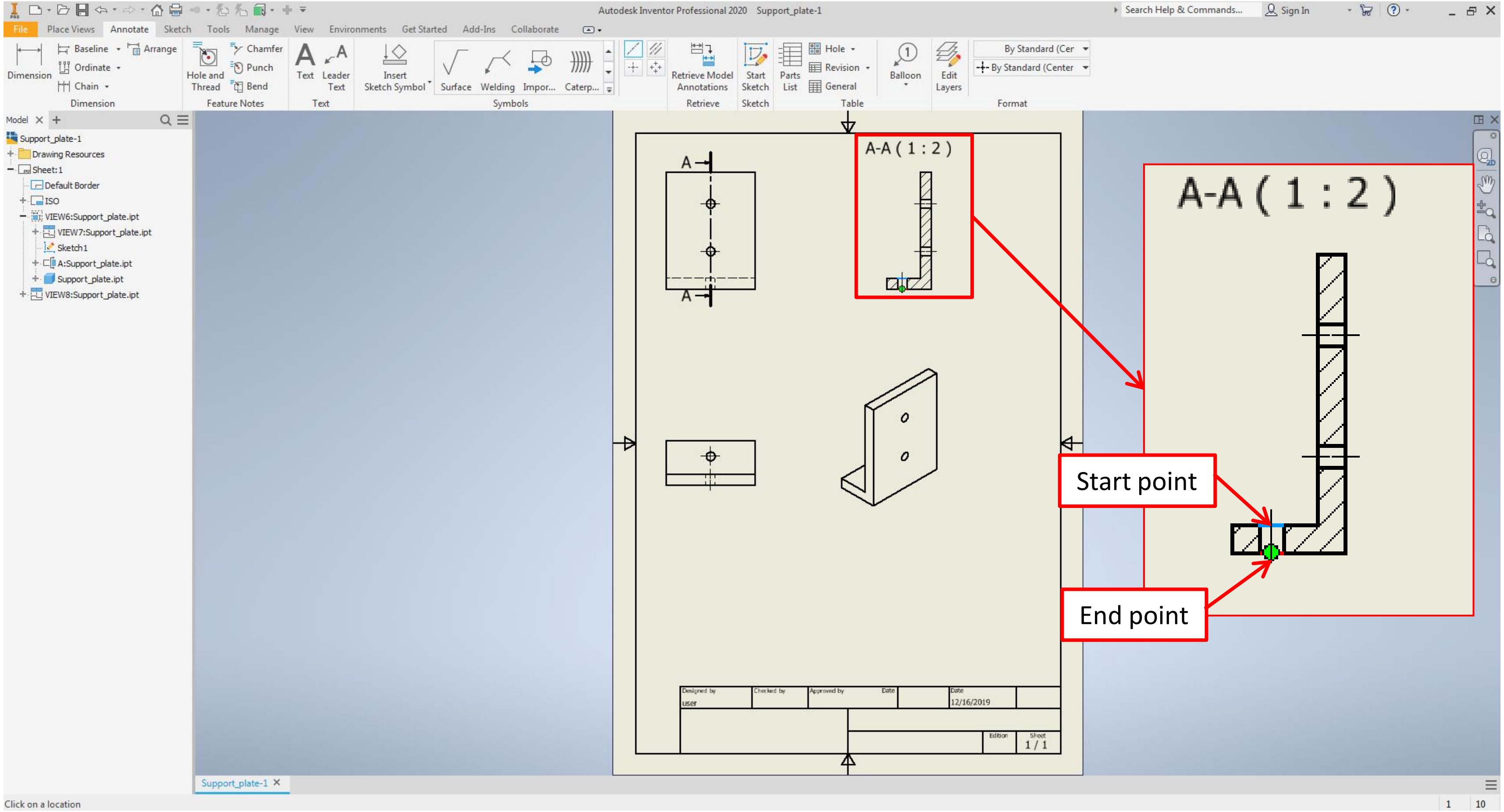
To add Centerlines
Click on “Centerline”



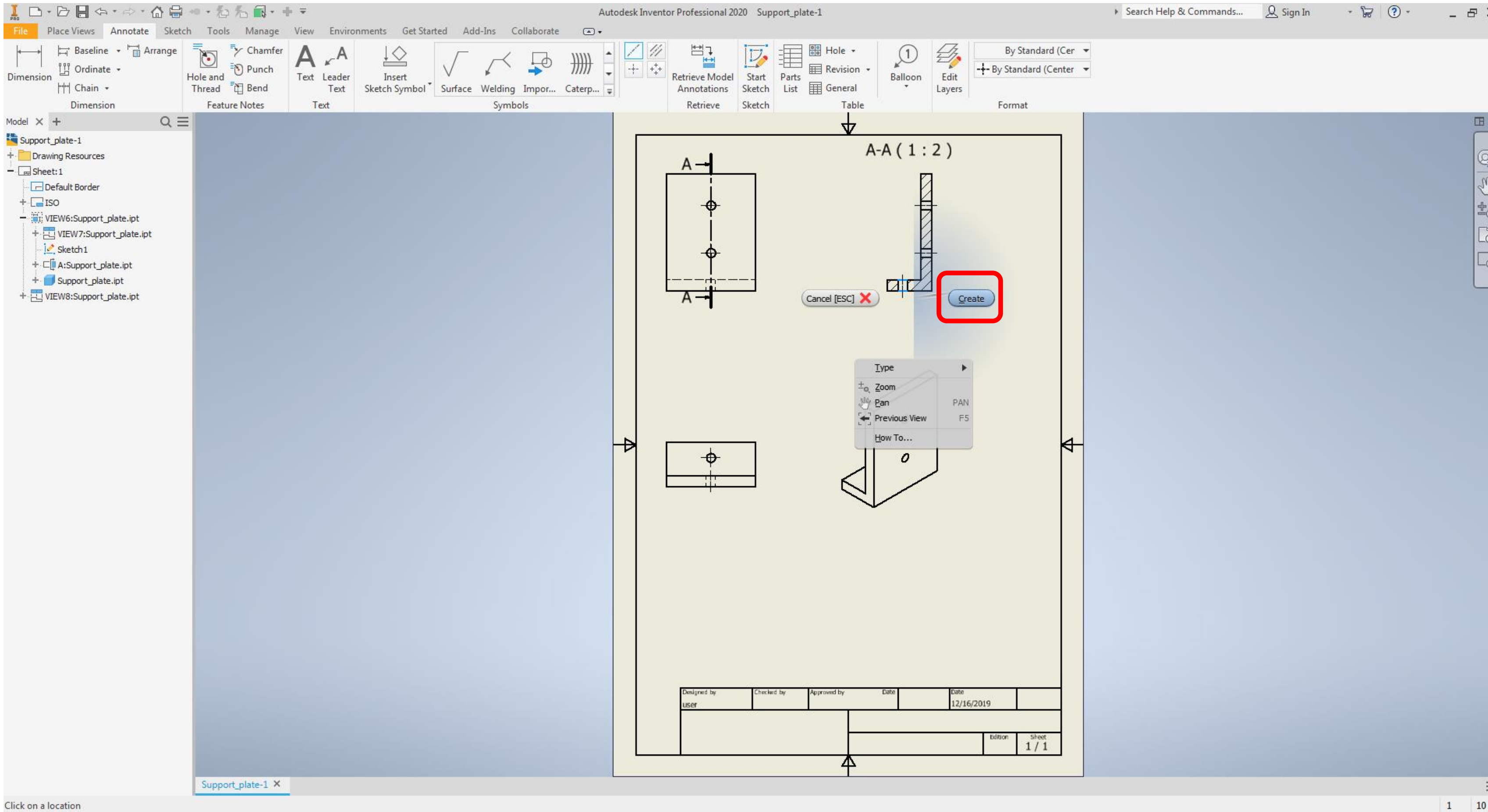
Draw centerlines for holes as shown below

*While selecting the start and end points of the line, make sure to select midpoints as shown below. You should see a green dot when you hover your mouse around the line indicating the midpoint

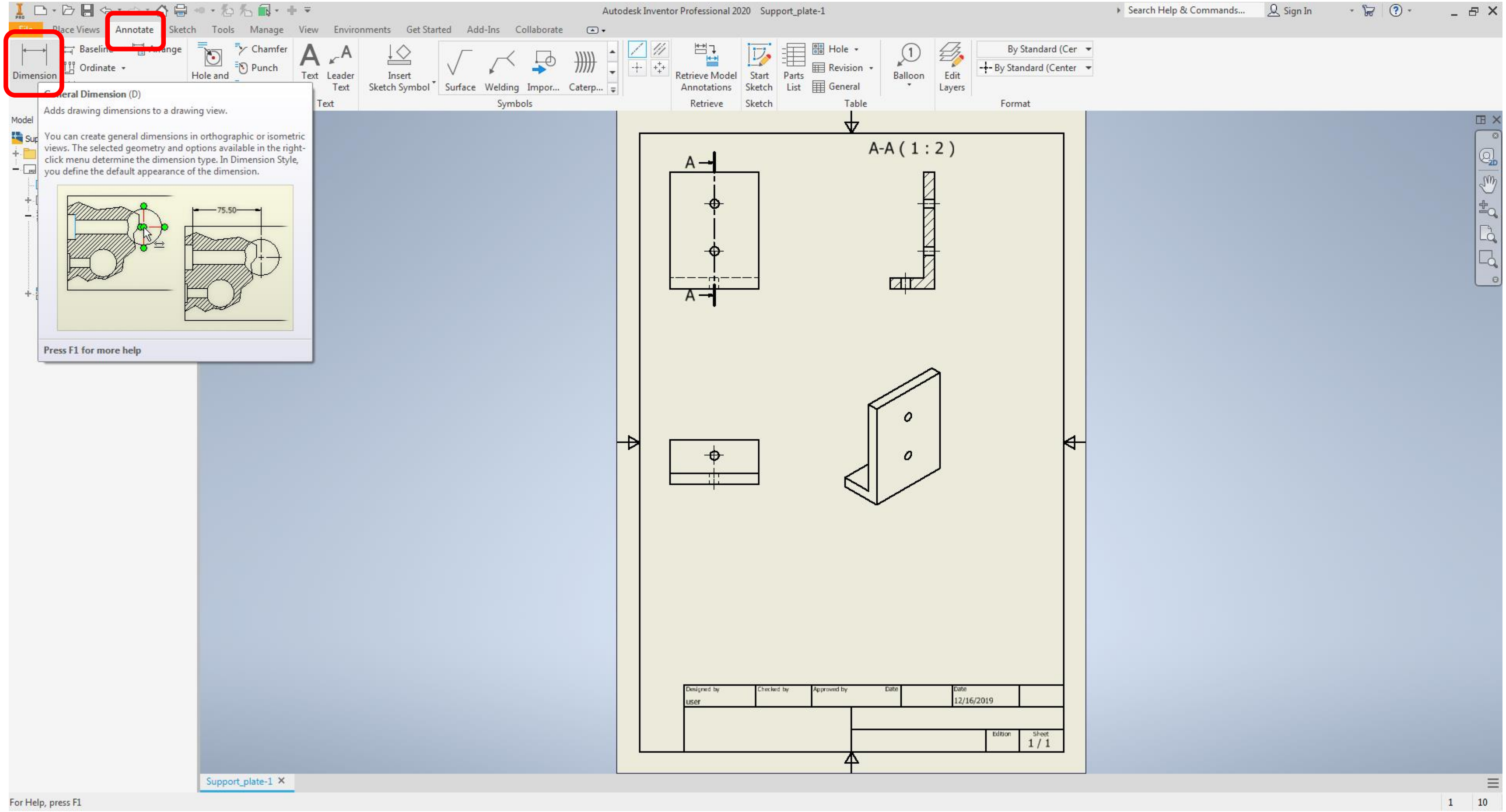
Zoom In, if necessary



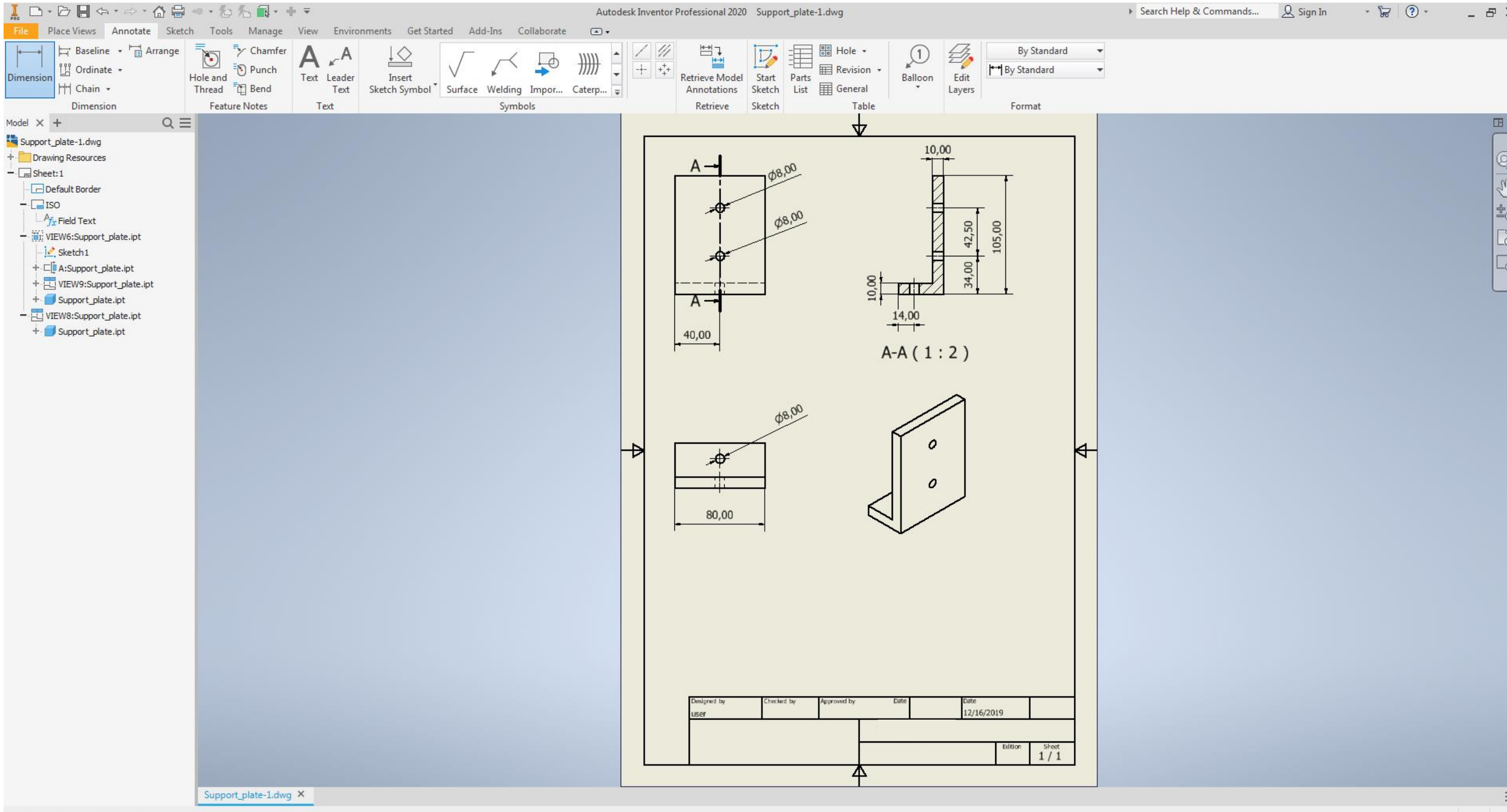
Right click and select “Create” to create Centerlines



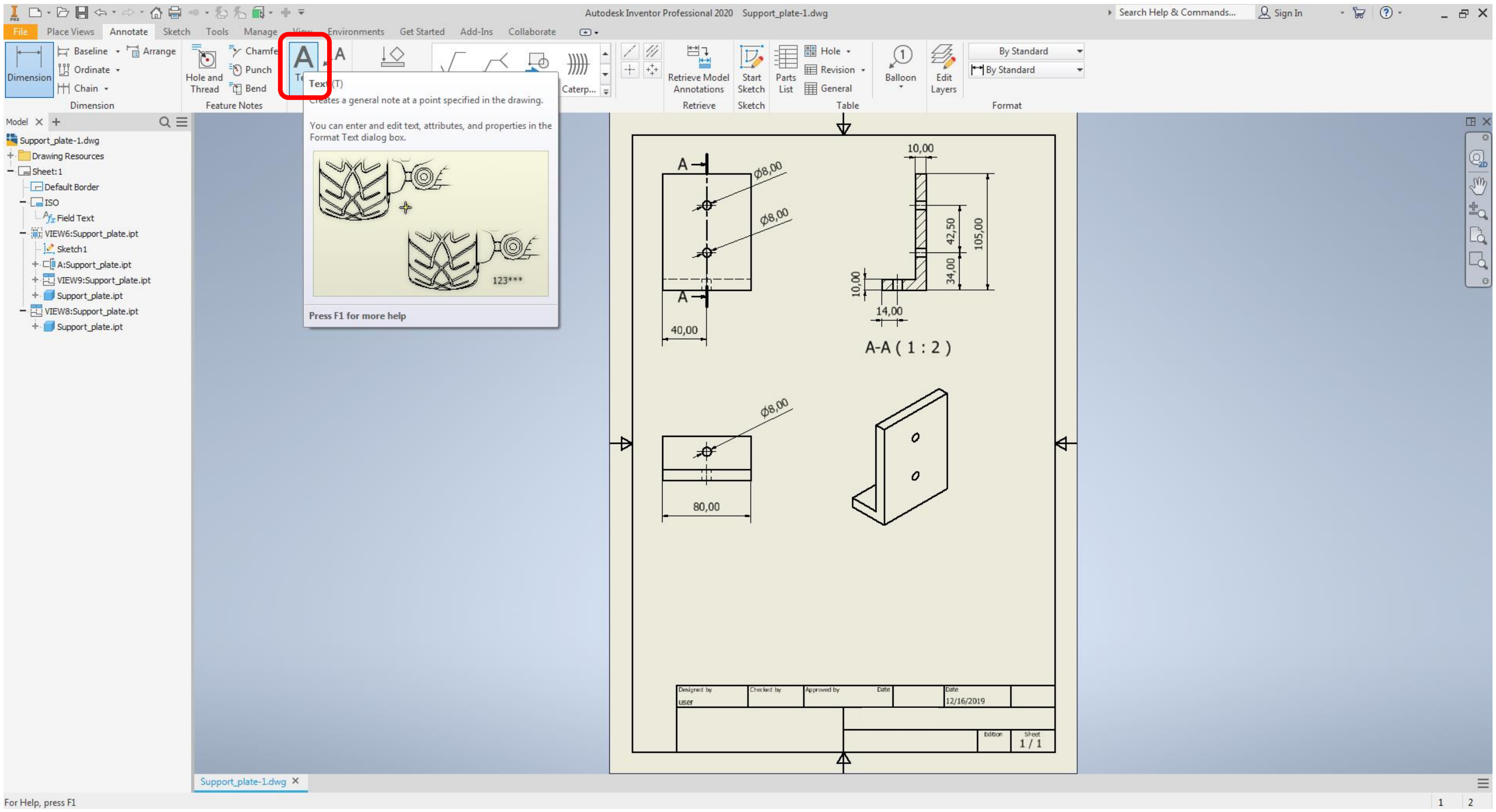
To give dimensions:
Click on “Annotate” and click on “Dimension”



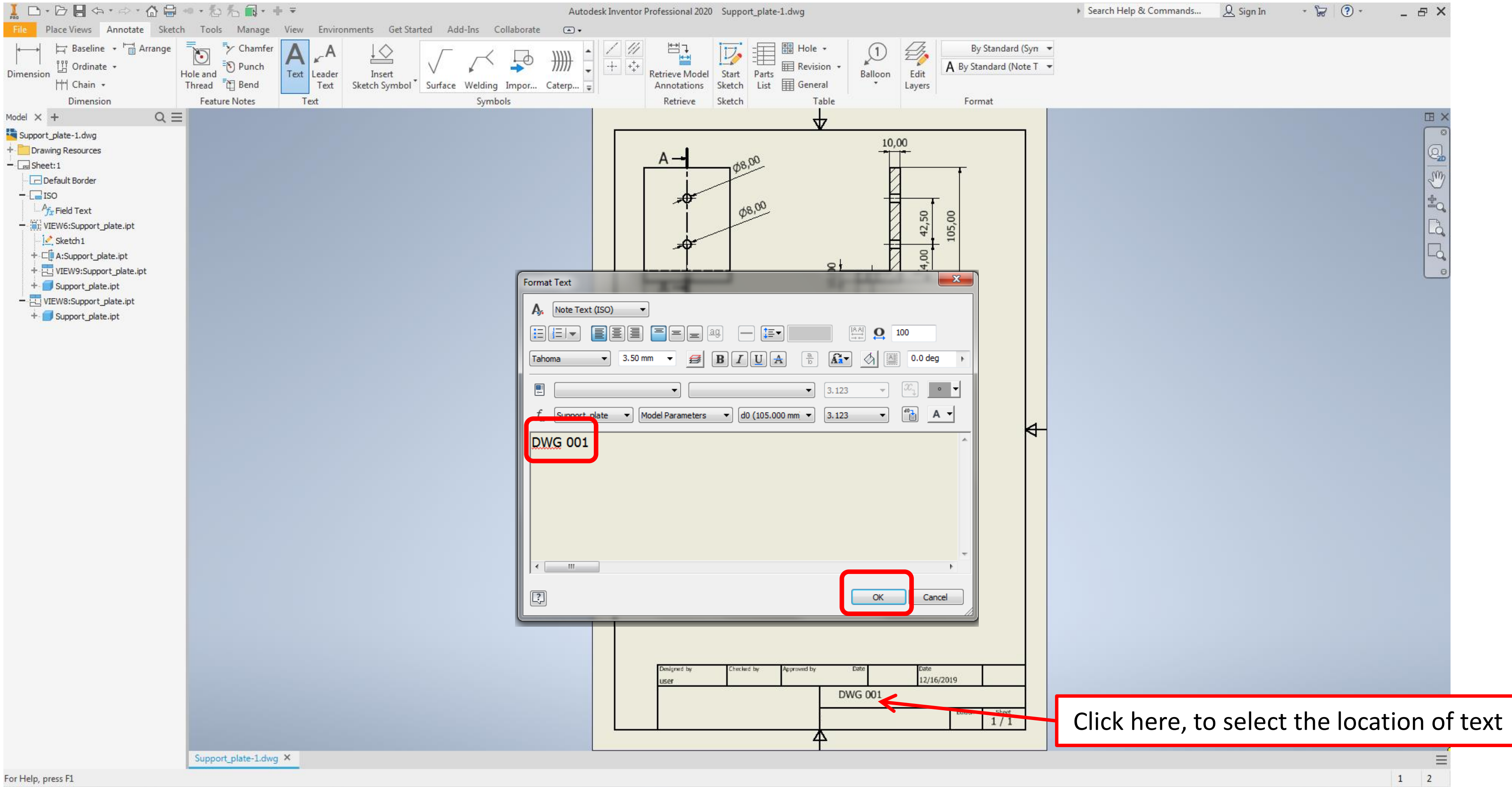
Your drawing may look something like this after dimensioning



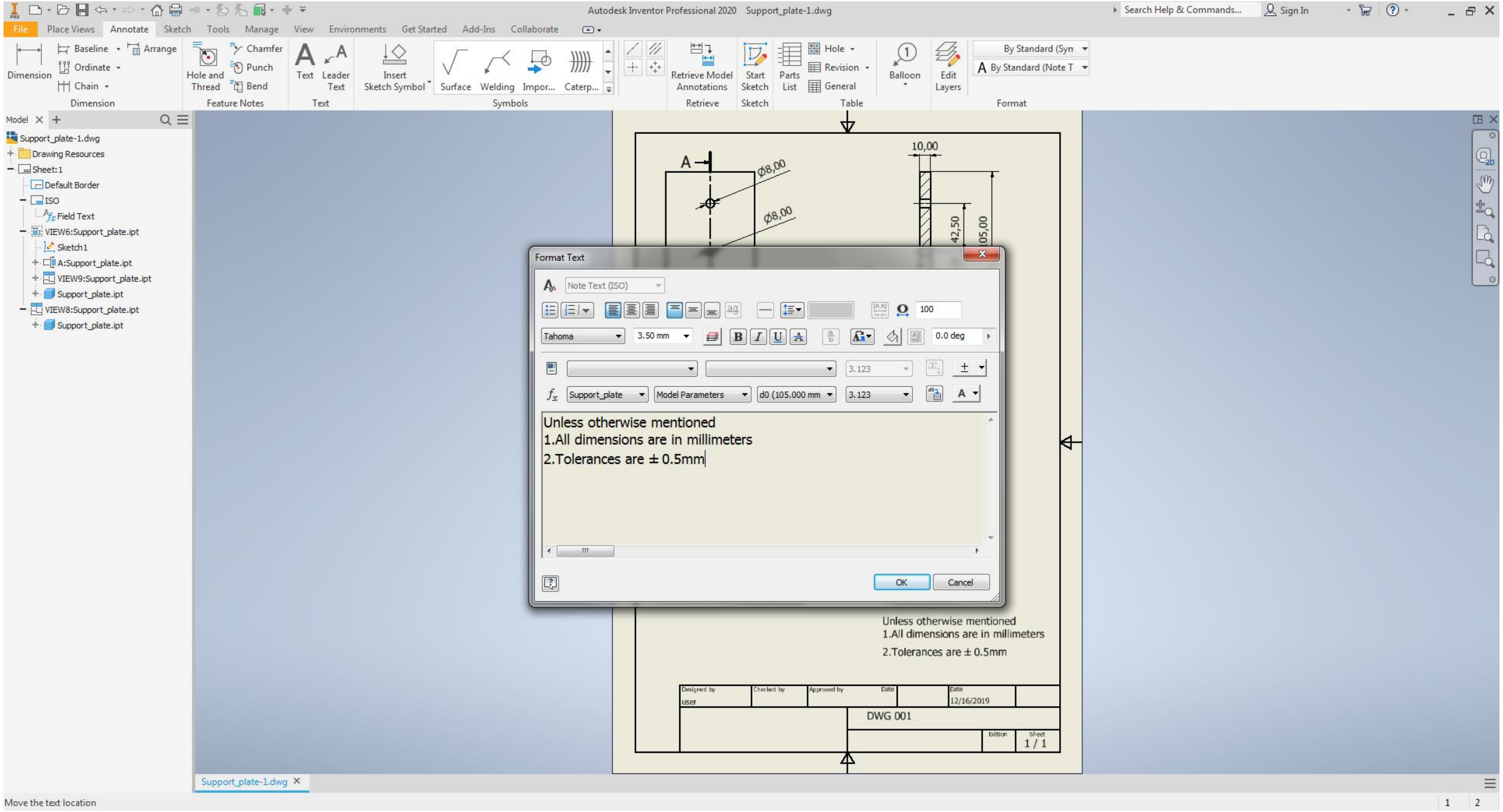
To add text in the drawing sheet
Click on “Text”



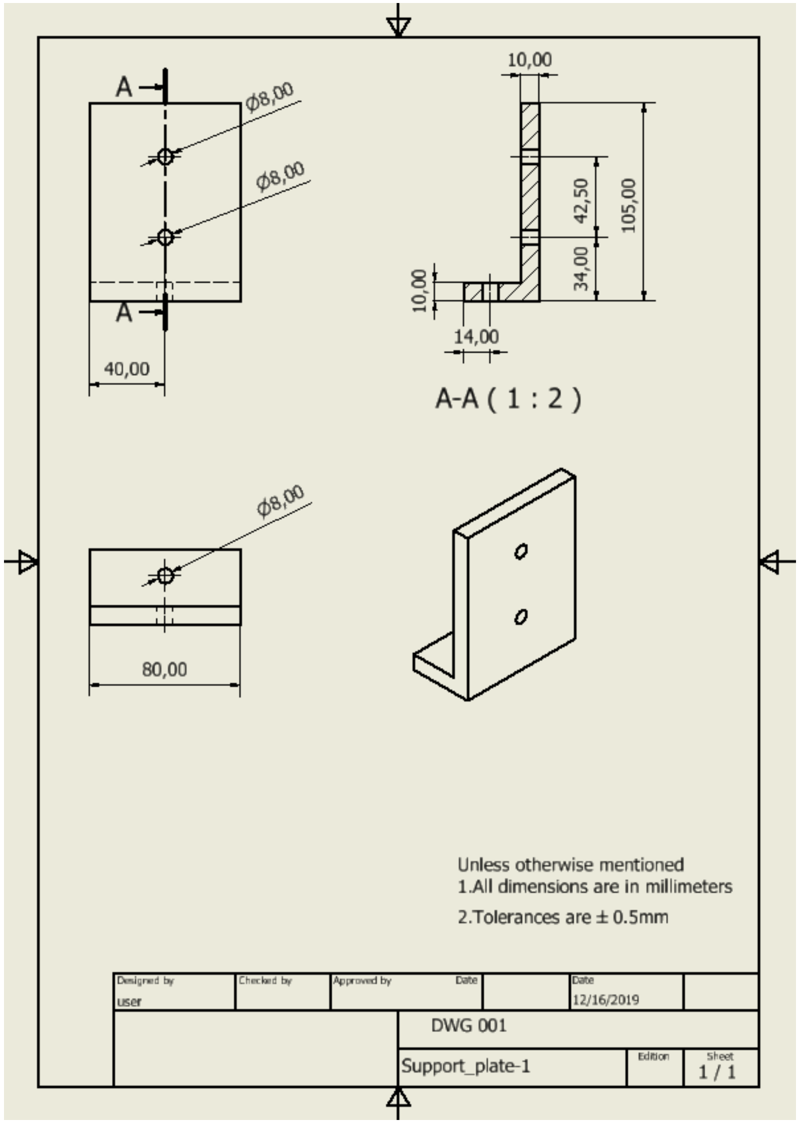
Select a location where you would like to place your text.
Type the text and click on OK



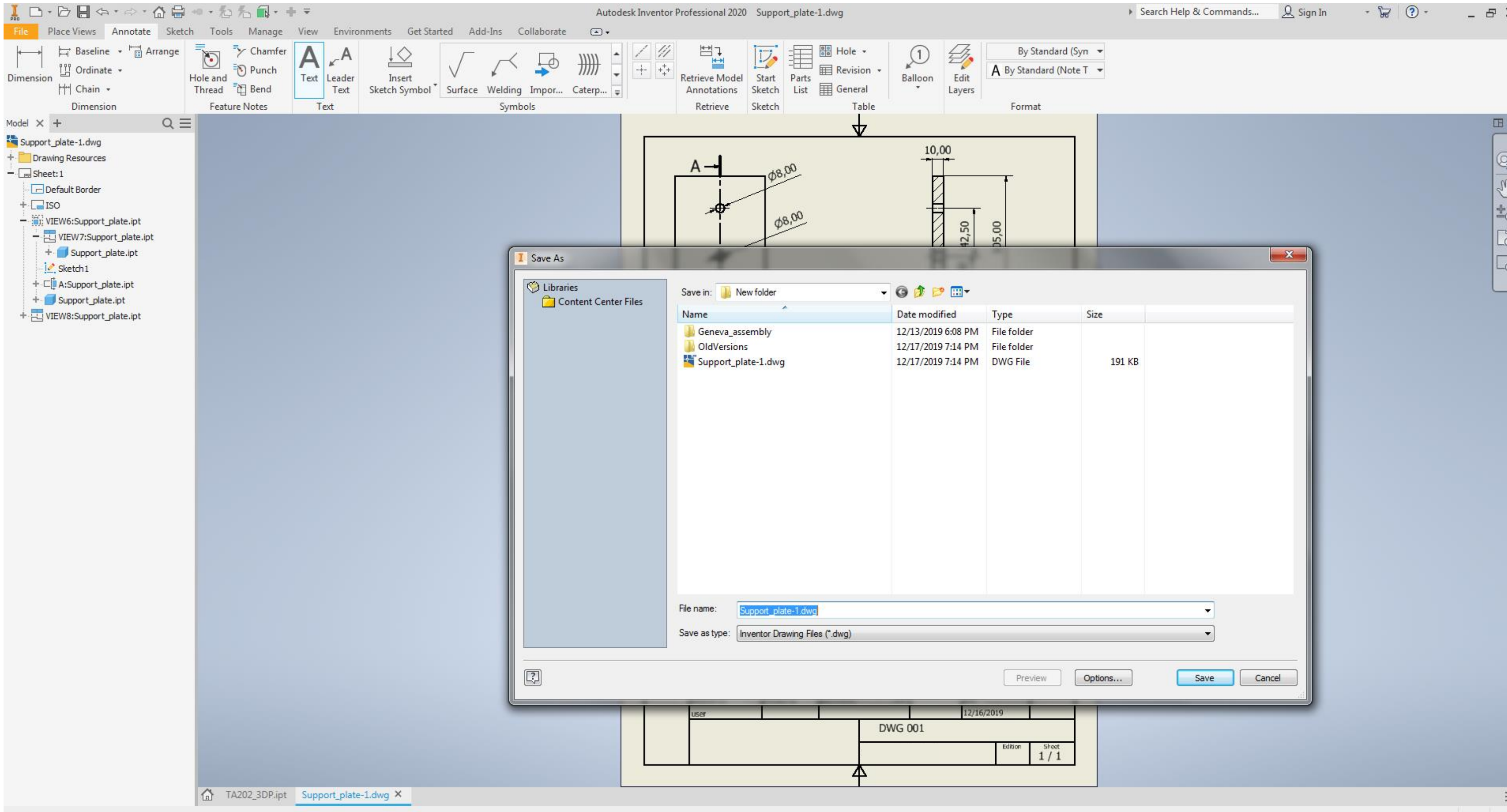
Following the aforementioned steps add the text as shown below



Your drawing should look like this



Click on File and Save your drawing



This is the end of this exercise. If you’ve reached this far, well done.

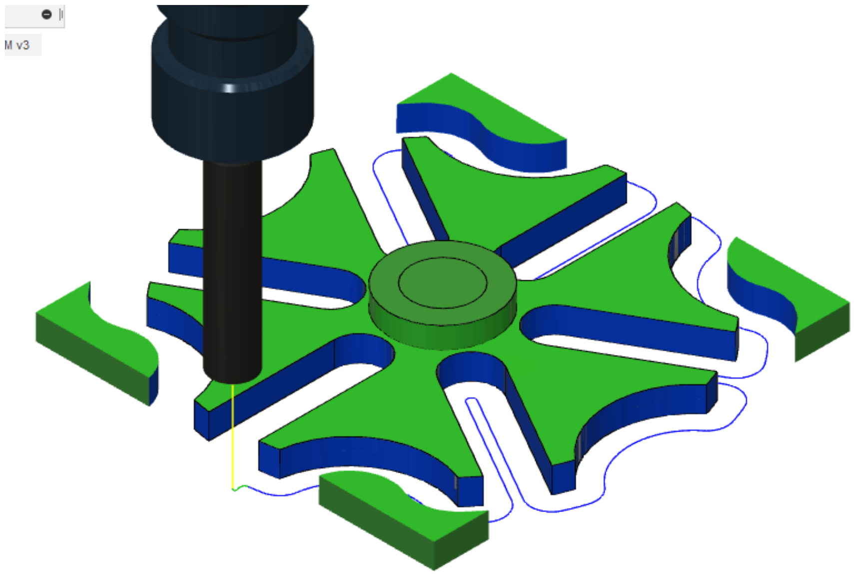
CAM Task A

Generate a G code for a part to be CNC machined (milled)

Contributed by:
Pankaj Deora (Y16), MadLab, IITK

CAM Exercise

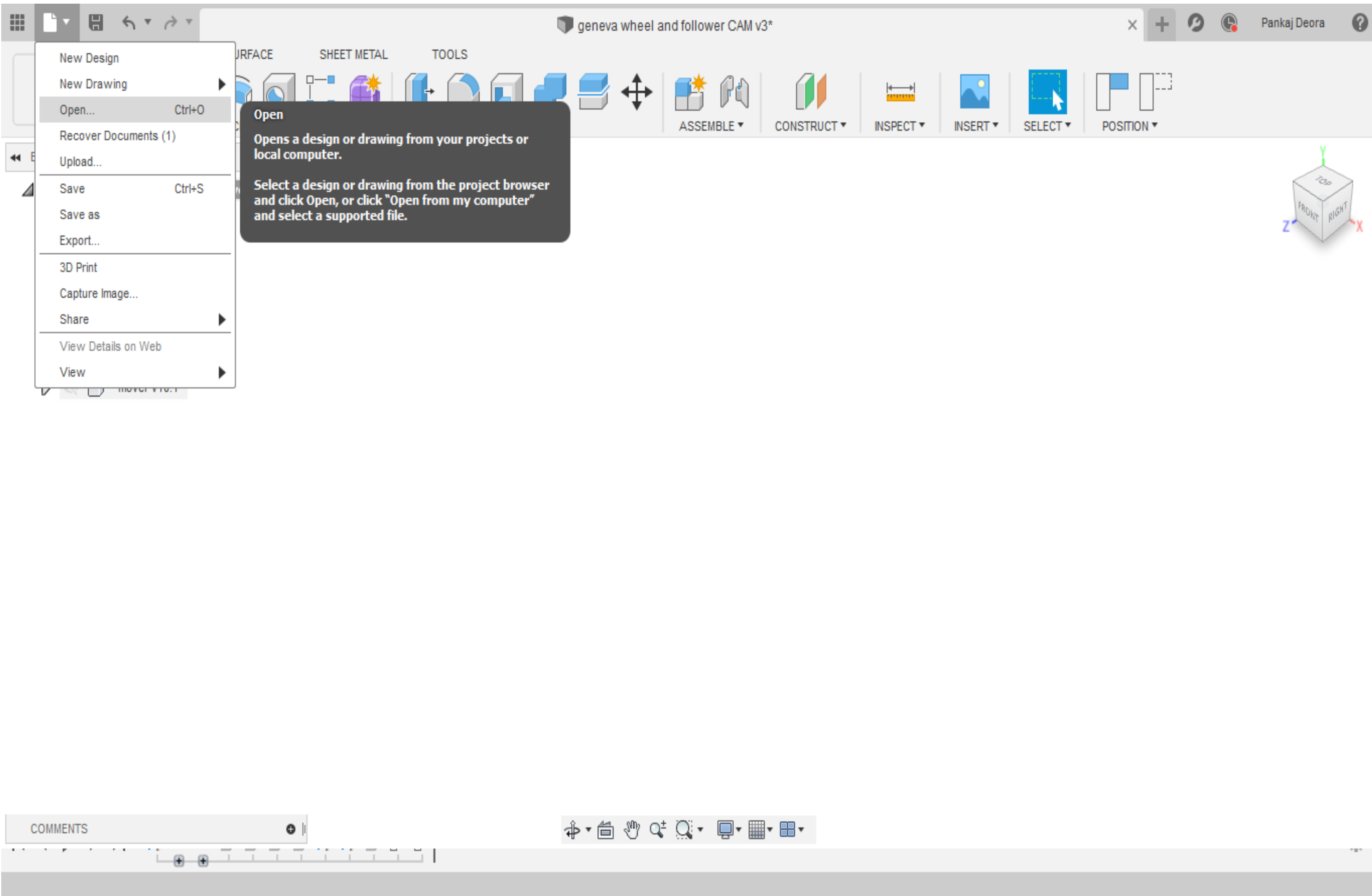
Objective: to learn how to make a computer aided manufacturing (machining) program using software. This program is otherwise known as a G code. The program can be directly loaded on to the machine to command it to make parts that you need and want. This example is representative of parts that you might need to make for your projects.



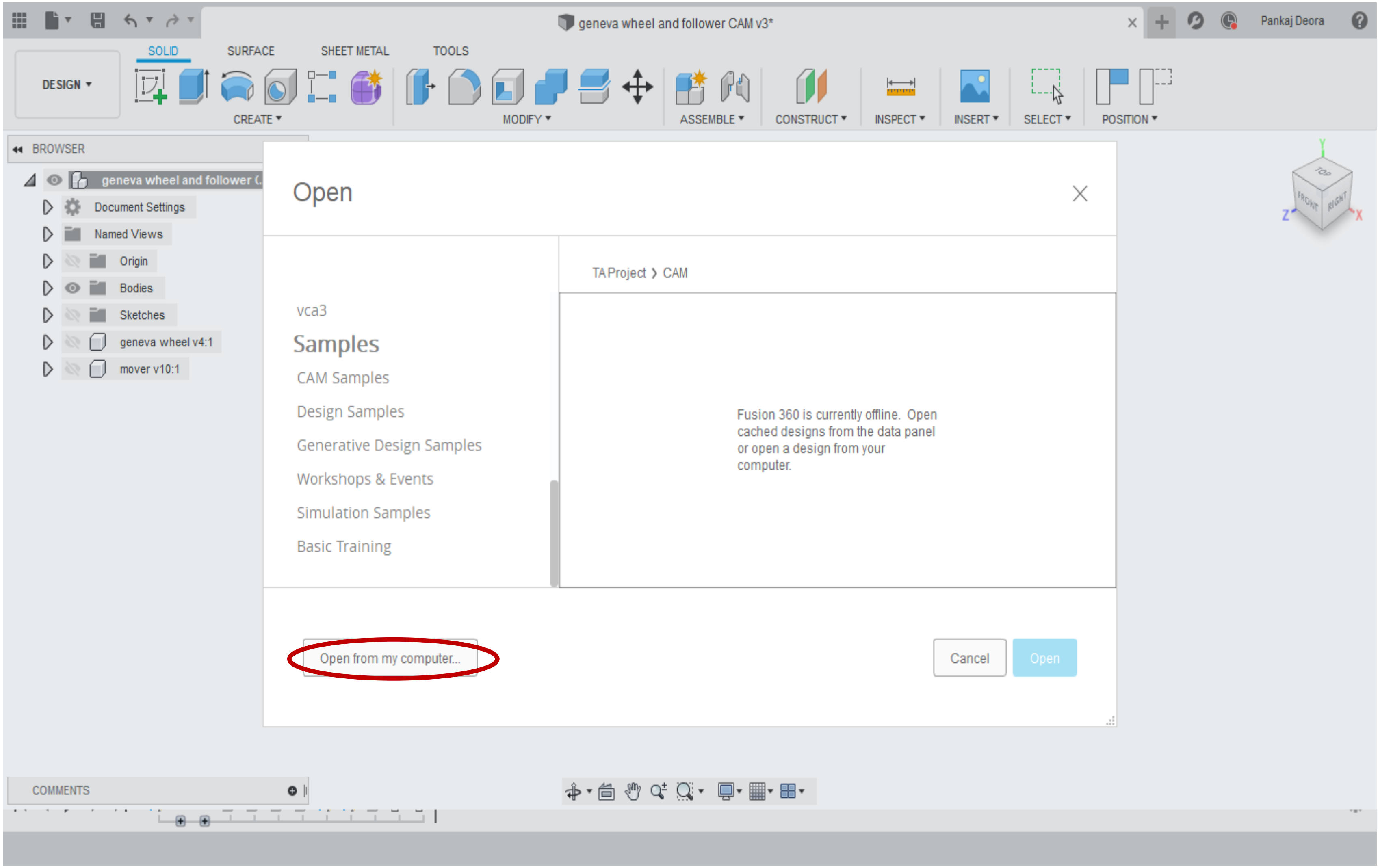
Preparatory steps

Load the software

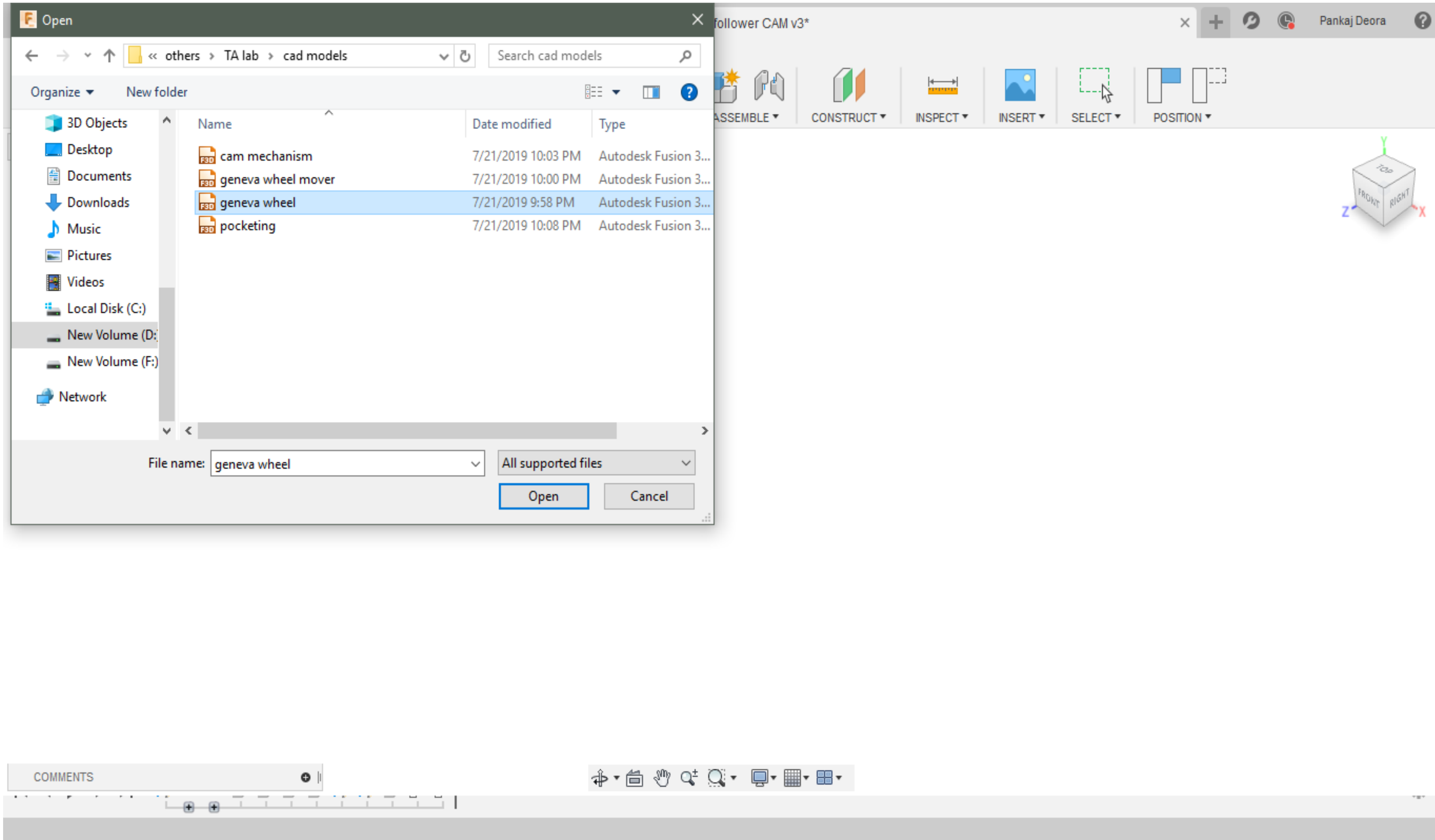
In general, to generate a CAM program, you need a CAD file. The CAD file has features and dimensions of the part to be made. In this exercise, the CAD file is provided to you. To open the CAD file for the Geneva Wheel, click on “open” under “File ” tab.



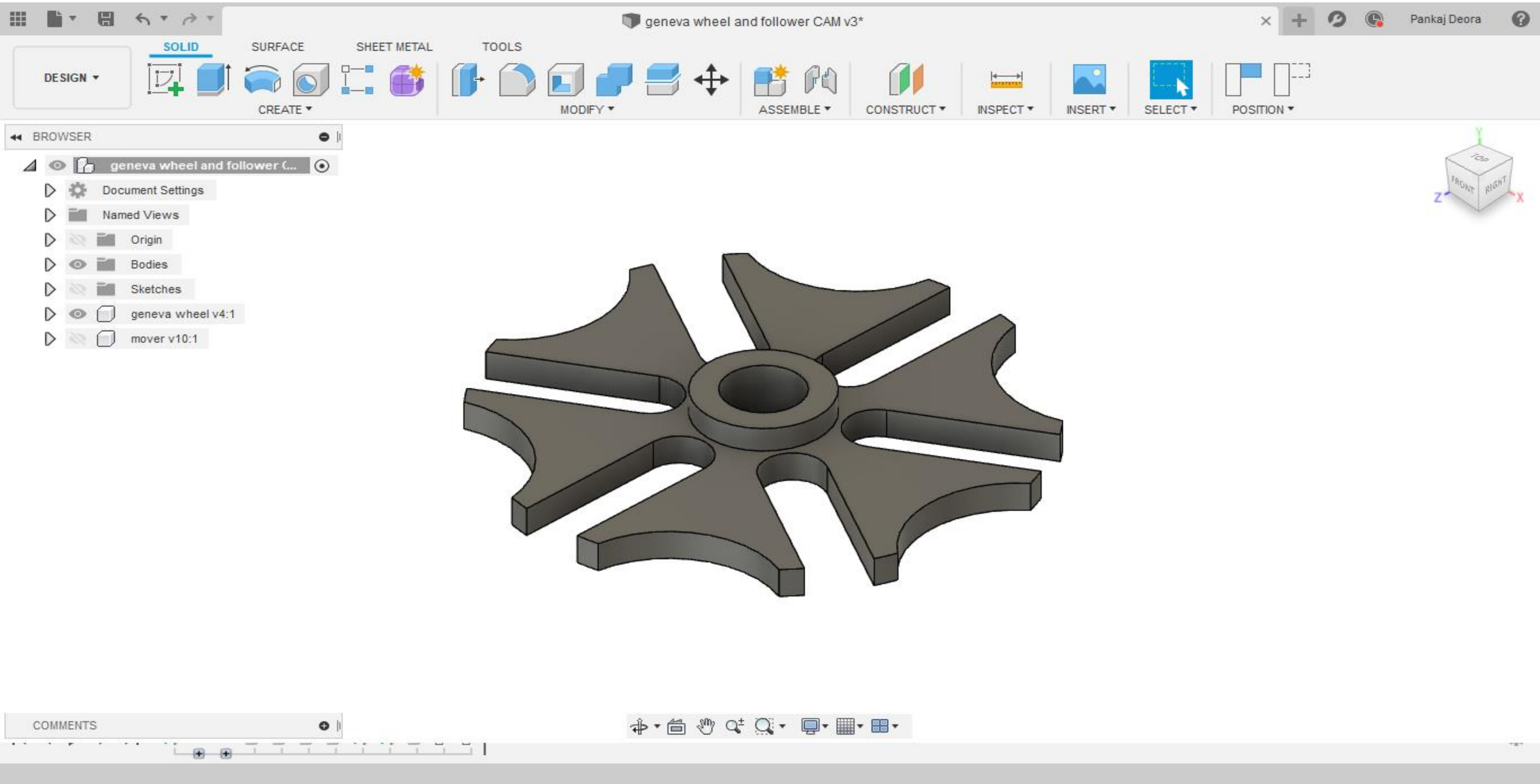
Click on the ”Open from my computer”



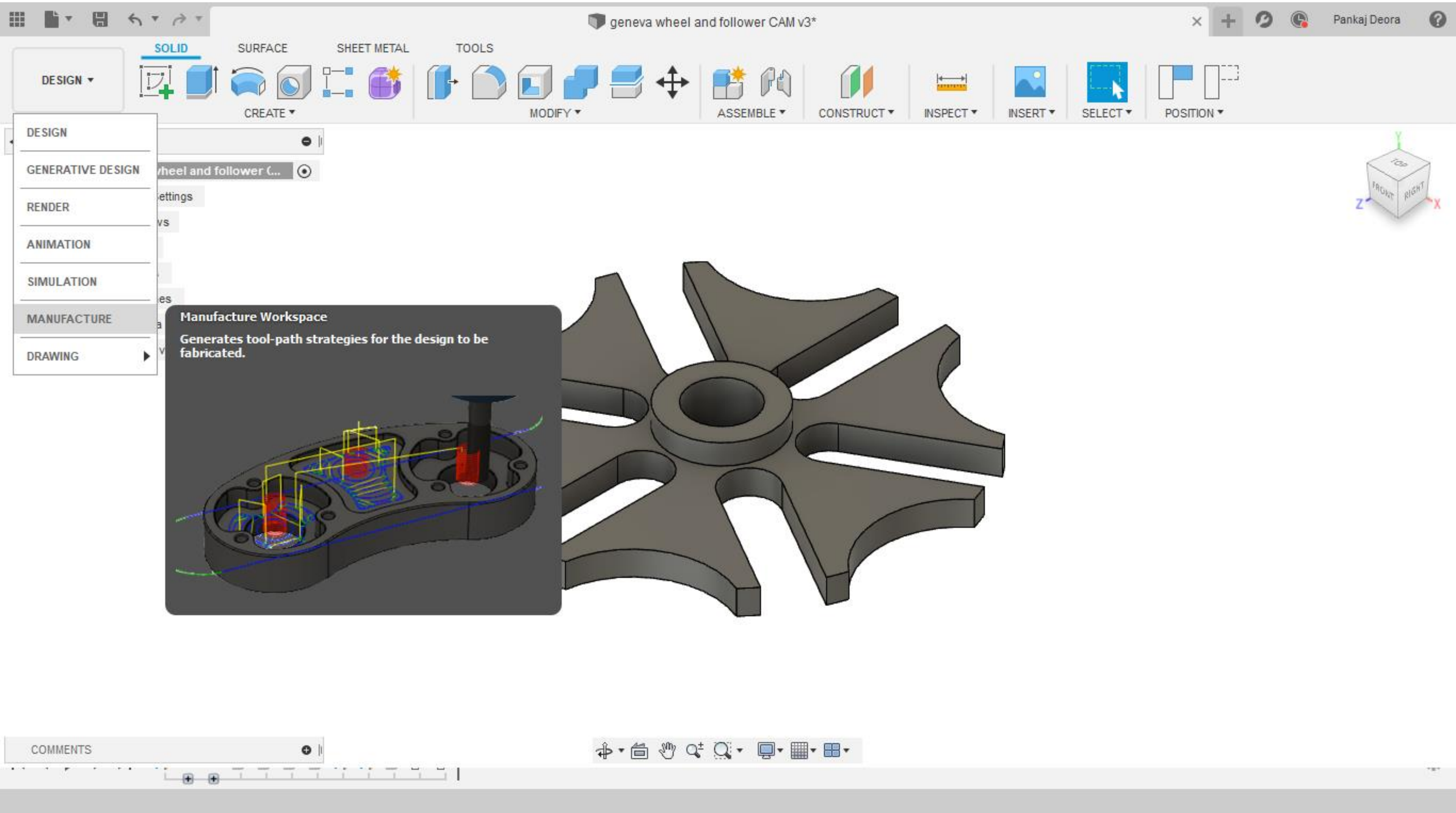
Locate the file and click “Open”



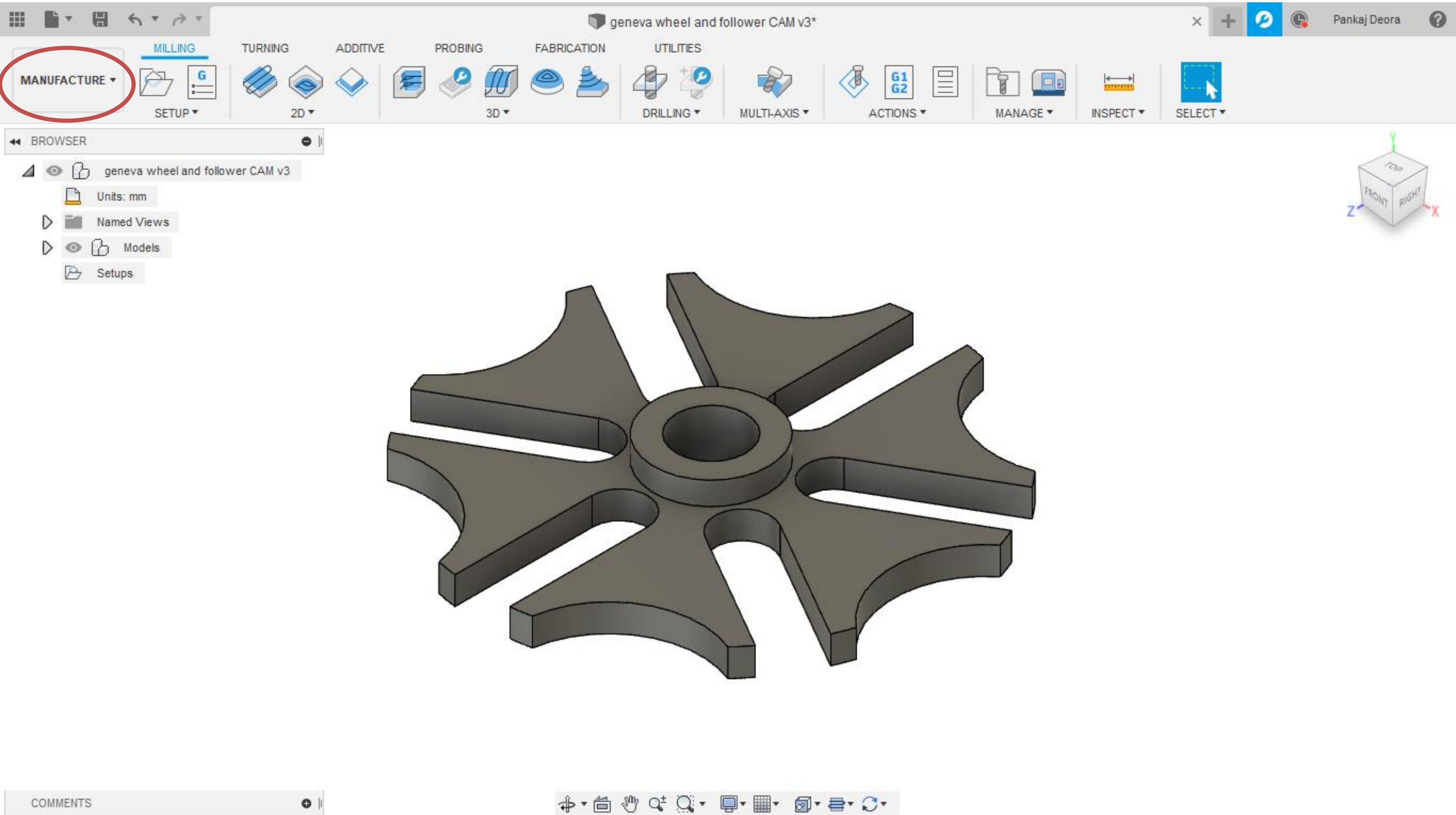
The window should look like this



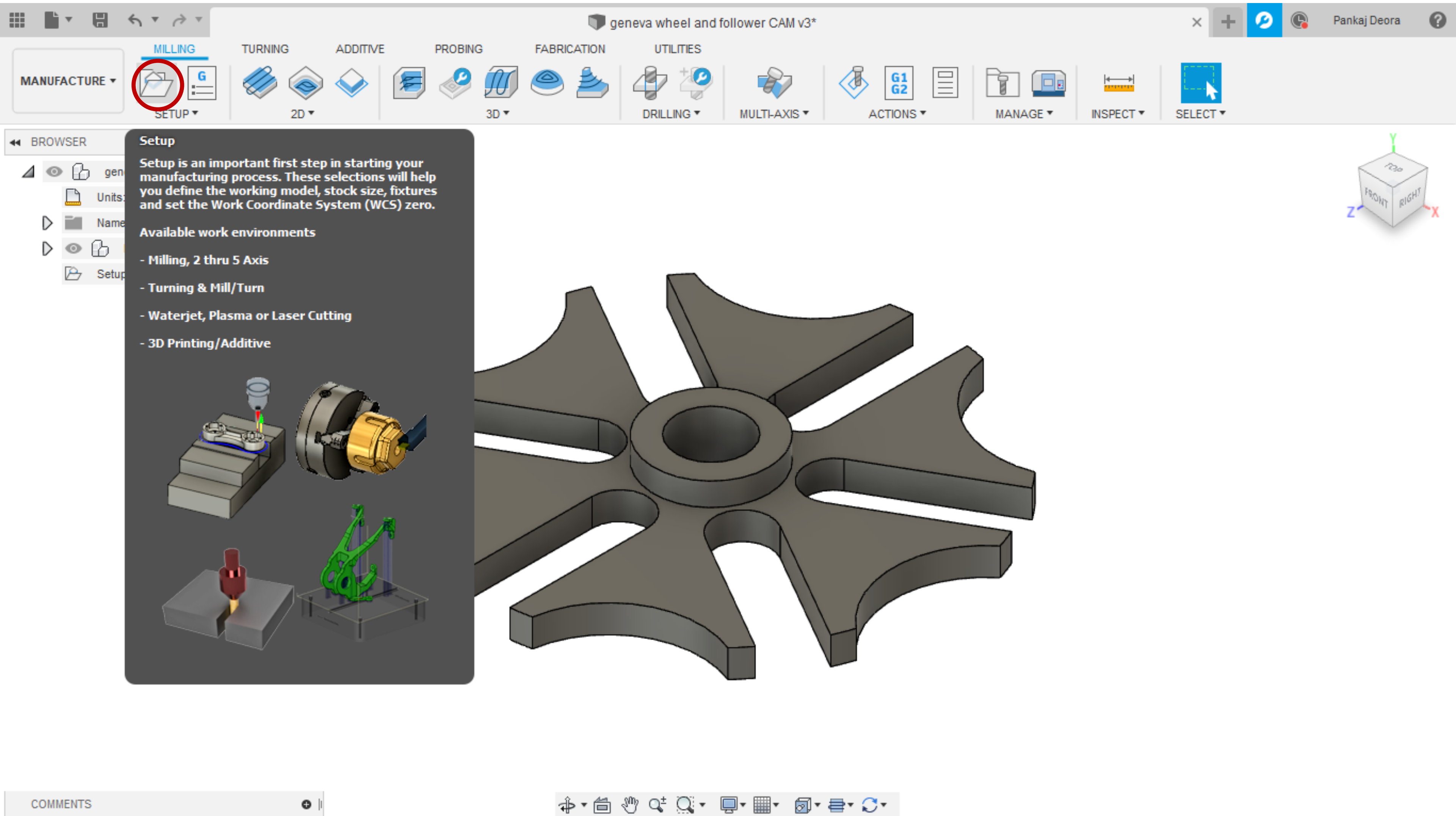
To start with the CAM exercise, click on “Manufacture” from the drop down as shown



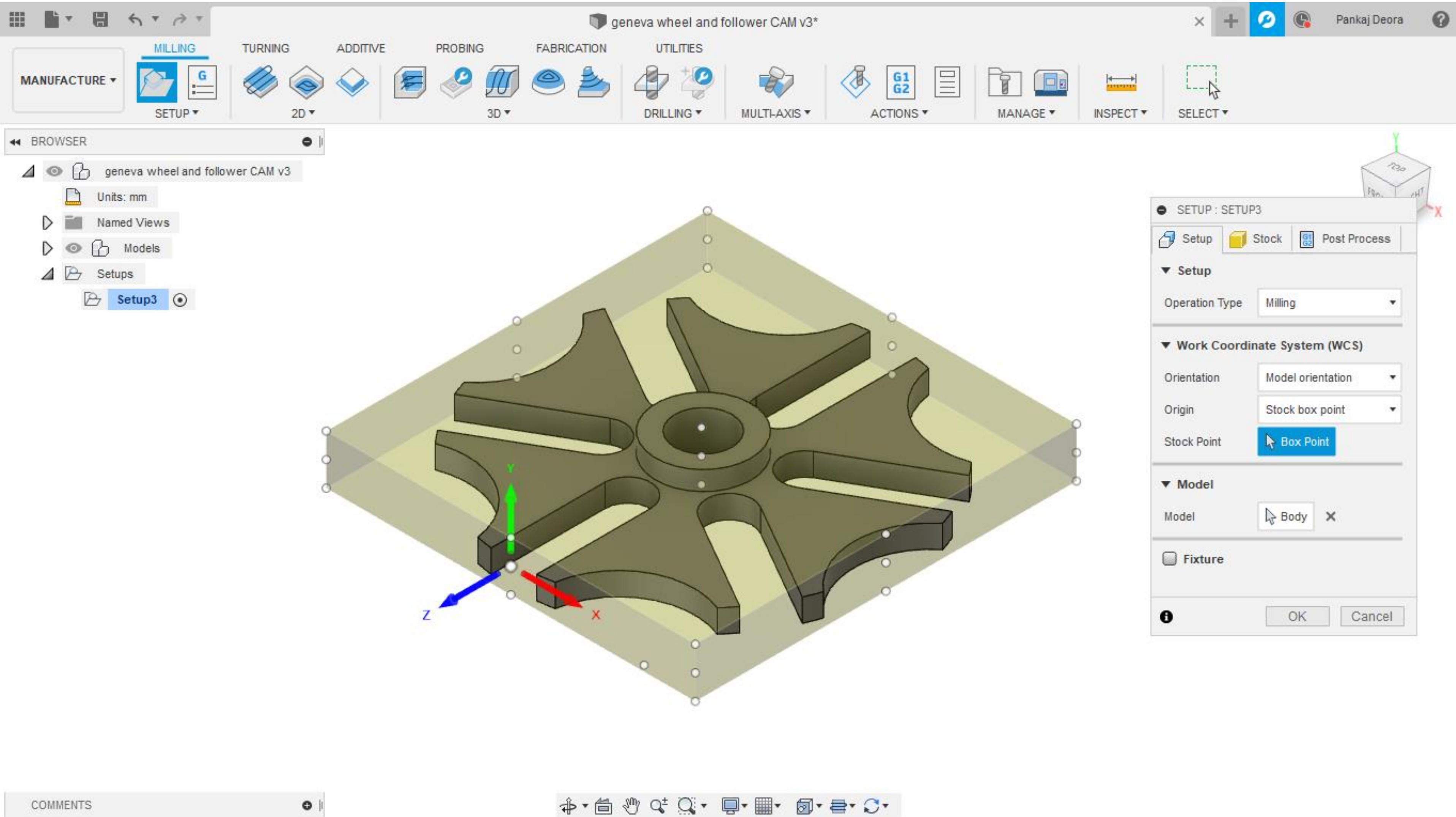
Make sure that “Manufacture ” tab is selected



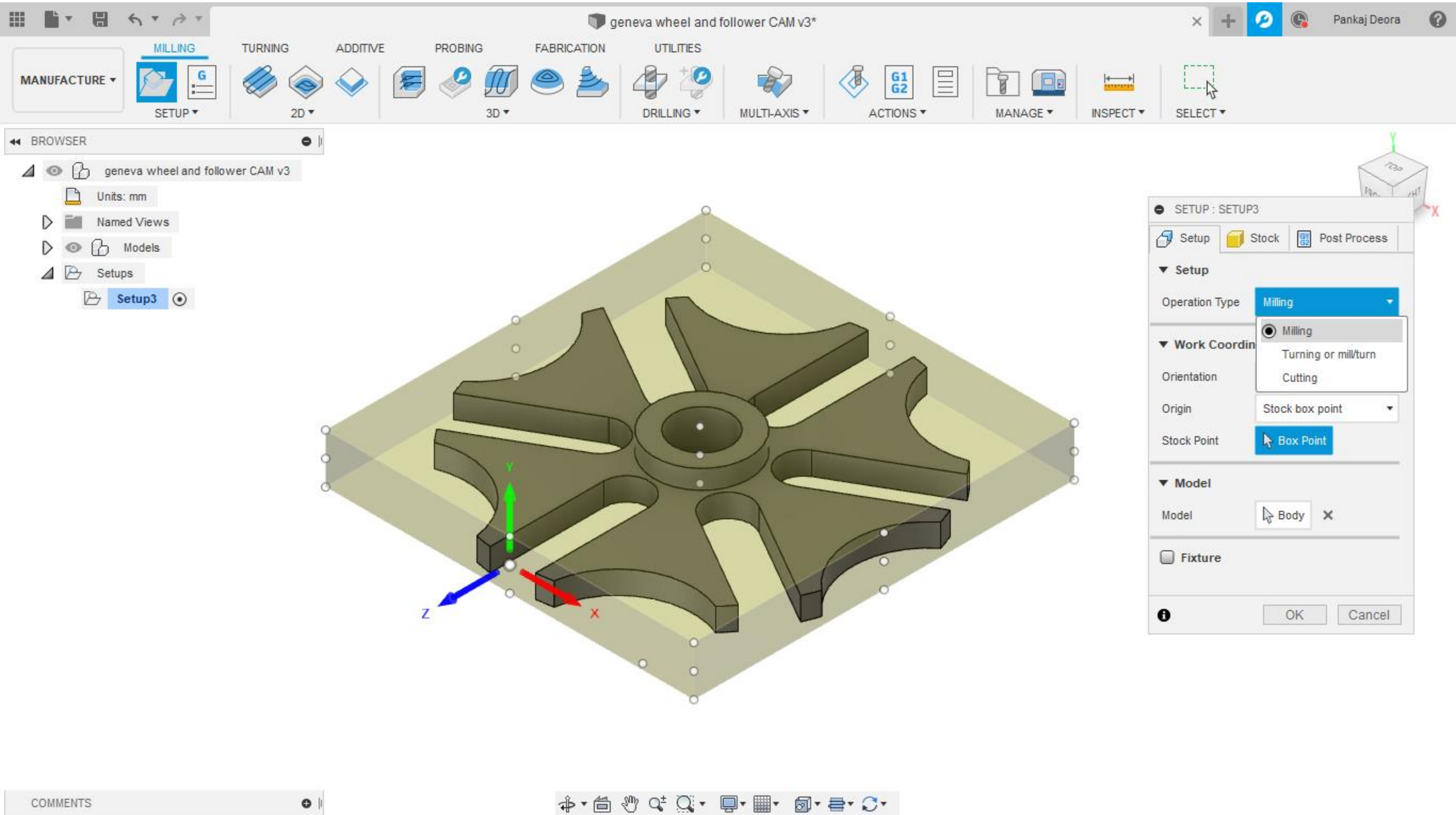
First we will setup the environment. To start with, click on the “setup” option



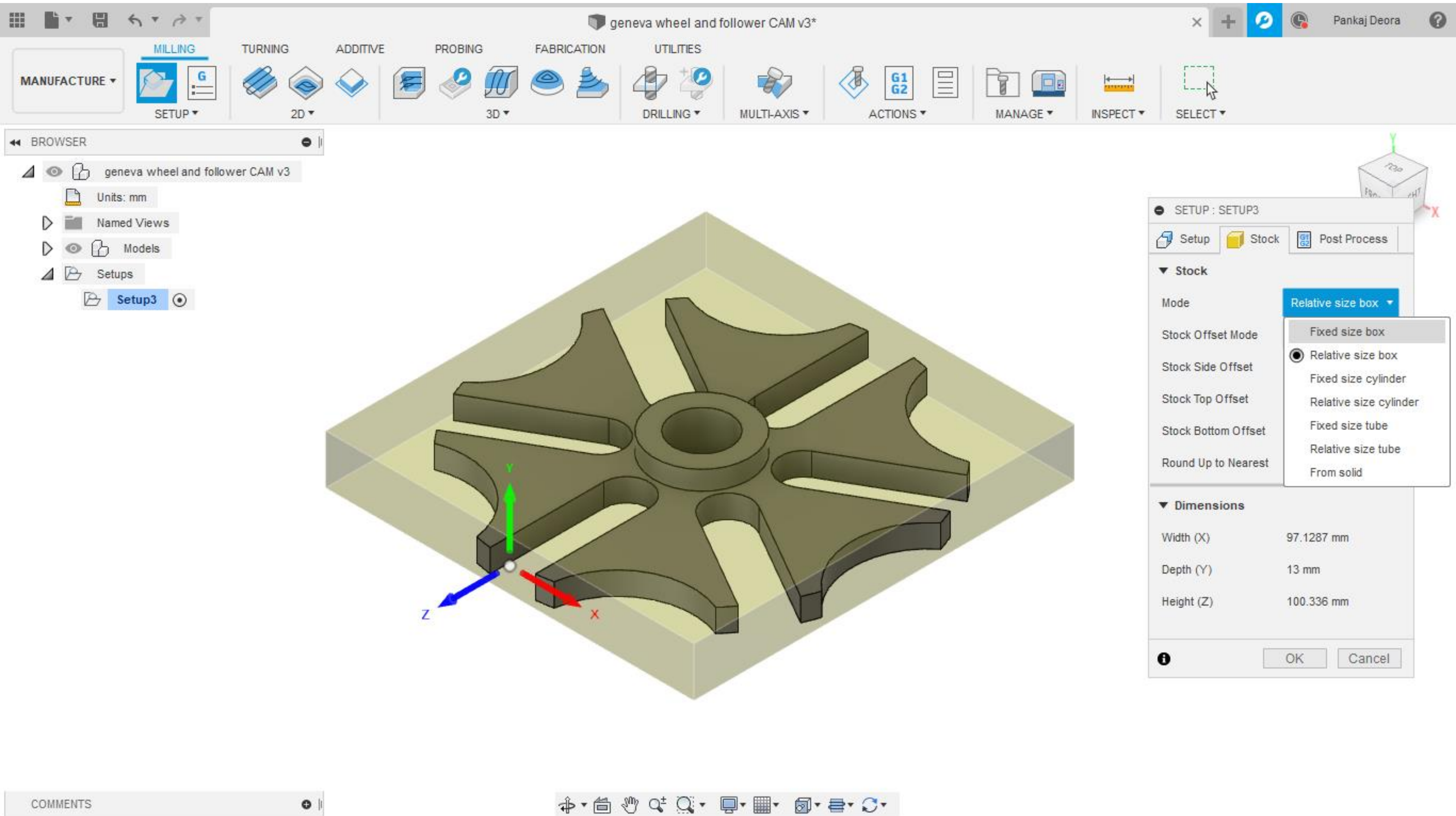
A box will appear on the right. This is where we define the stock (raw material size) for our model



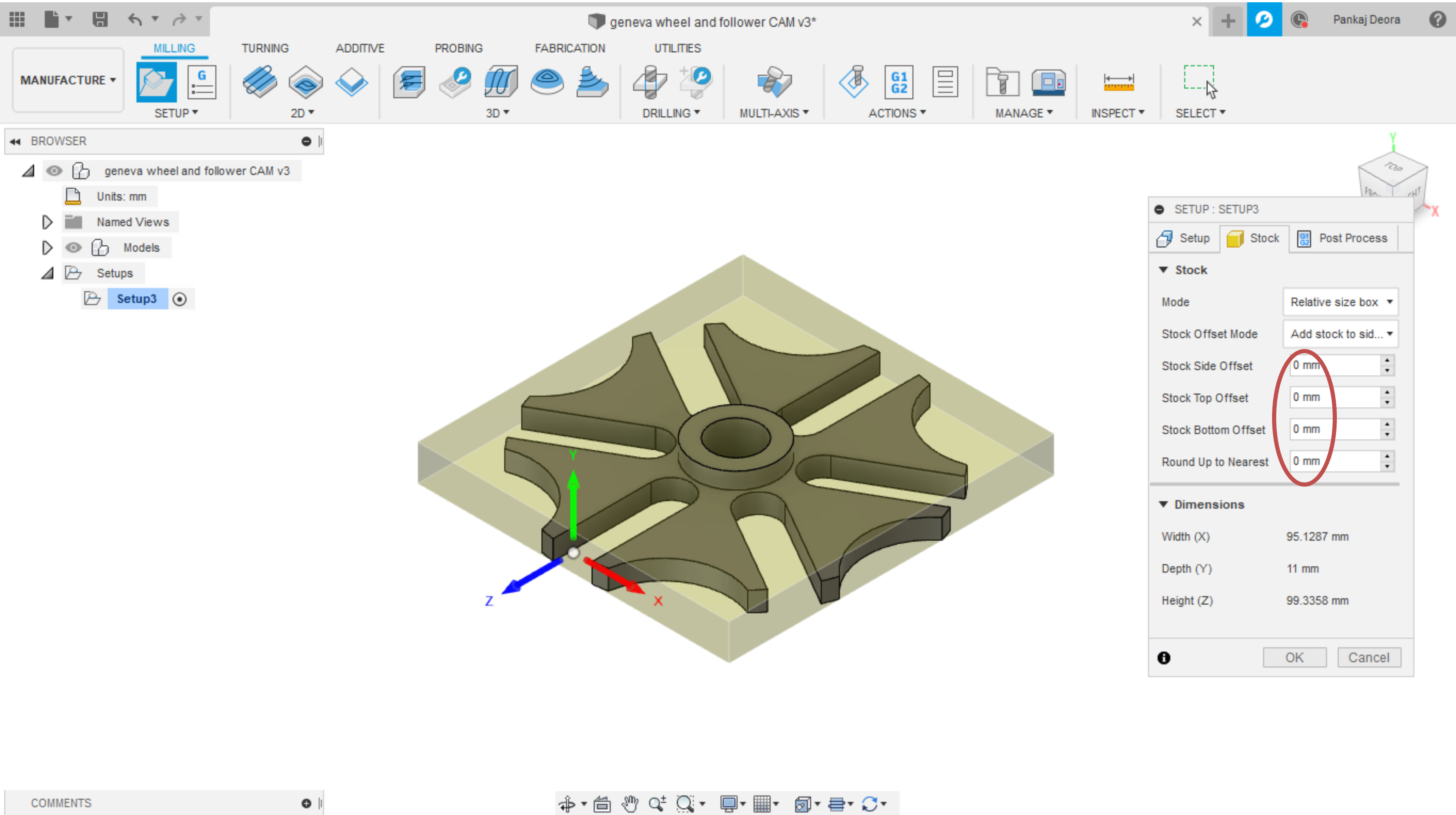
We are going to perform milling operation. In the “operation type ”, make sure that “milling ” option is selected.



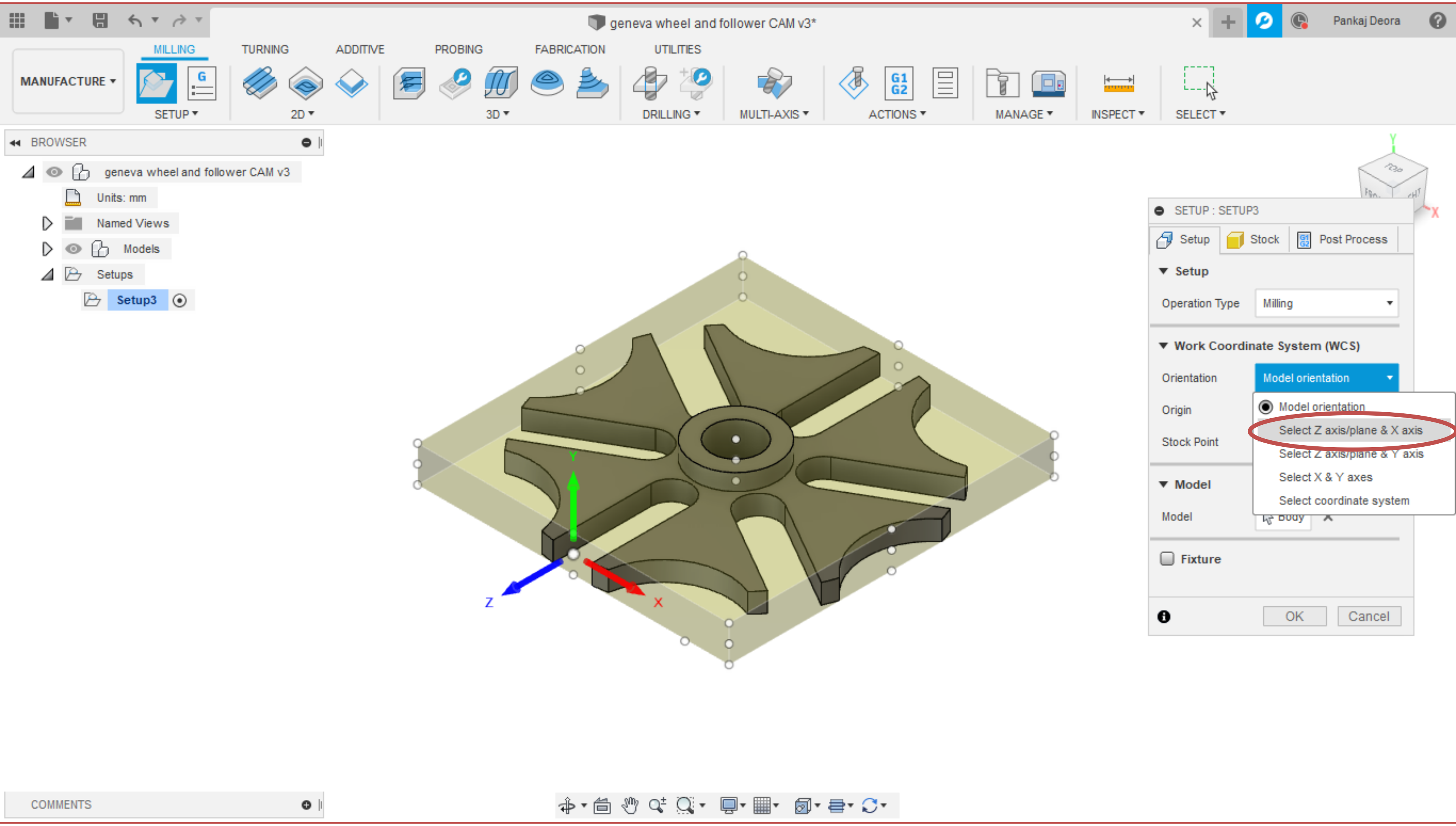
Under the “stock” tab, select “Relative size box” under “mode” tab



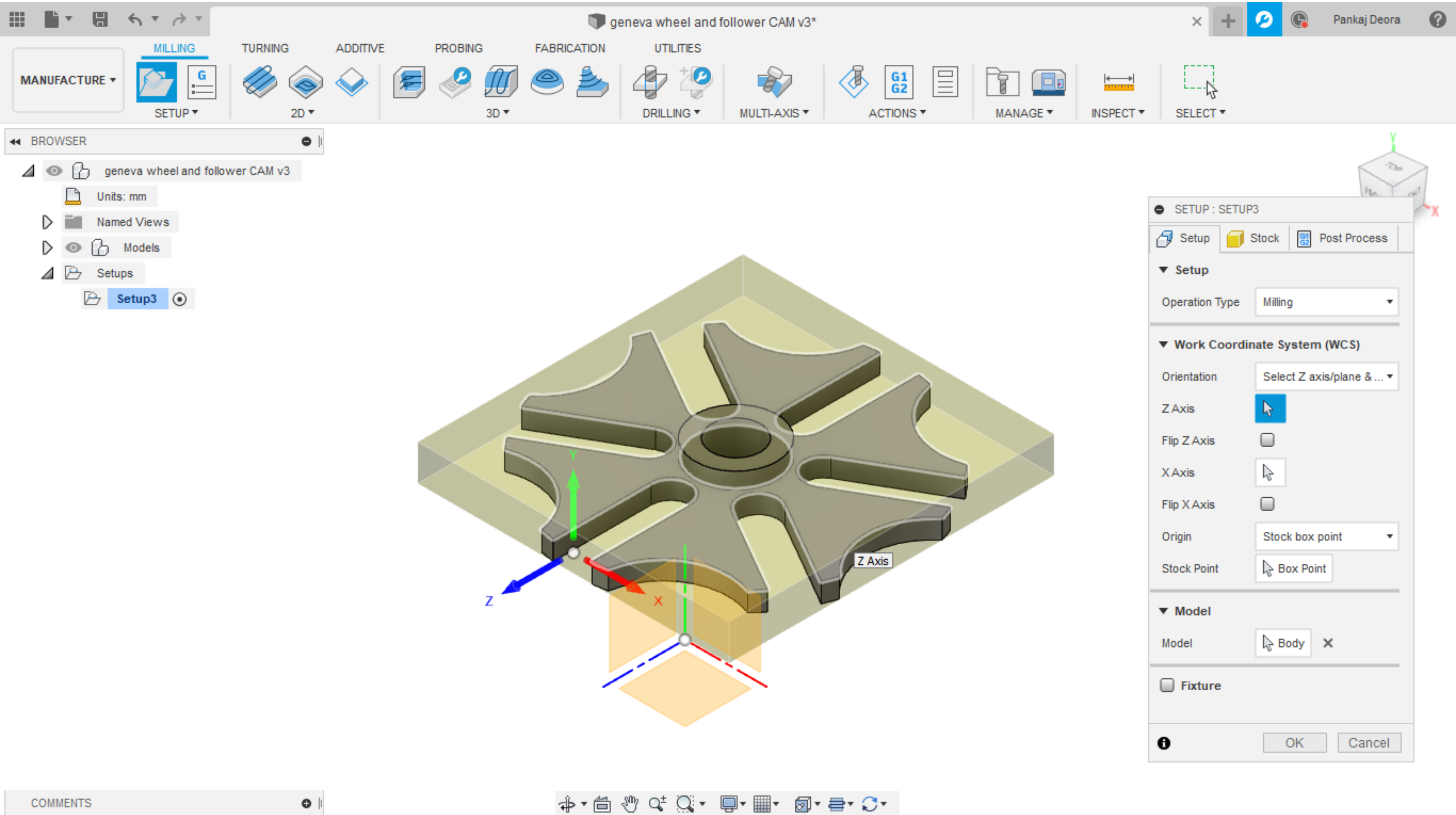
Make all the entries shown in the circle below ‘zero’ because we don’t need any extra stock for our work



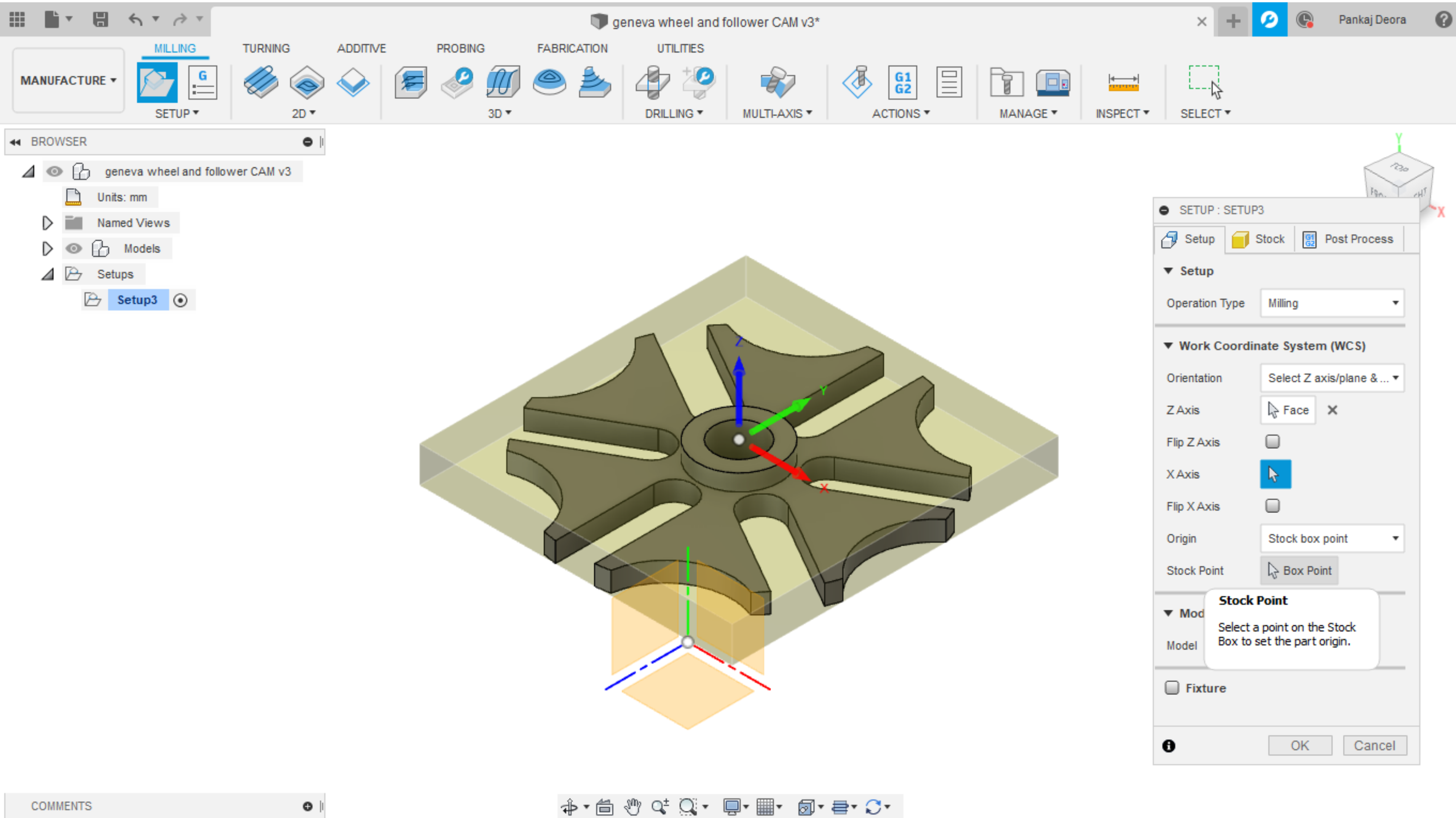
Again go to "setup" and select the option shown under "orientation" option



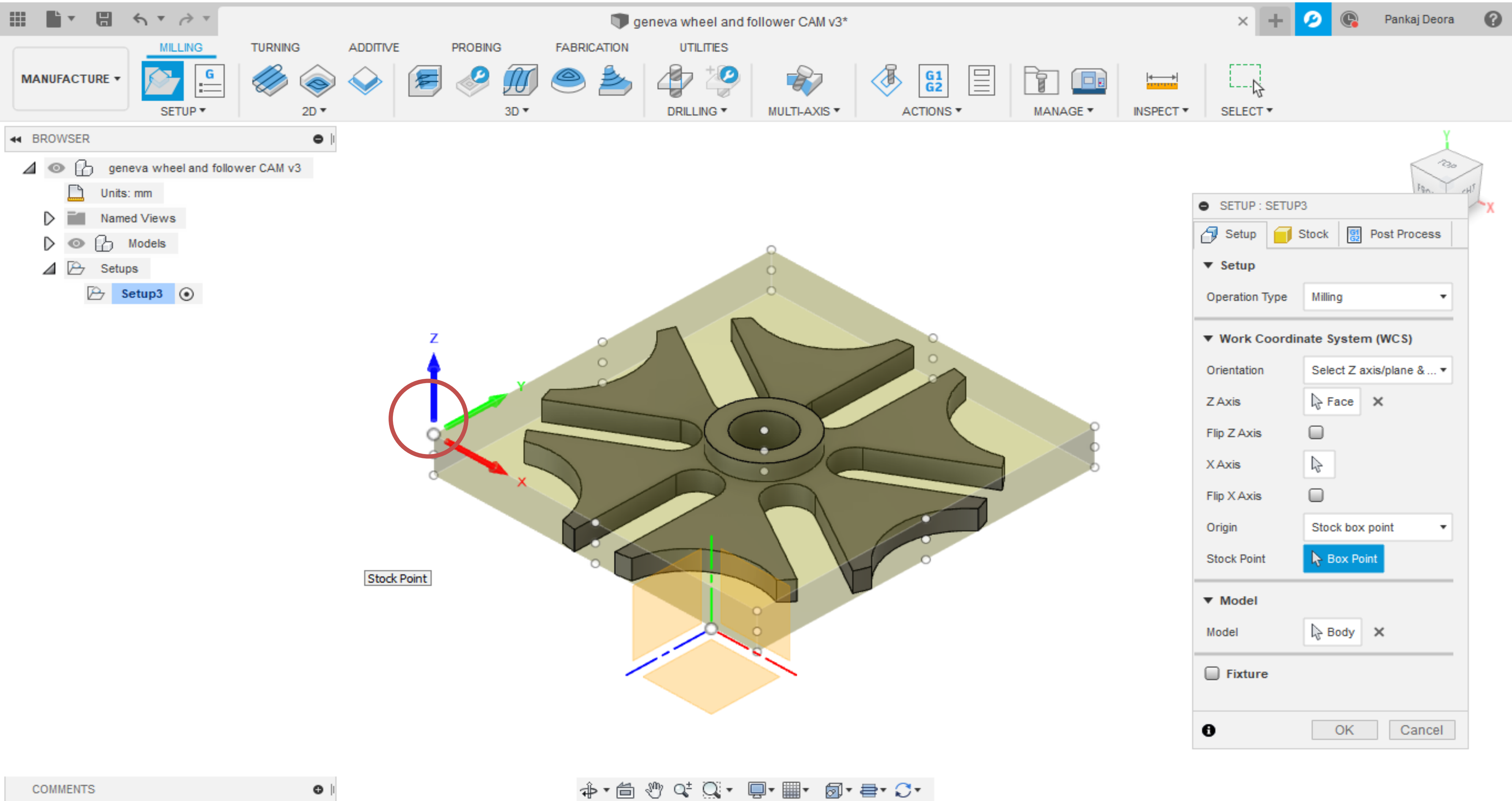
Then select the top surface of the Geneva wheel by clicking on it



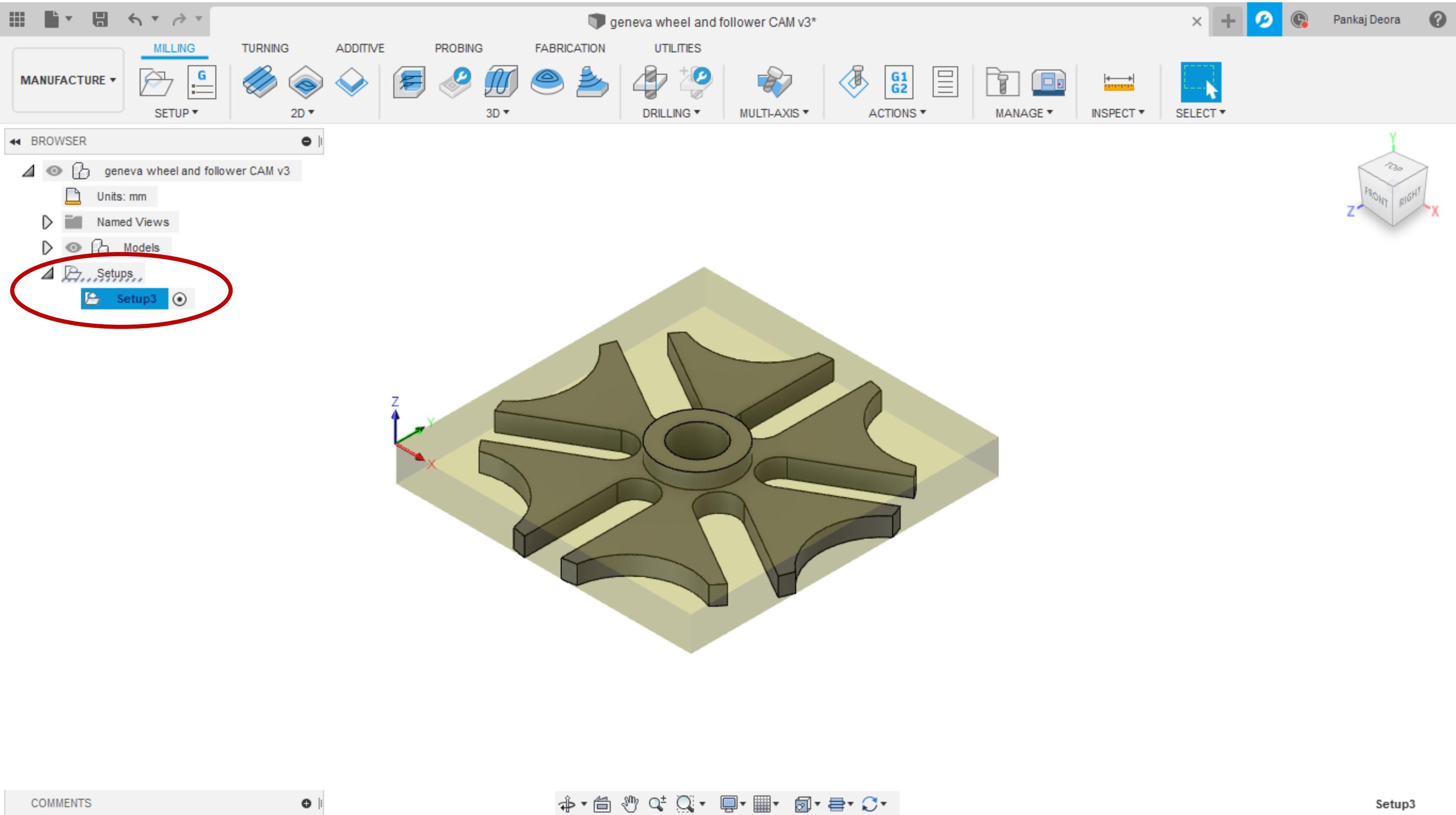
Now we have to define the work coordinate system (WCS) for the work-piece. For which, click on the "Box Point" option



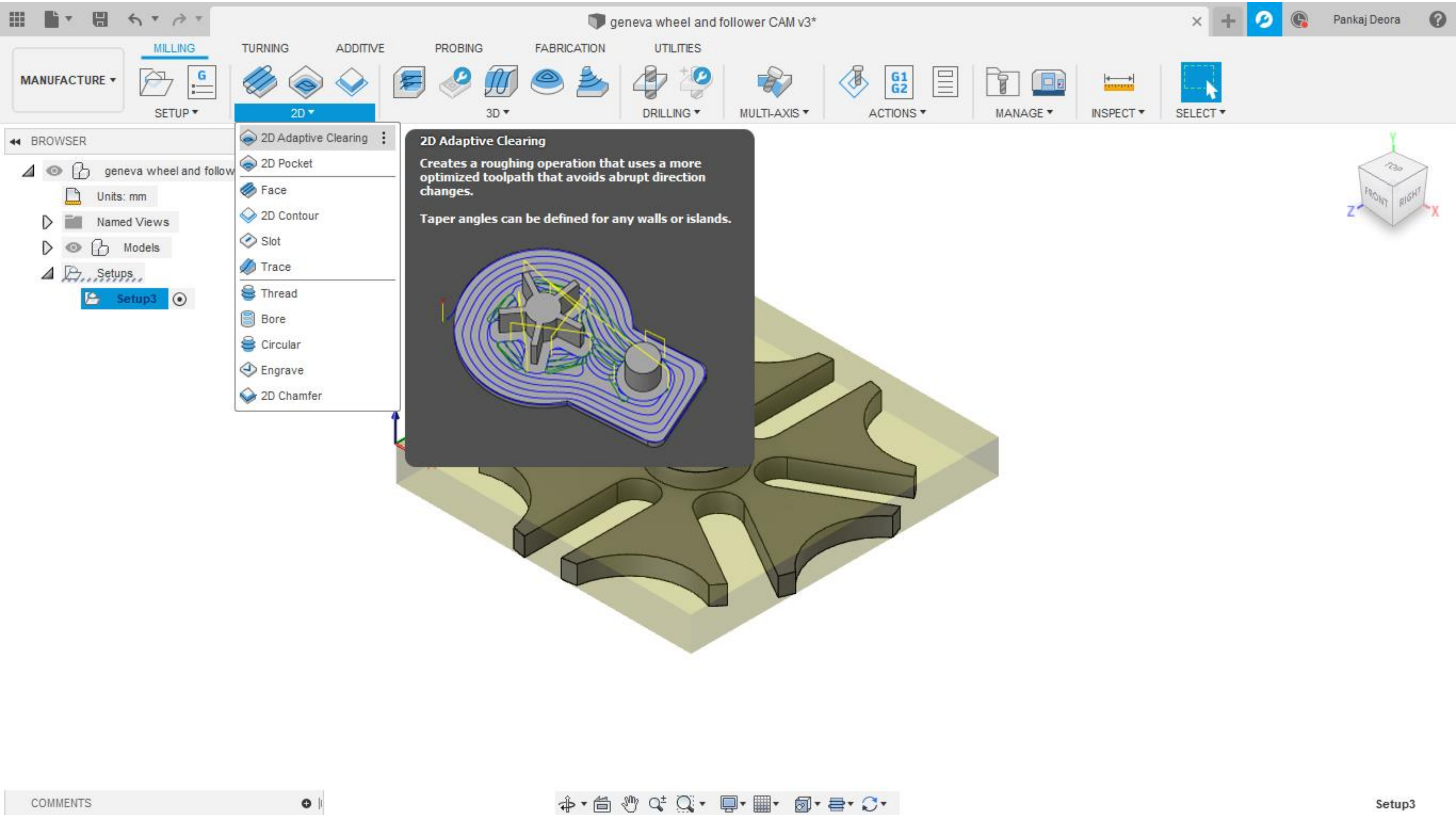
Click on the dot as shown. This will be the WCS and the G Code generated will be defined relative to this coordinate system now. Ensure that WCS orientation as shown below. If yes, Click “OK”.



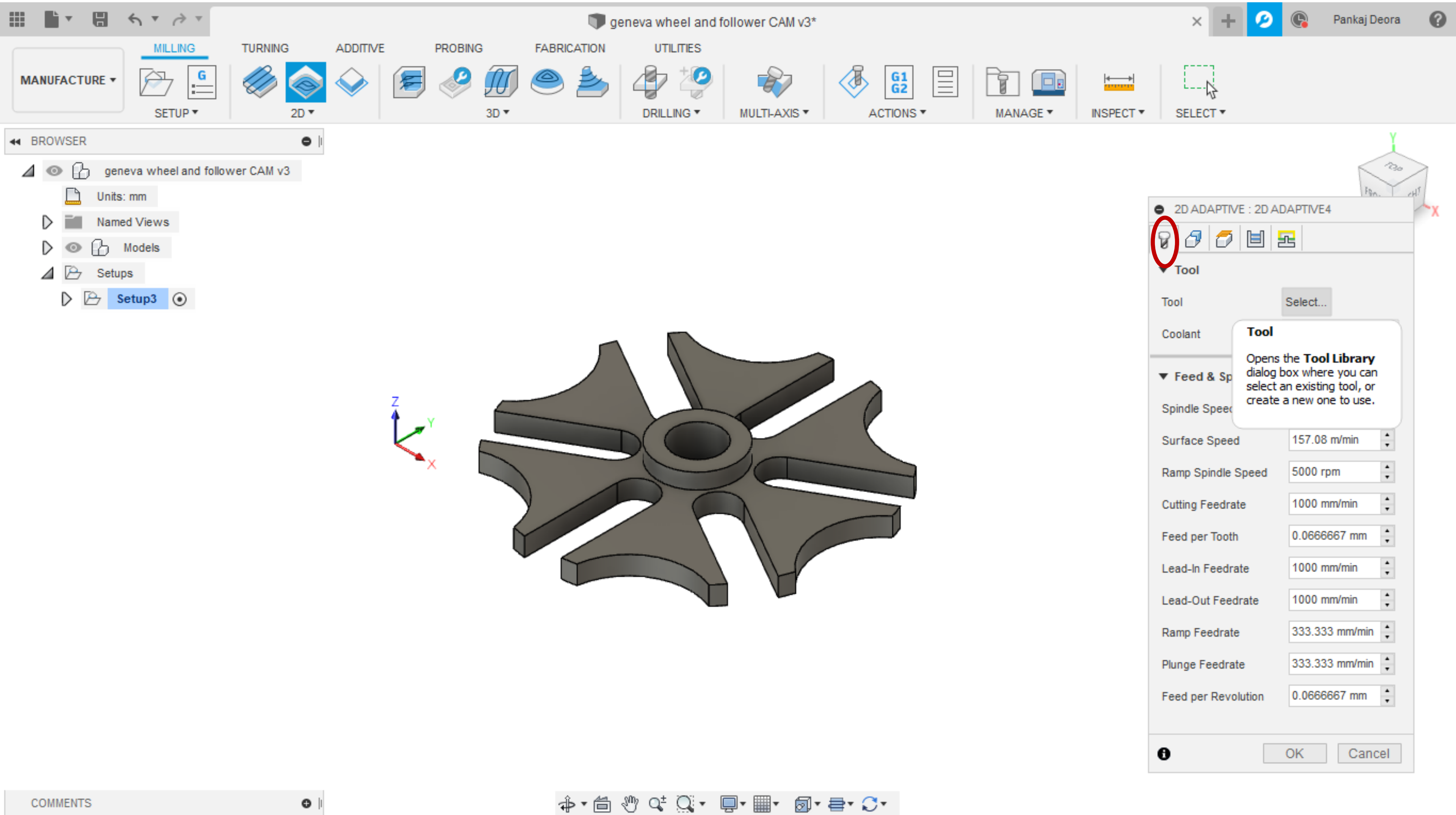
You can now see that your setup is ready on the left side of the window. Now we will cut the stock to manufacture the Geneva



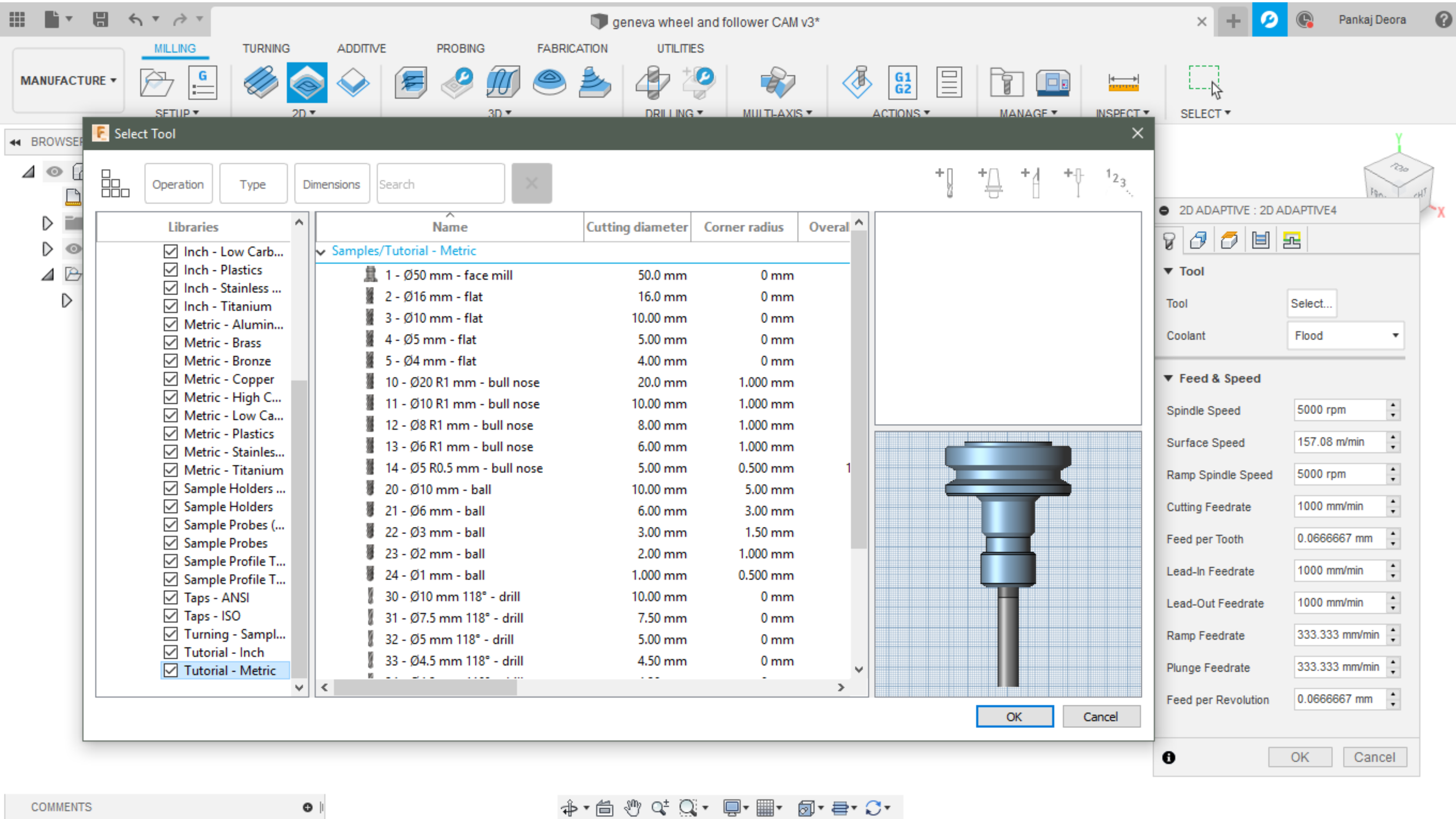
Click on the “2D Adaptive clearing” under “2D” drop down. You can read about 2D adaptive clearing by positioning the mouse cursor on it for a while



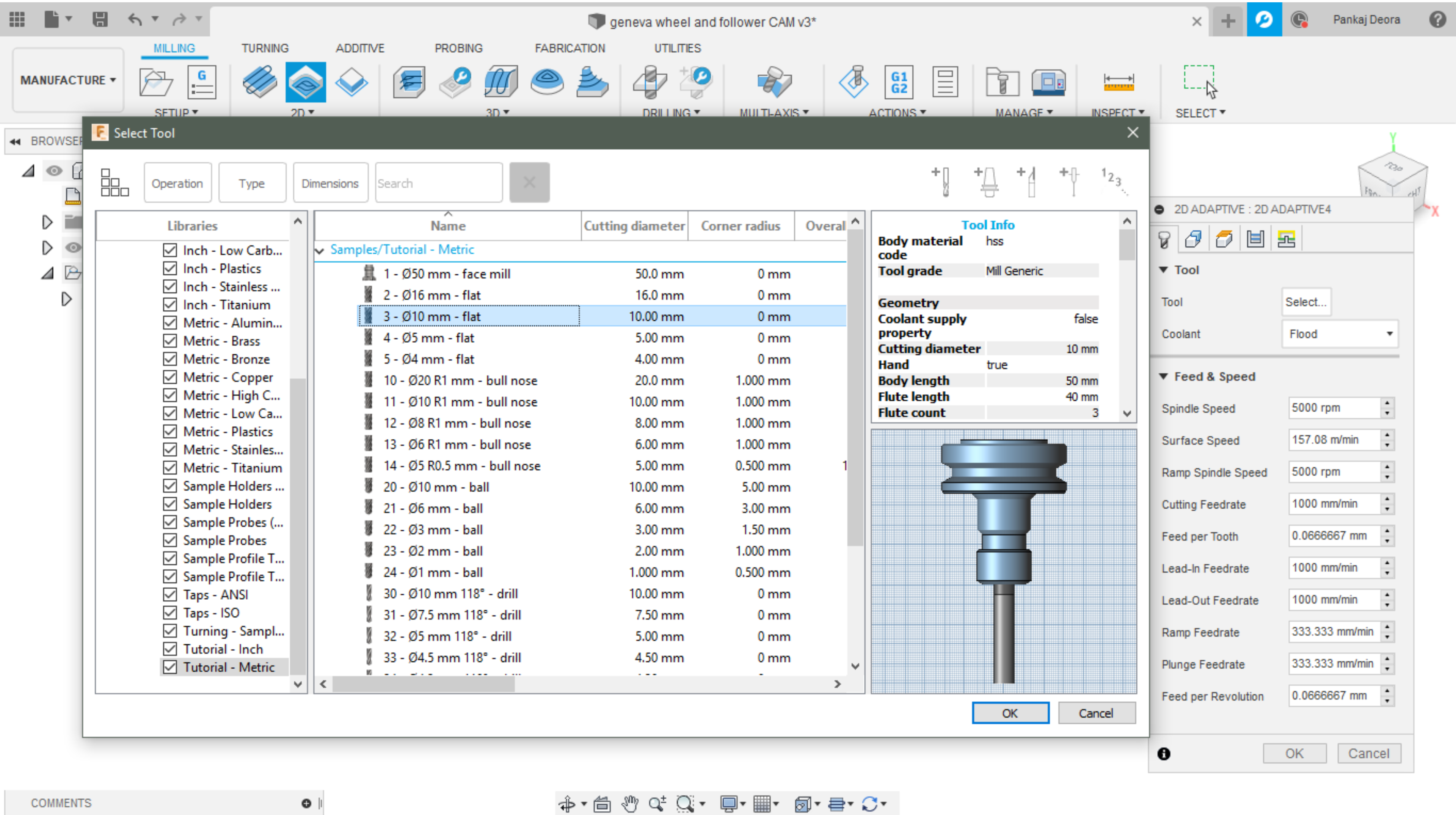
A box will appear. Click on “select” as shown. This is where we select the tool for our job



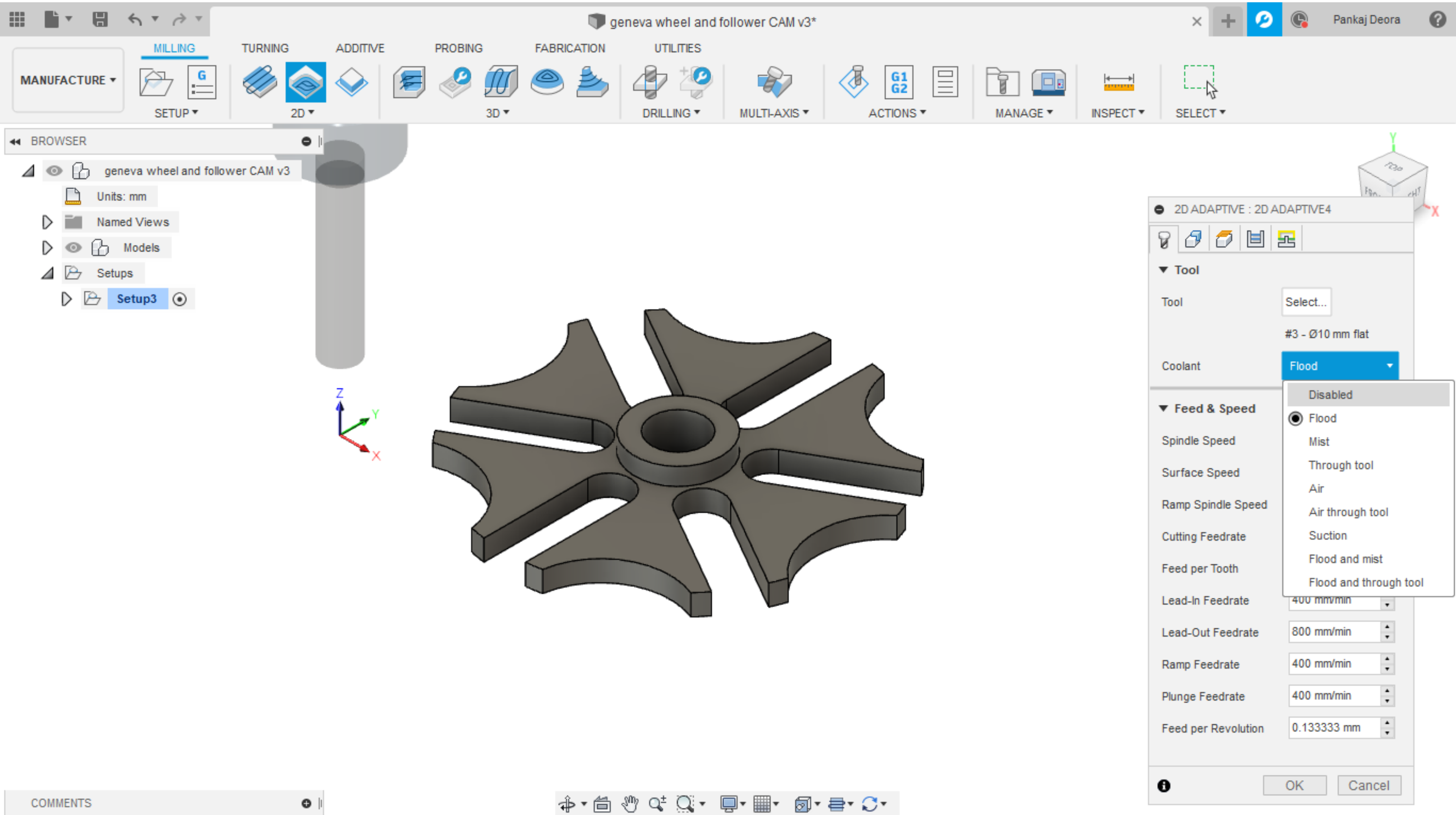
In the new window, scroll down to the end, and click on “Tutorial - Metric”. It will show all the tools that we can select for the milling operation



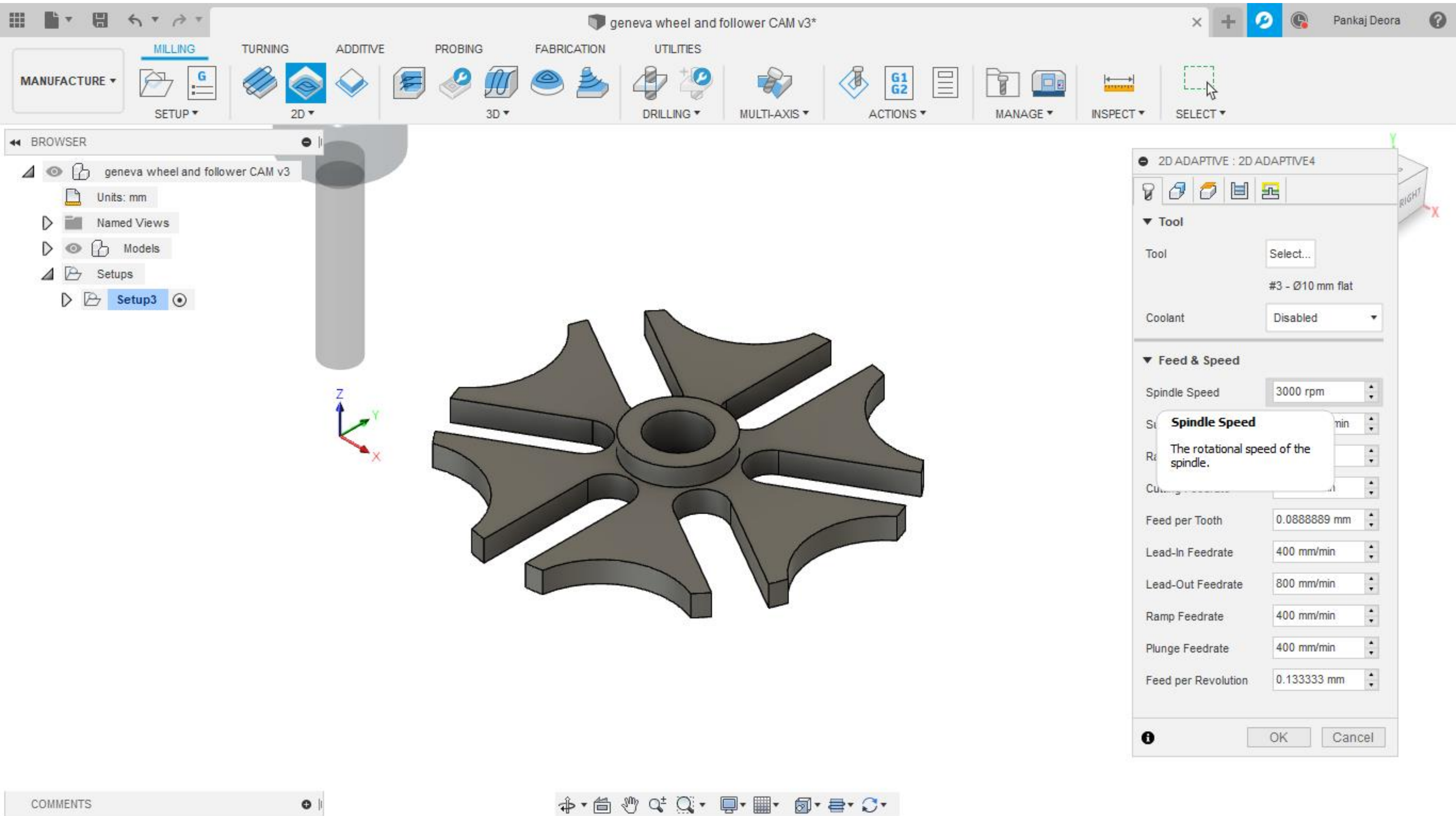
Select a regular end mill with 10mm diameter as shown. Click "OK"



Disable the coolant by clicking on “Disable” under “coolant ” option



‘Feed and speed’ is explained on the next slide. Keep the values as is for now



If you had to, this is how you can define speeds and feeds

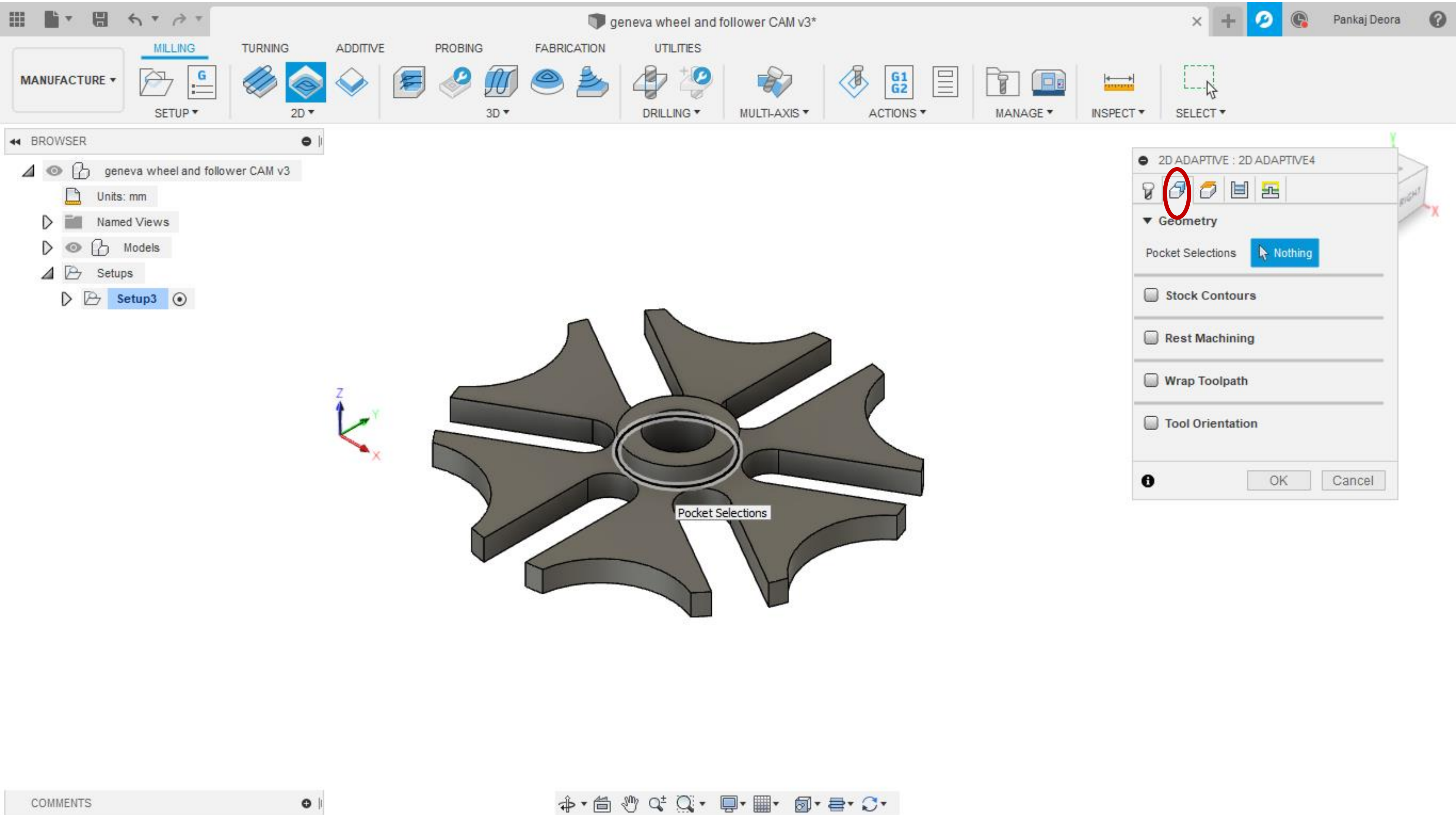
Cutting Speed $$Vc = \frac{\pi \times D \times n}{1{,}000}$$

Feed $$Vf = n \times fz \times Z$$

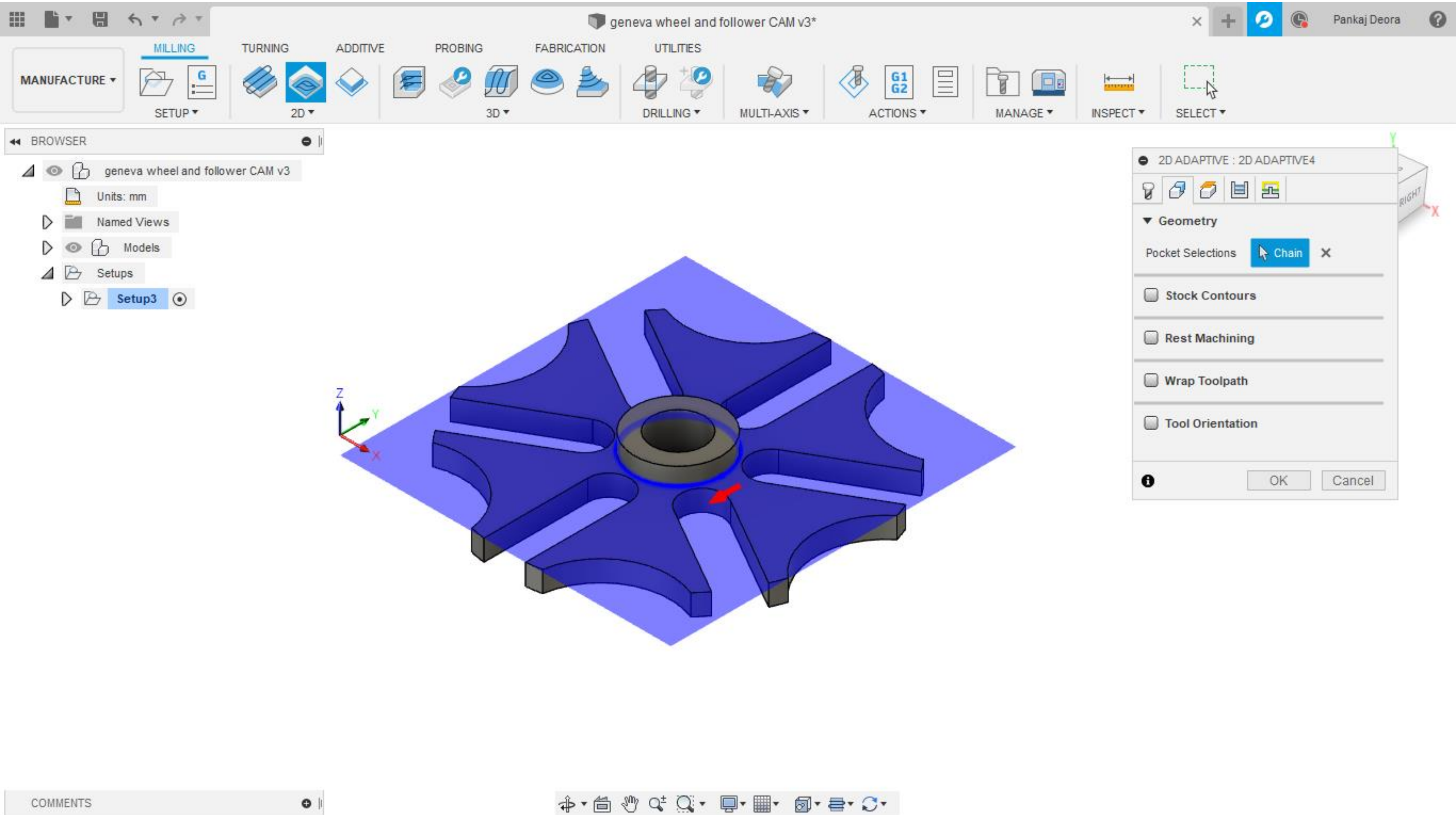
Where,
V_f = feed (mm/min)
n = Spindle speed (rpm)
f_z= feed (mm/tooth)
D = tool diameter (mm)
V_z = cuttting speed (m/min)
Z = no of flutes

For a 4 flute, 12mm dia solid carbide regular end mill,
For aluminum,
Cutting speed = (150 – 215) m/min
Feed = (0.089 – 0.127) mm/tooth
And for Steel,
Cutting speed = (105 - 150) m/min
Feed = (0.051 – 0.089) mm/tooth

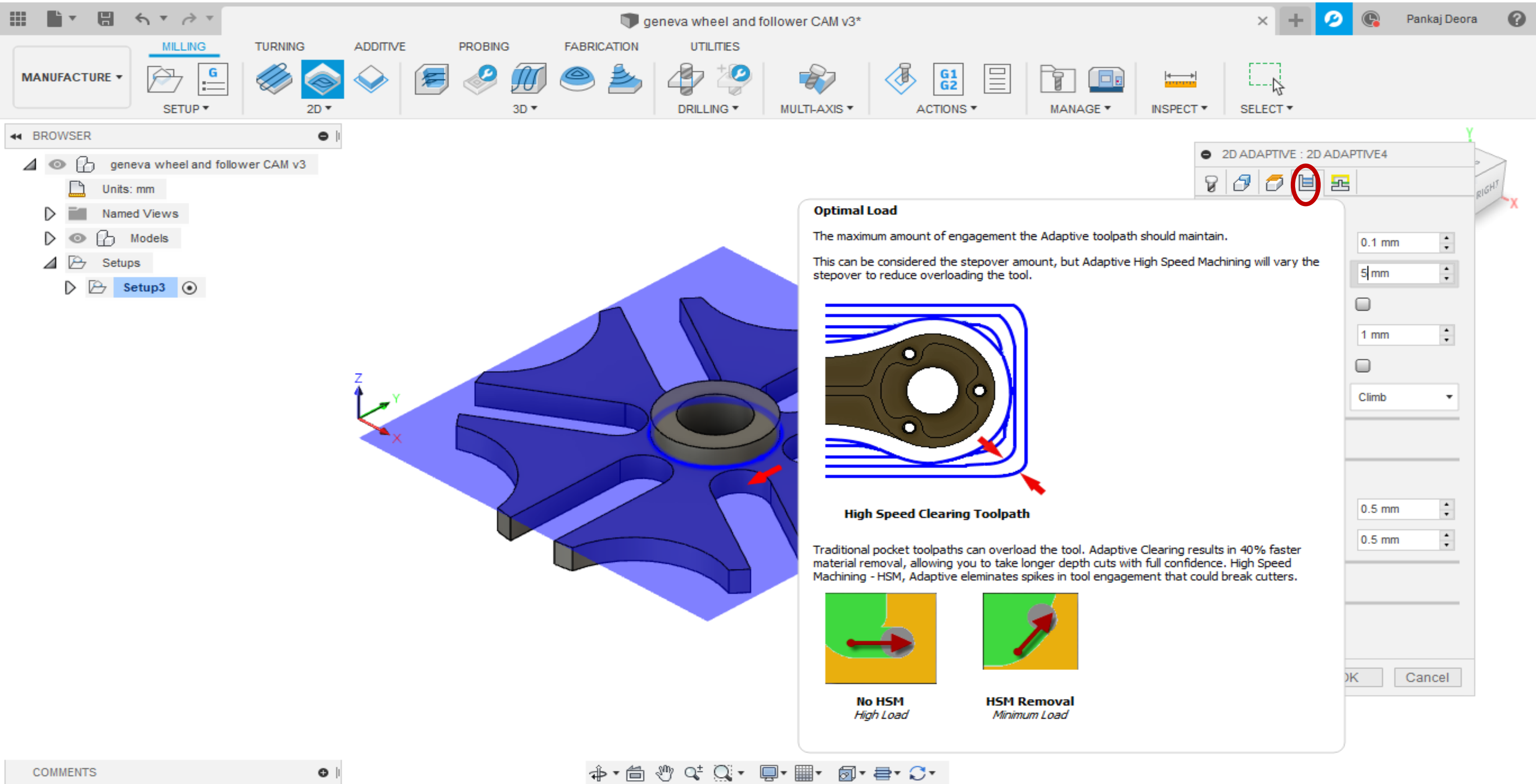
Click on the “Geometry ” tab. It asks us to select a pocket which is ‘nothing’ by default. For selecting a pocket, click on the boundary of the wheel as shown



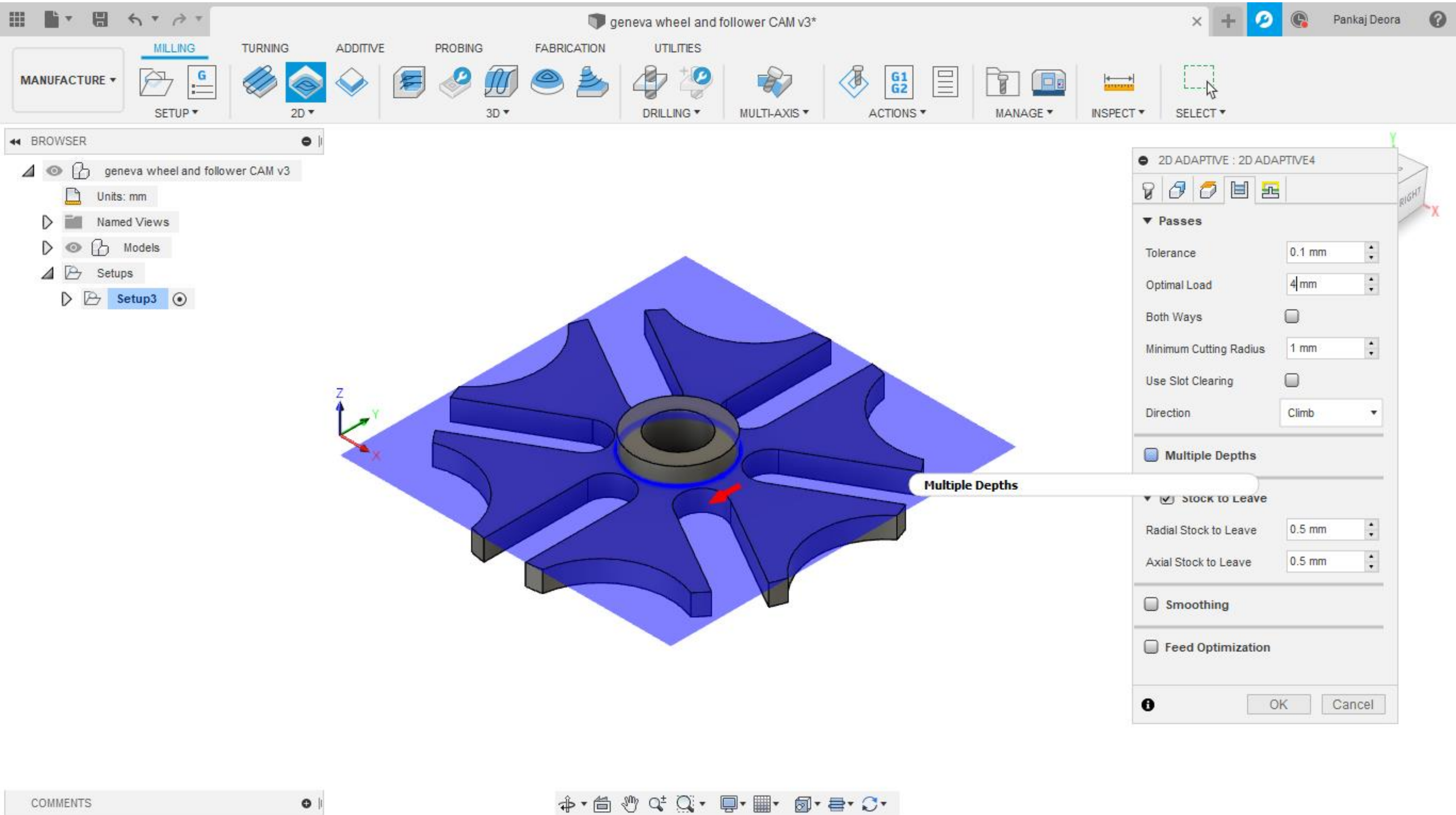
You can now see that a “chain” is selected.



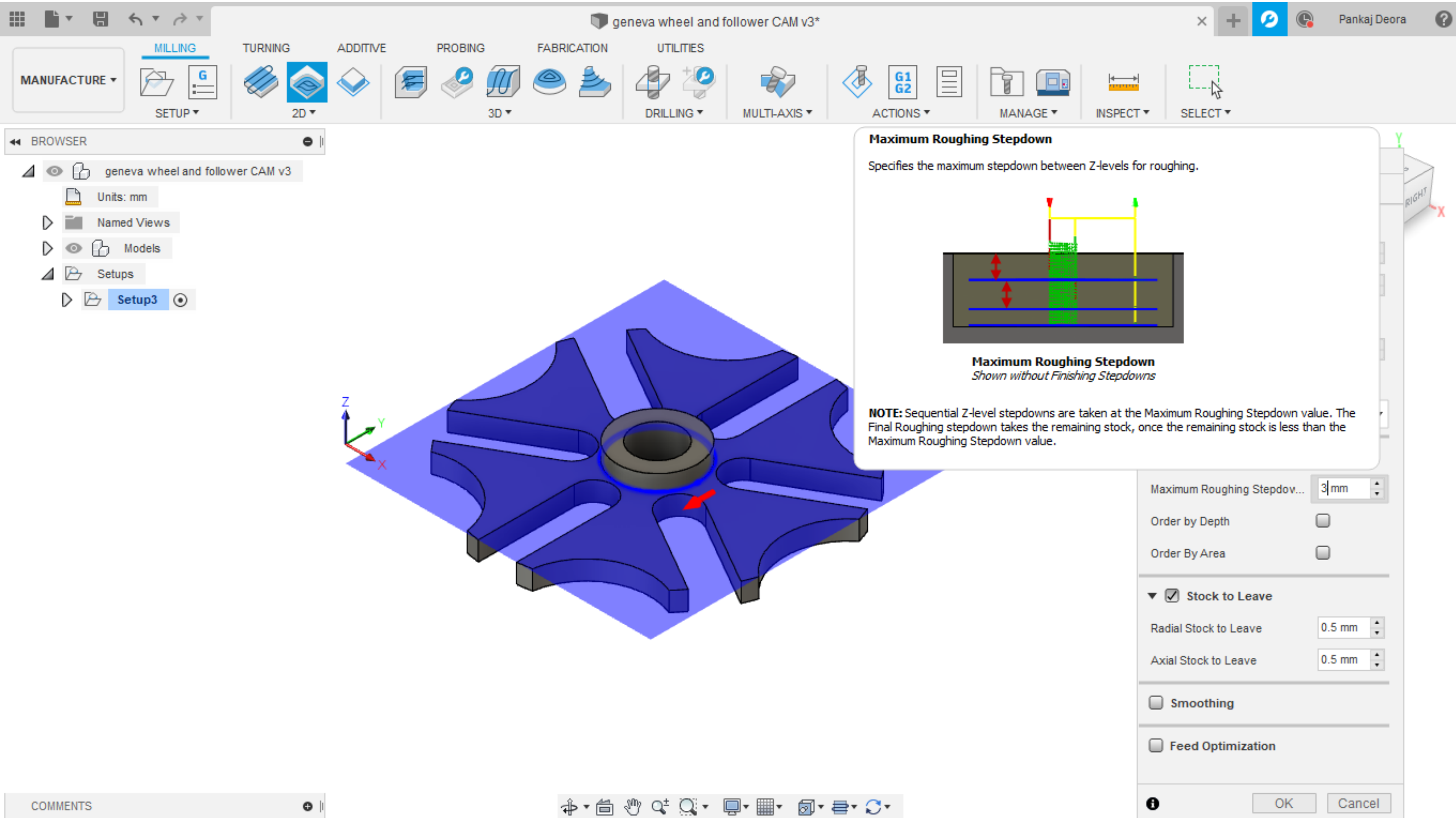
Under “Passes” tab, enter the value “5mm” in the “optimal load” option. Optimal load on a tool is the percentage of the tool that is engaged with the work-piece. You can read more about this by placing the cursor on it



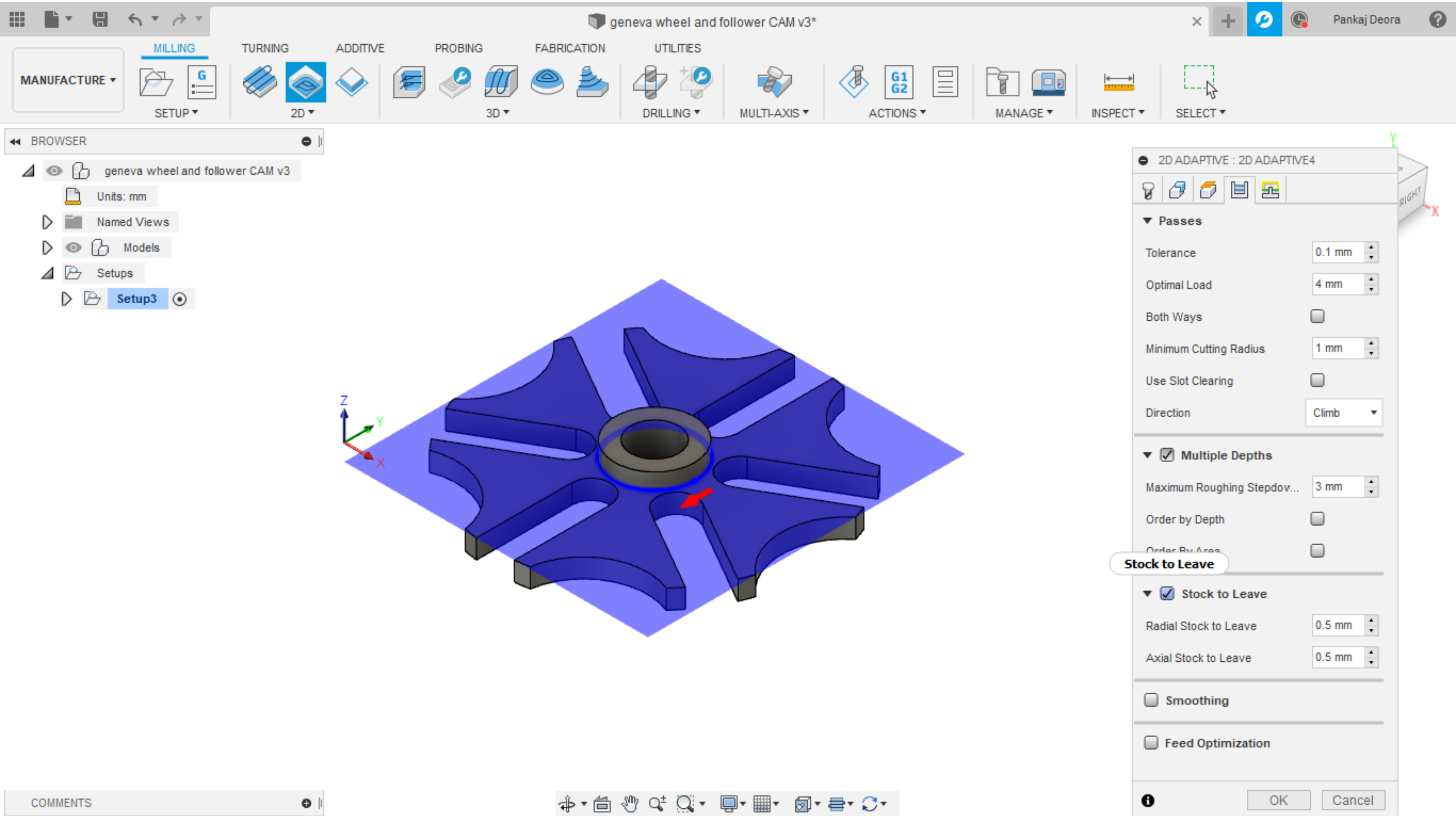
Check the “Multiple depths” box



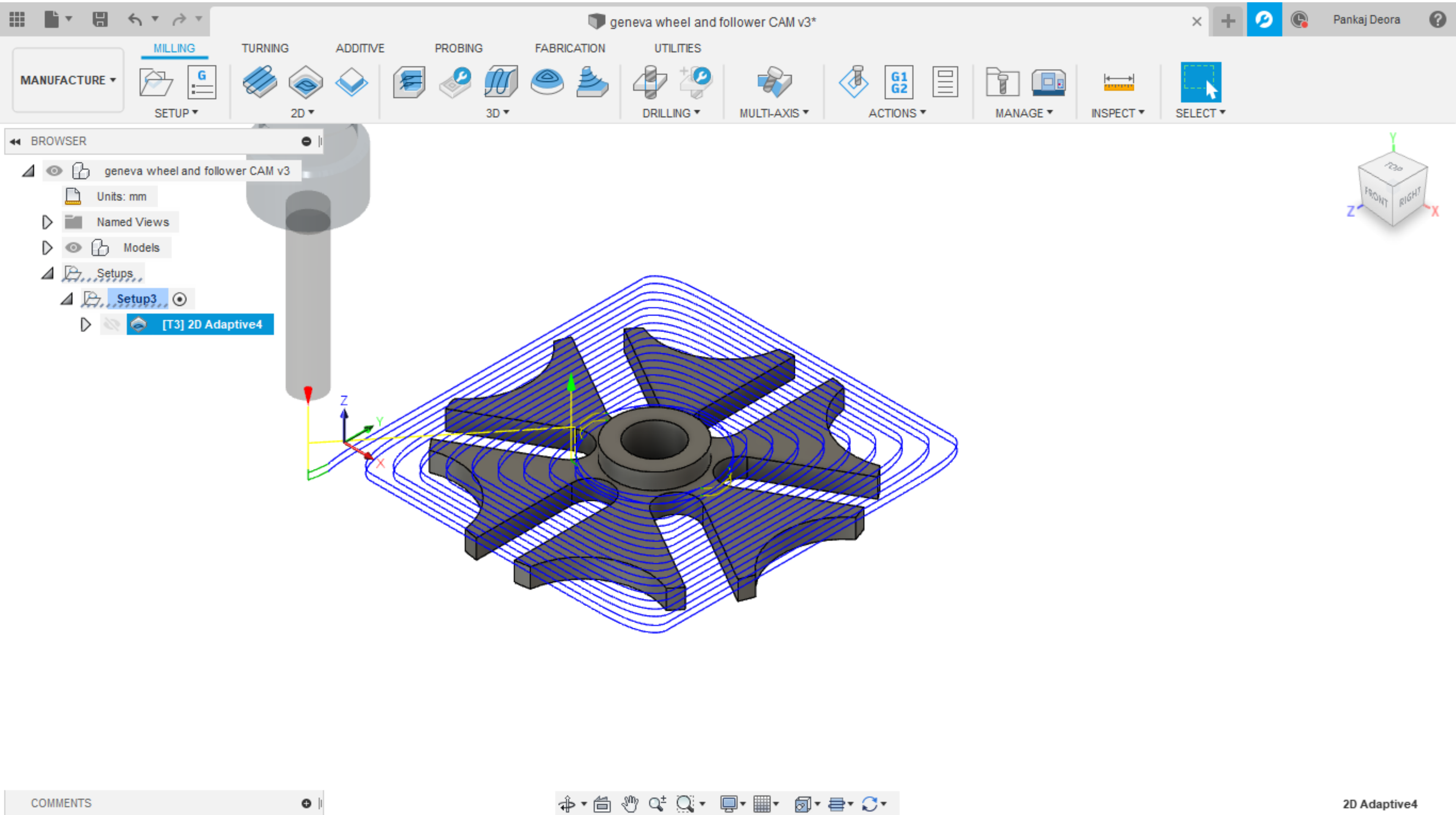
Under “Maximum roughing stepdown” option, put a value of 3mm. Stepdown means that how much part of the tool is engaged with the workpiece in the vertical direction (z-axis)

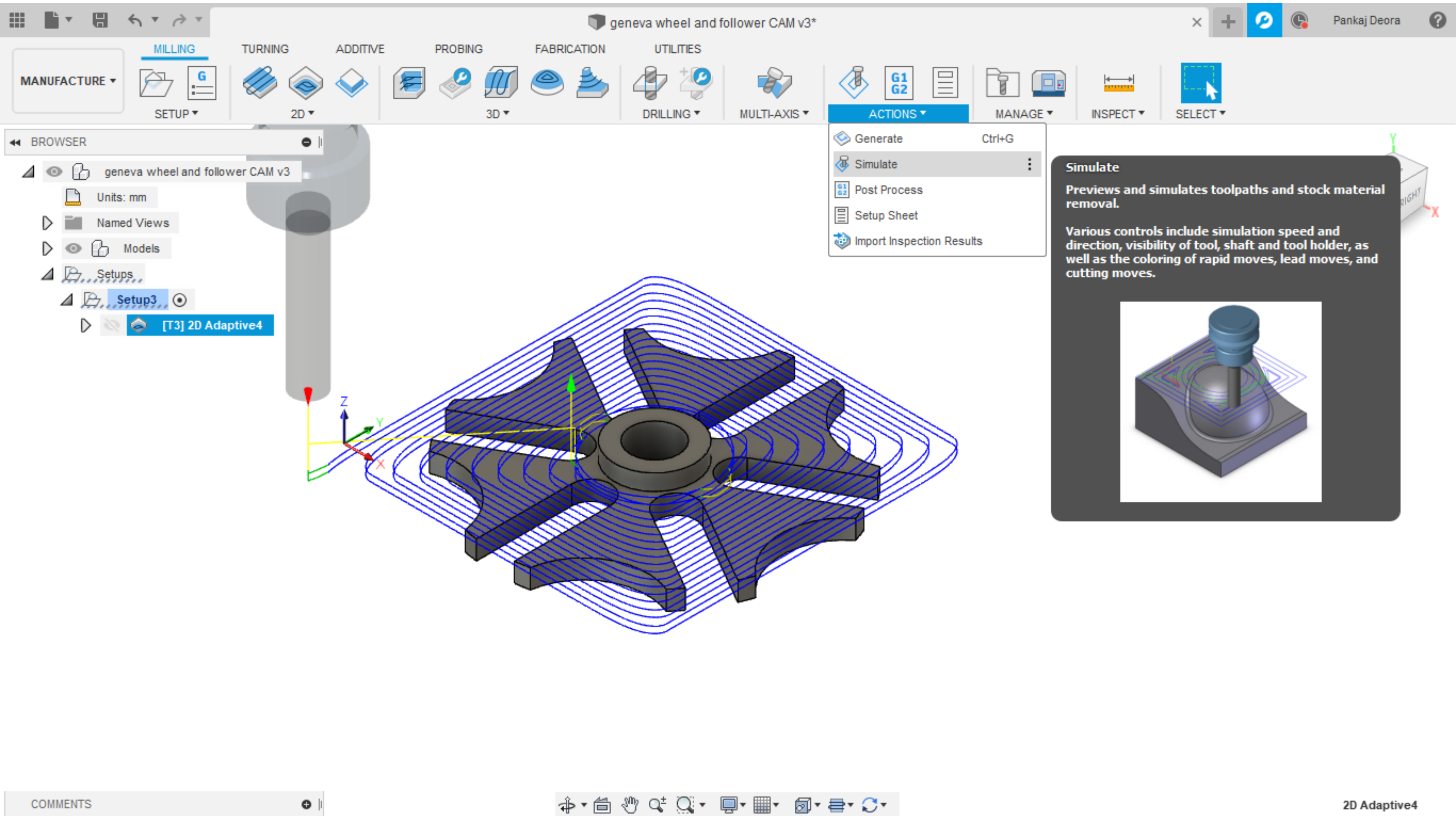


Uncheck the “Stock to leave” box. Click “OK”

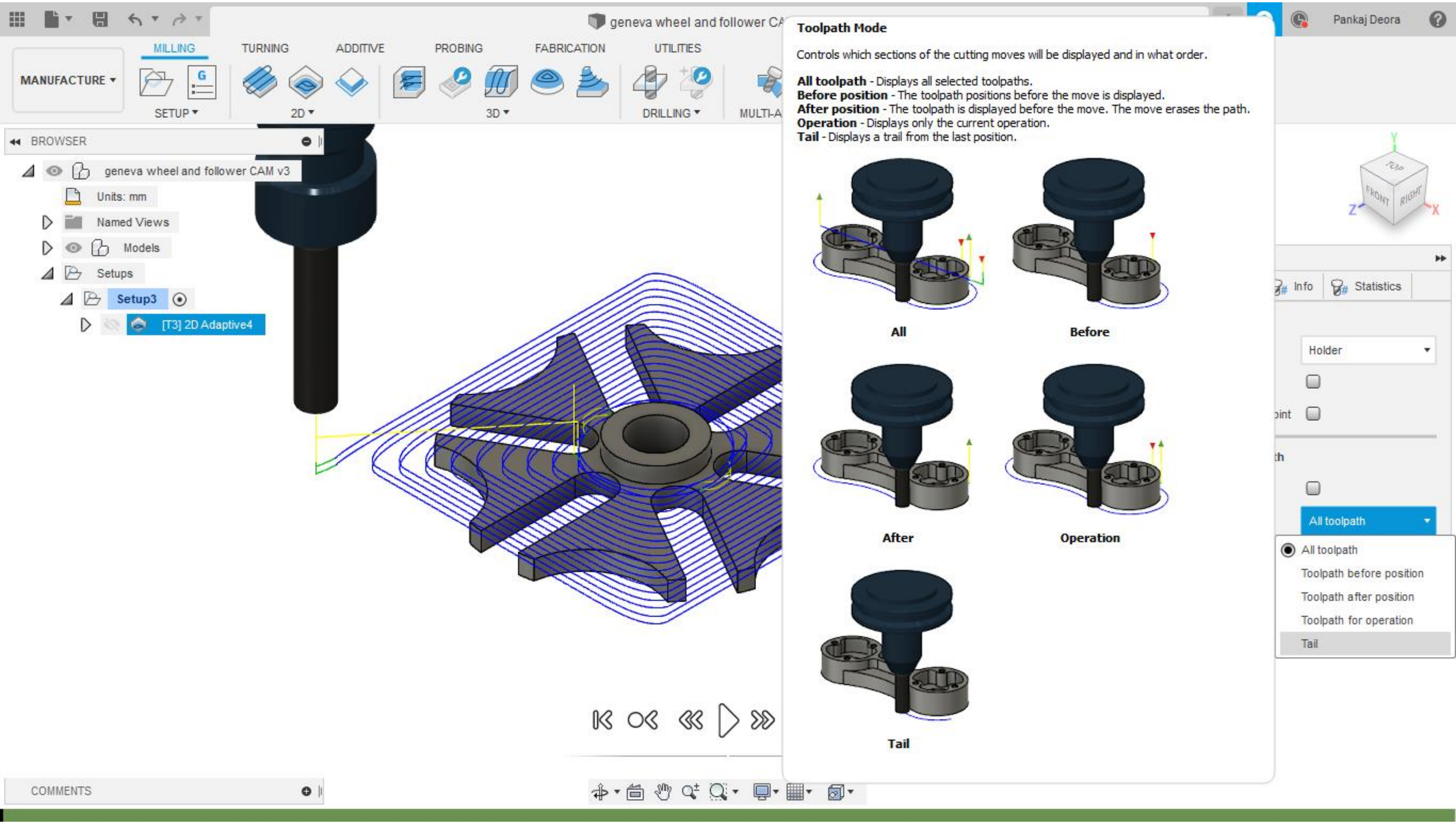


This will generate a toolpath. You can simulate the tool path generated by clicking on "Simulate" as shown in the next slide

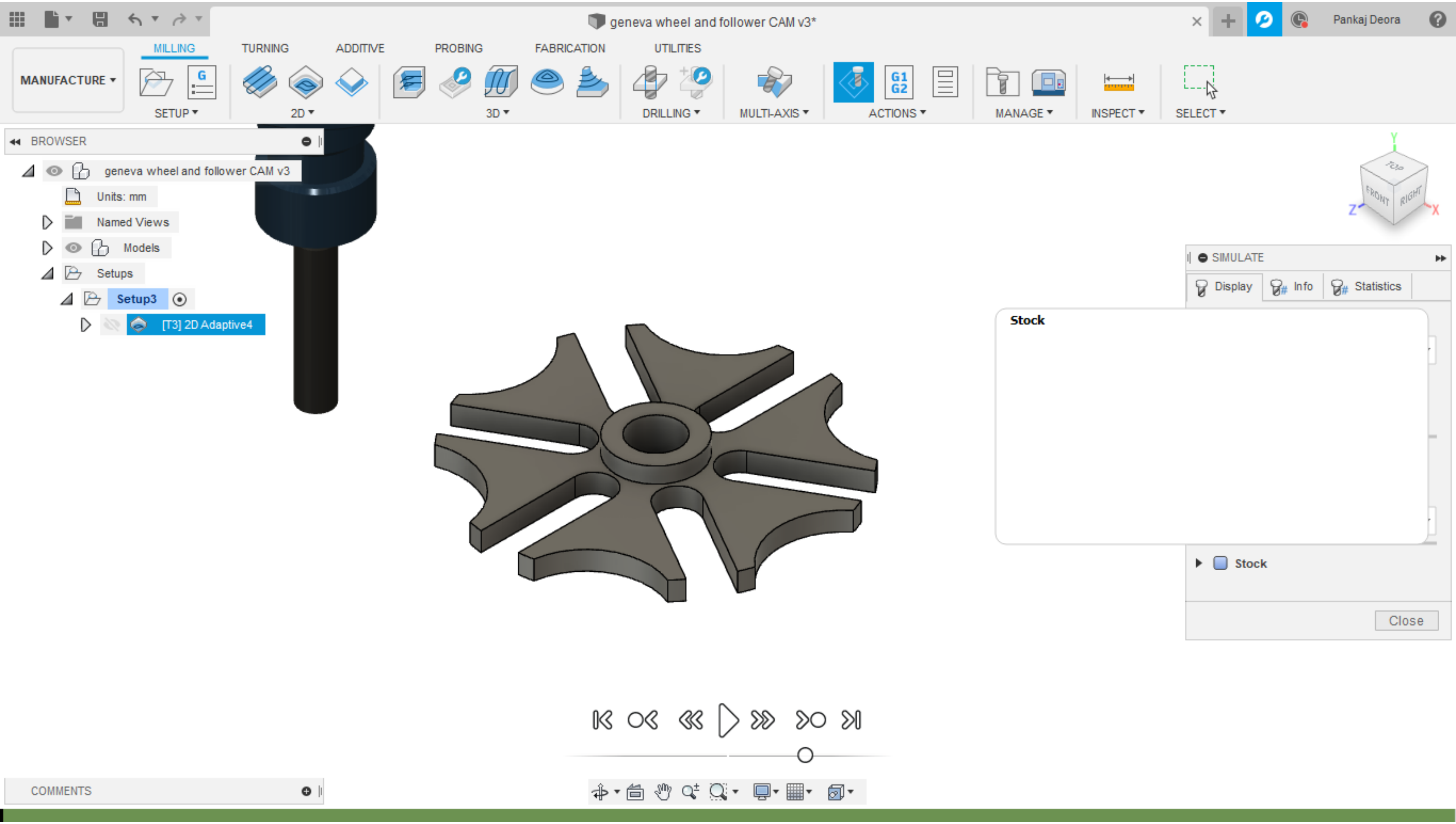



geneva wheel and follower CAM v3*
Pankaj Deora
MILLING
TURNING
ADDITIVE
PROBING
FABRICATION
UTILITIES
MANUFACTURE
SETUP
2D
3D
DRILLING
MULTI-AXIS
ACTIONS
MANAGE
INSPECT
SELECT
BROWSER
geneva wheel and follower CAM v3
Units: mm
Named Views
Models
Setups
Setup3
[T3] 2D Adaptive4
Generate Ctrl+G
Simulate
Post Process
Setup Sheet
Import Inspection Results
Simulate
Previews and simulates toolpaths and stock material removal.
Various controls include simulation speed and direction, visibility of tool, shaft and tool holder, as well as the coloring of rapid moves, lead moves, and cutting moves.
COMMENTS
2D Adaptive4

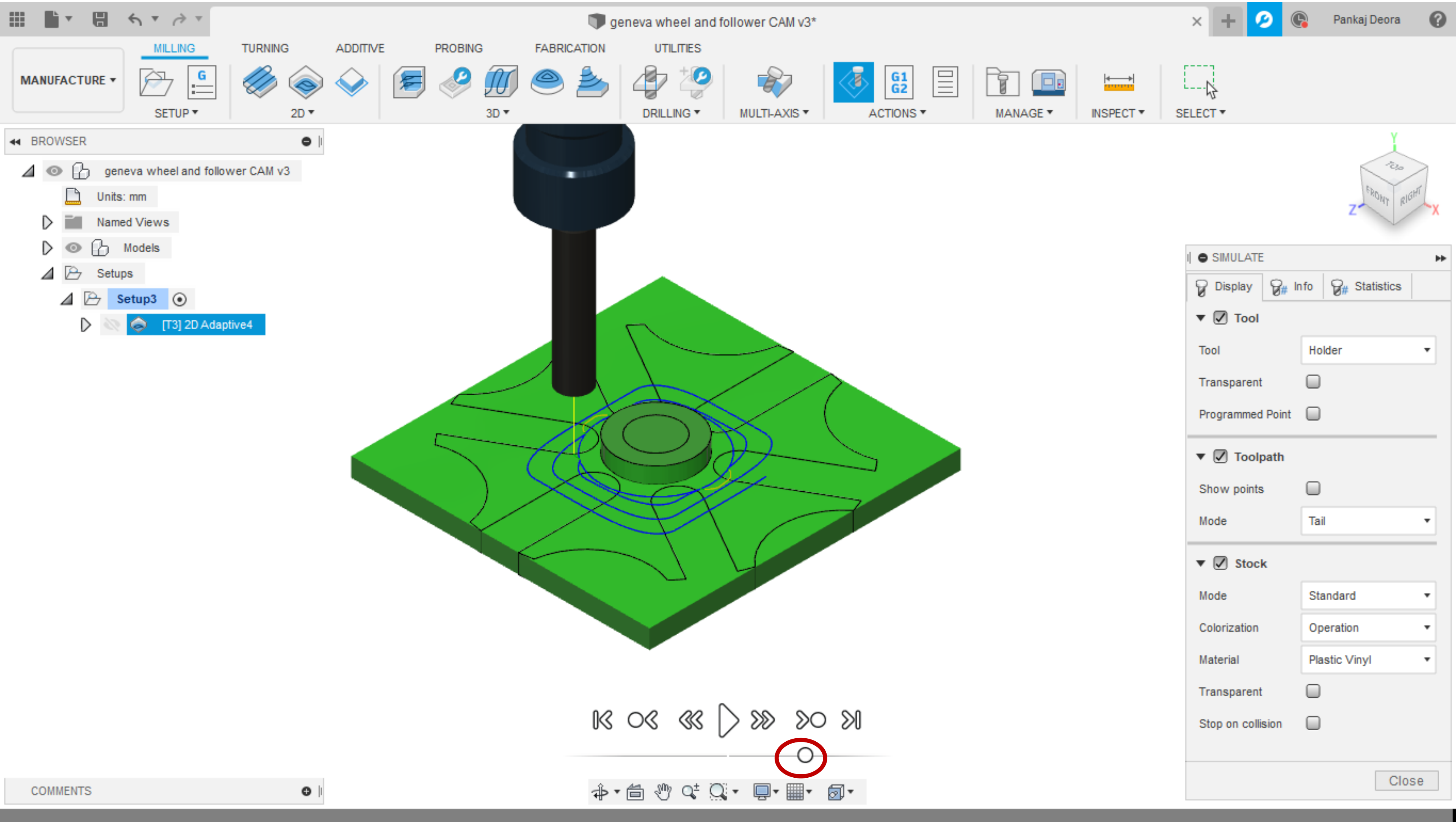
Under “Mode” option, select “Tail”. This is only done for clarity



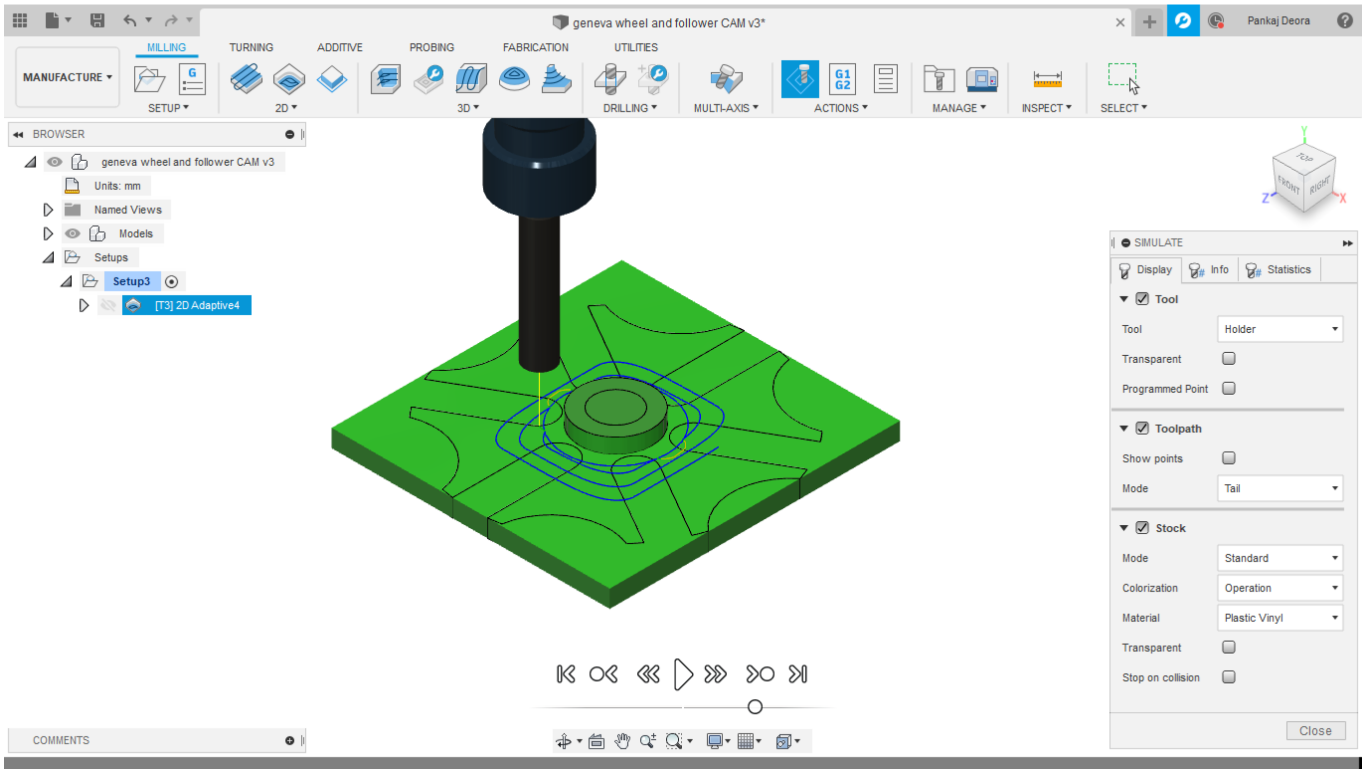
Check the “Stock” box



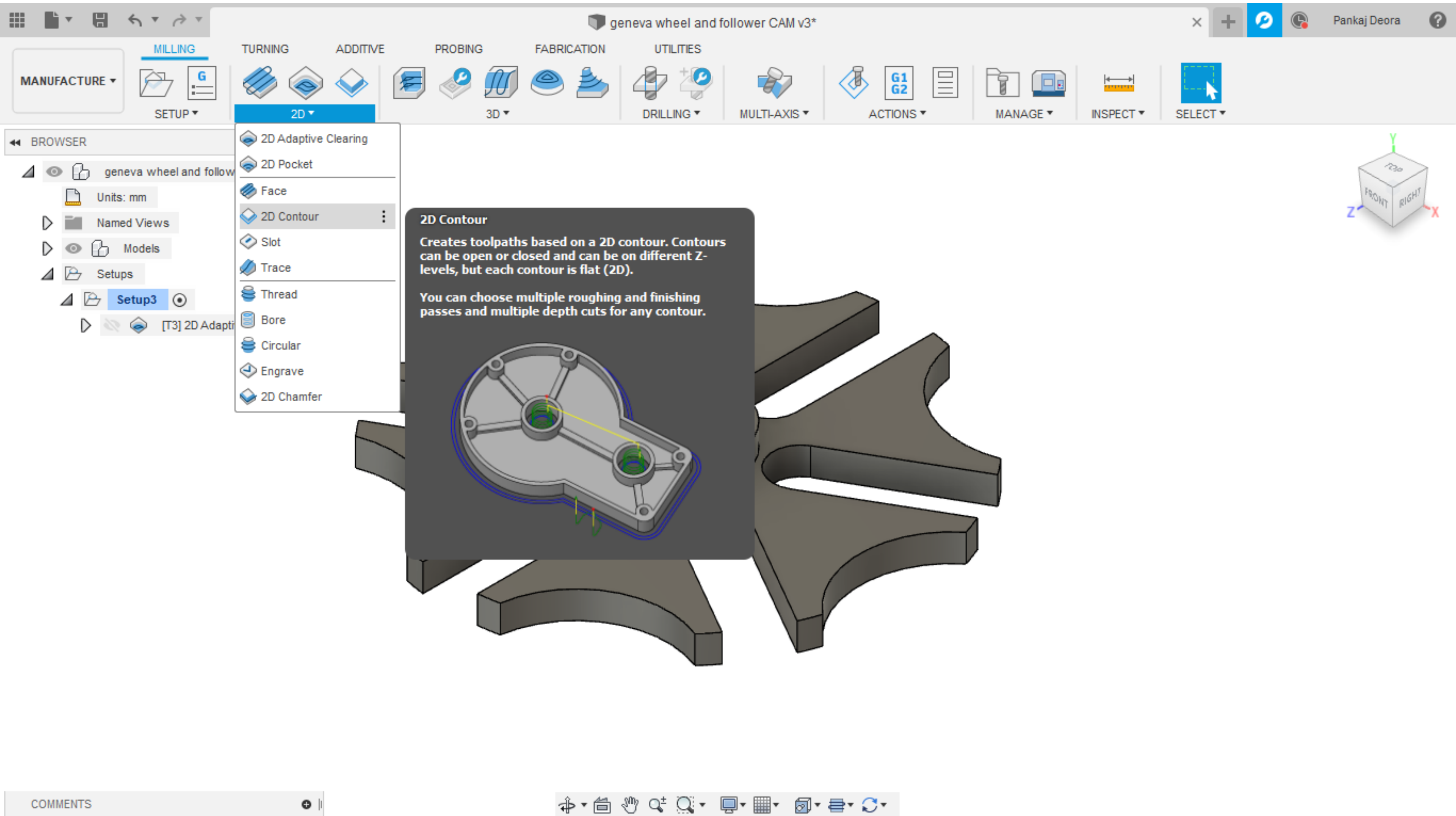
Click on the “Play” button to start the simulation. You can control the simulation speed by dragging the pointer as shown in left or right direction



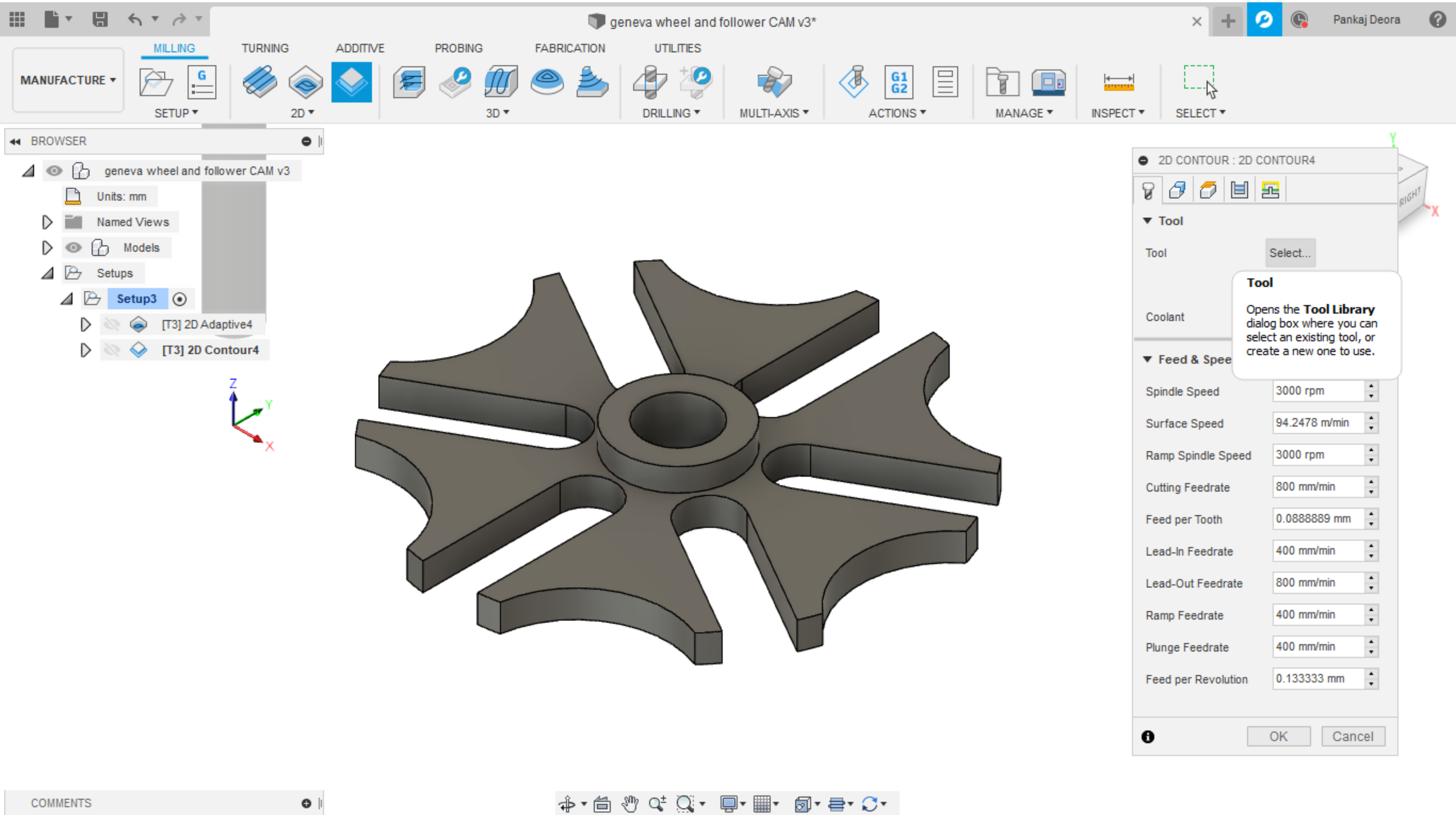
If you are able to successfully simulate this step, please show it to your guide/TA/Tutor before proceeding further.



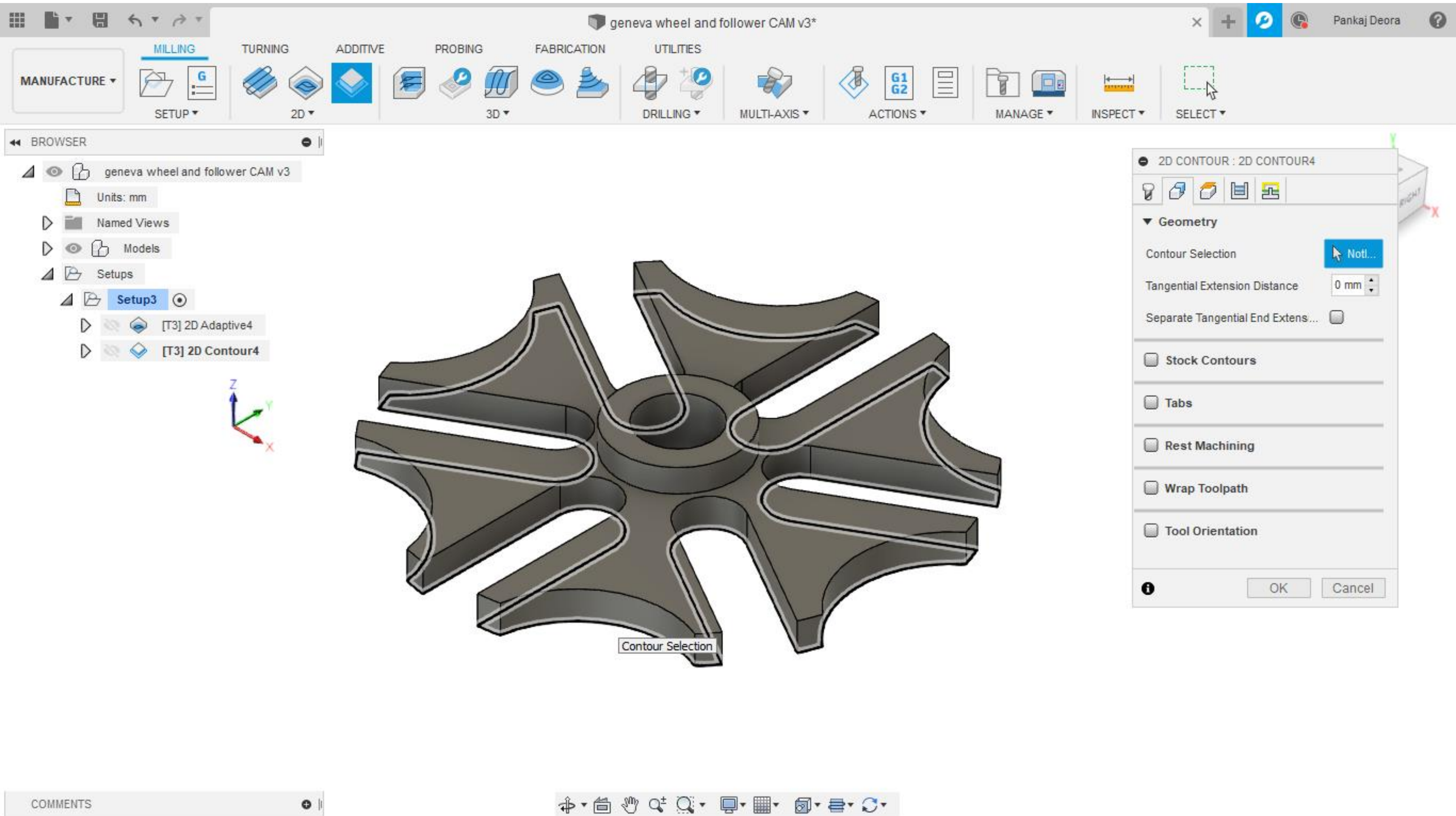
To cut the bottom part , we select a different strategy called ‘2D contour’ as shown



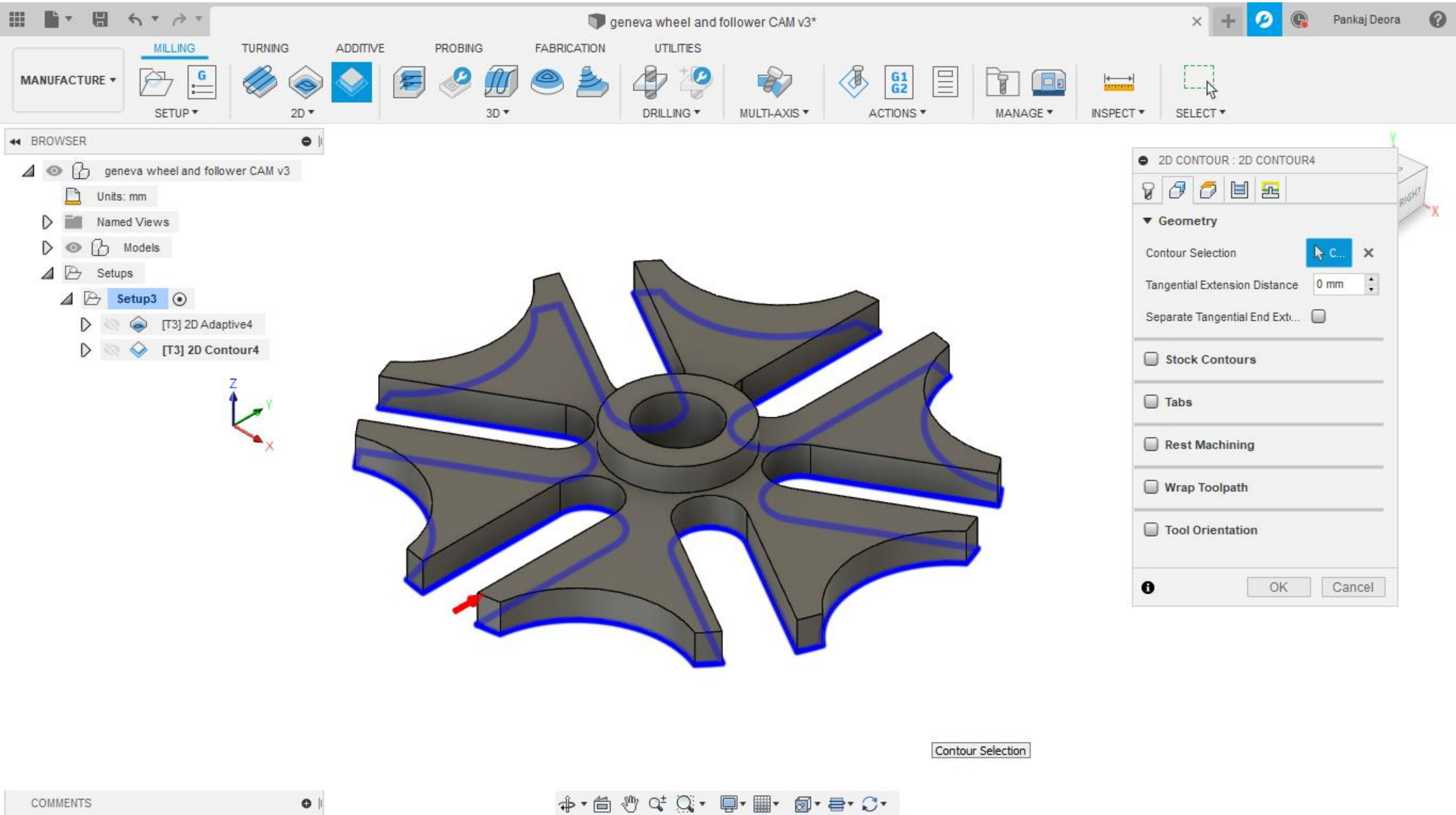
Again select the same 10mm diameter tool following the same steps defined earlier. Also ensure that the coolant is disabled



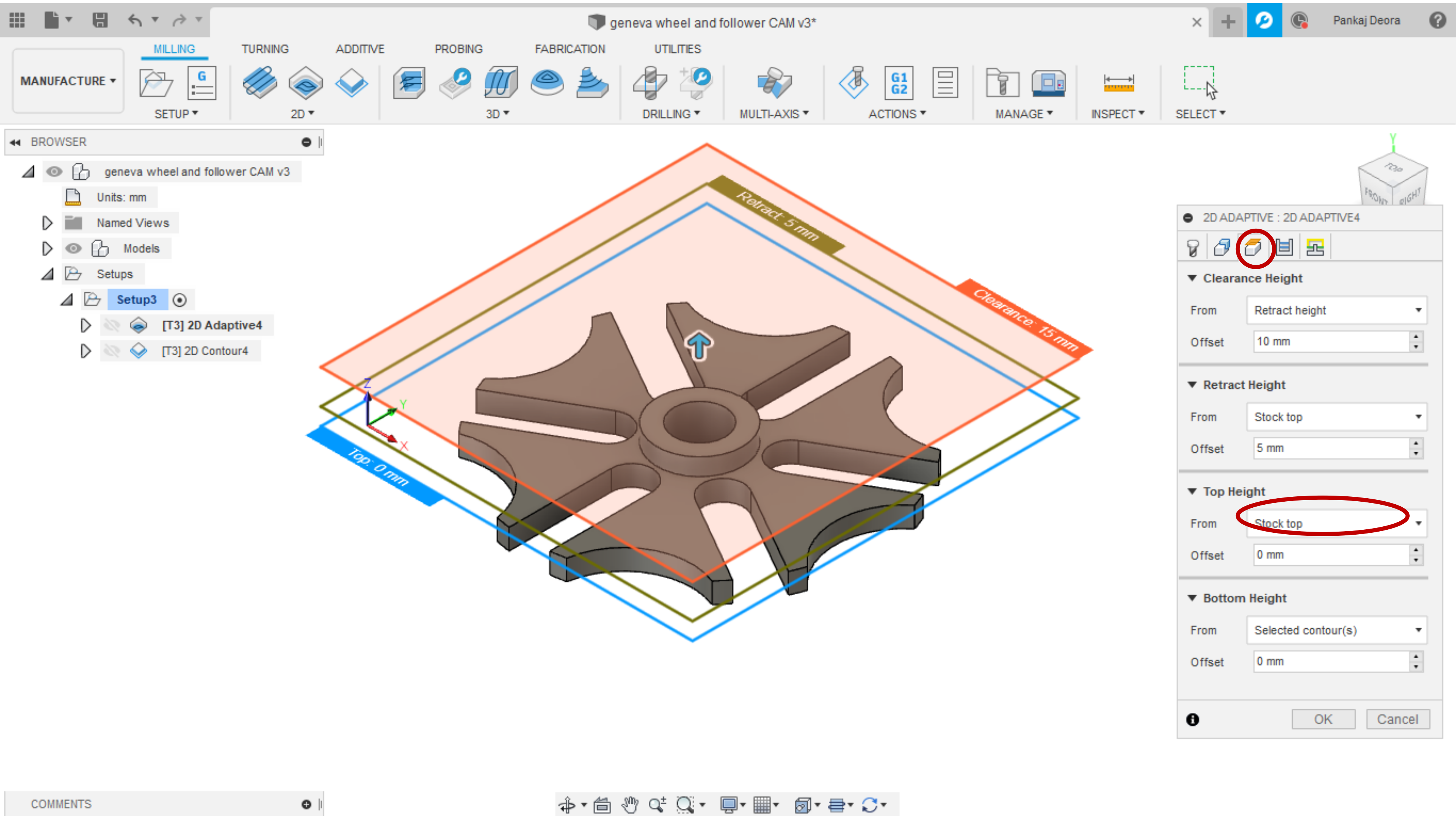
Go to “Geometry” tab and select the contour as shown below

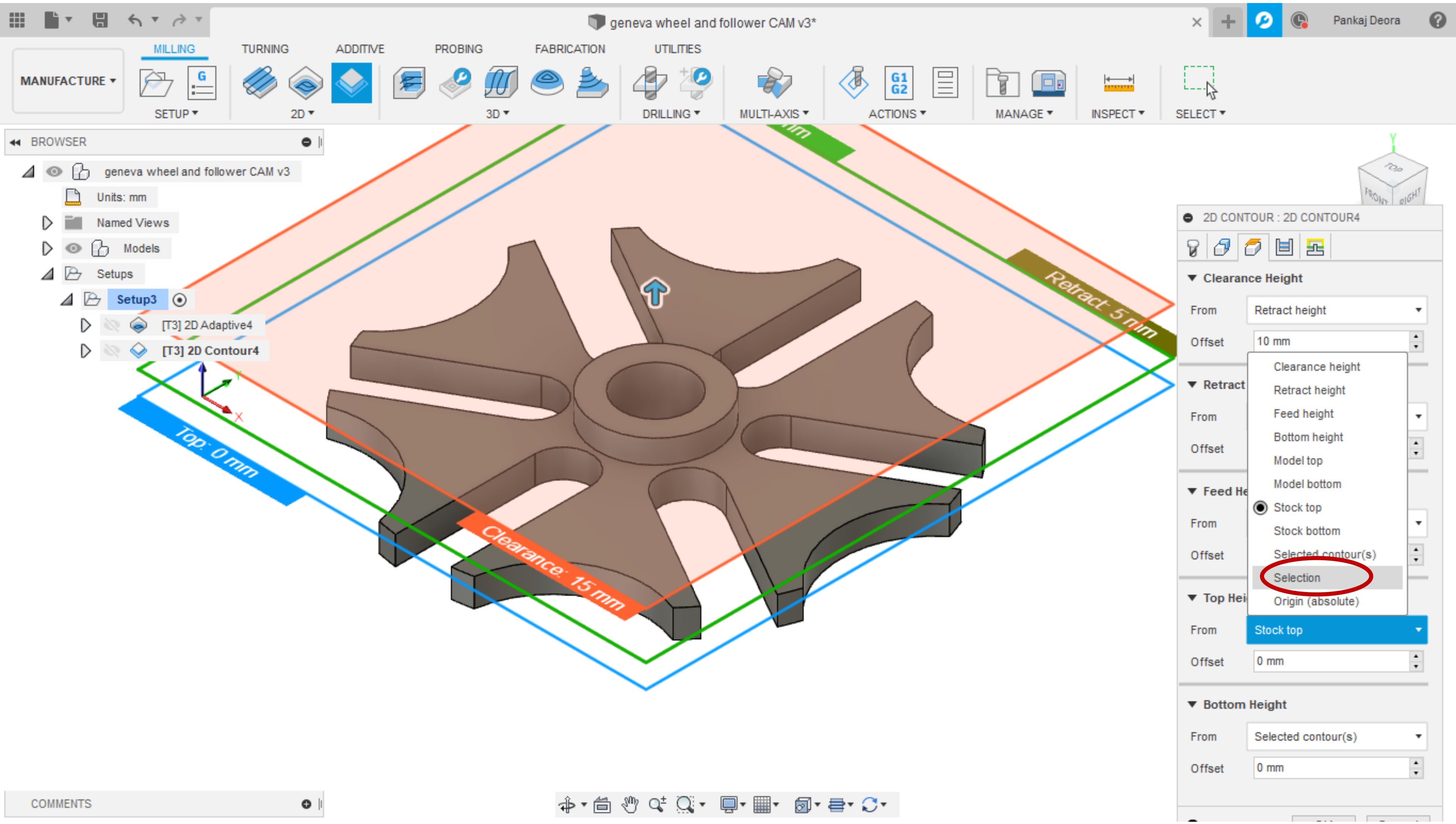


After selection, it will appear like this

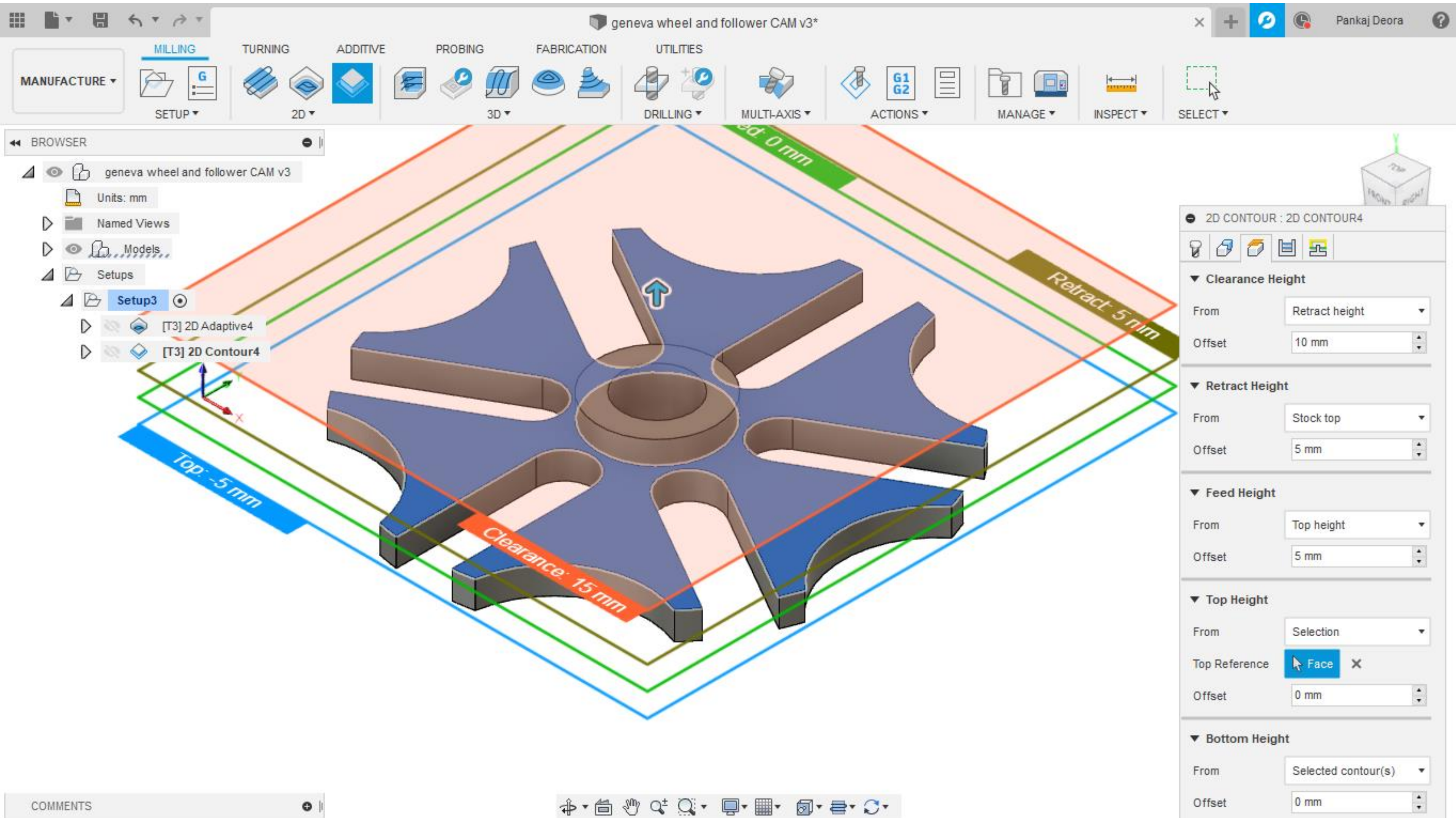


Go to "Heights" tab and click on "selection" under "Top height" option

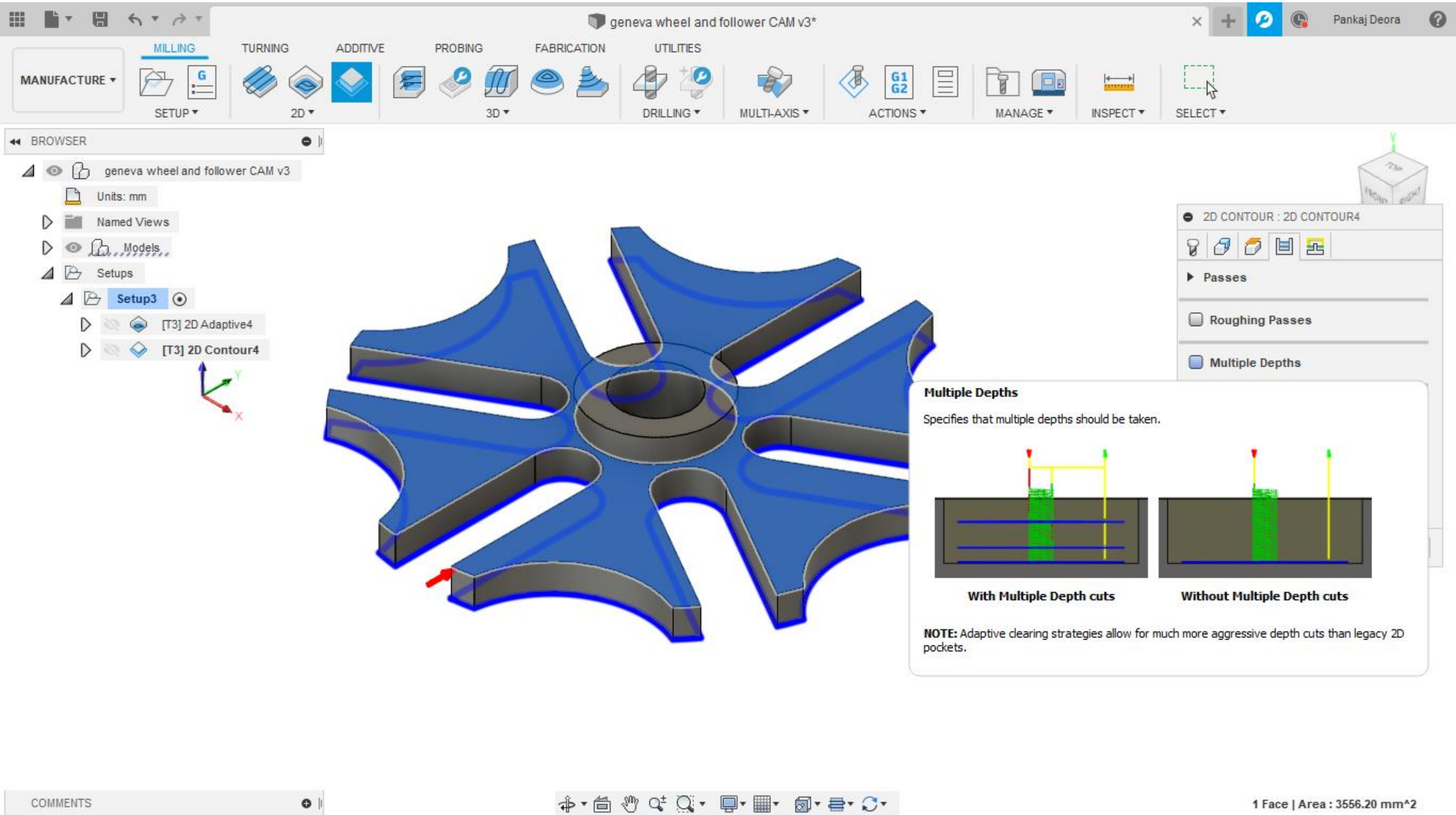



geneva wheel and follower CAM v3*
Pankaj Deora
MILLING
TURNING
ADDITIVE
PROBING
FABRICATION
UTILITIES
MANUFACTURE
SETUP
2D
3D
DRILLING
MULTI-AXIS
ACTIONS
MANAGE
INSPECT
SELECT
BROWSER
geneva wheel and follower CAM v3
Units: mm
Named Views
Models
Setups
Setup3
[T3] 2D Adaptive4
[T3] 2D Contour4
COMMENTS
Top: 0 mm
Clearance: 15 mm
Retract: 5 mm
2D CONTOUR : 2D CONTOUR4
Clearance Height
From
Retract height
Offset
10 mm
Retract
Feed He
Top Hei
Clearance height
Retract height
Feed height
Bottom height
Model top
Model bottom
Stock top
Stock bottom
Selected contour(s)
Selection
Origin (absolute)
Stock top
0 mm
Bottom Height
Selected contour(s)
0 mm

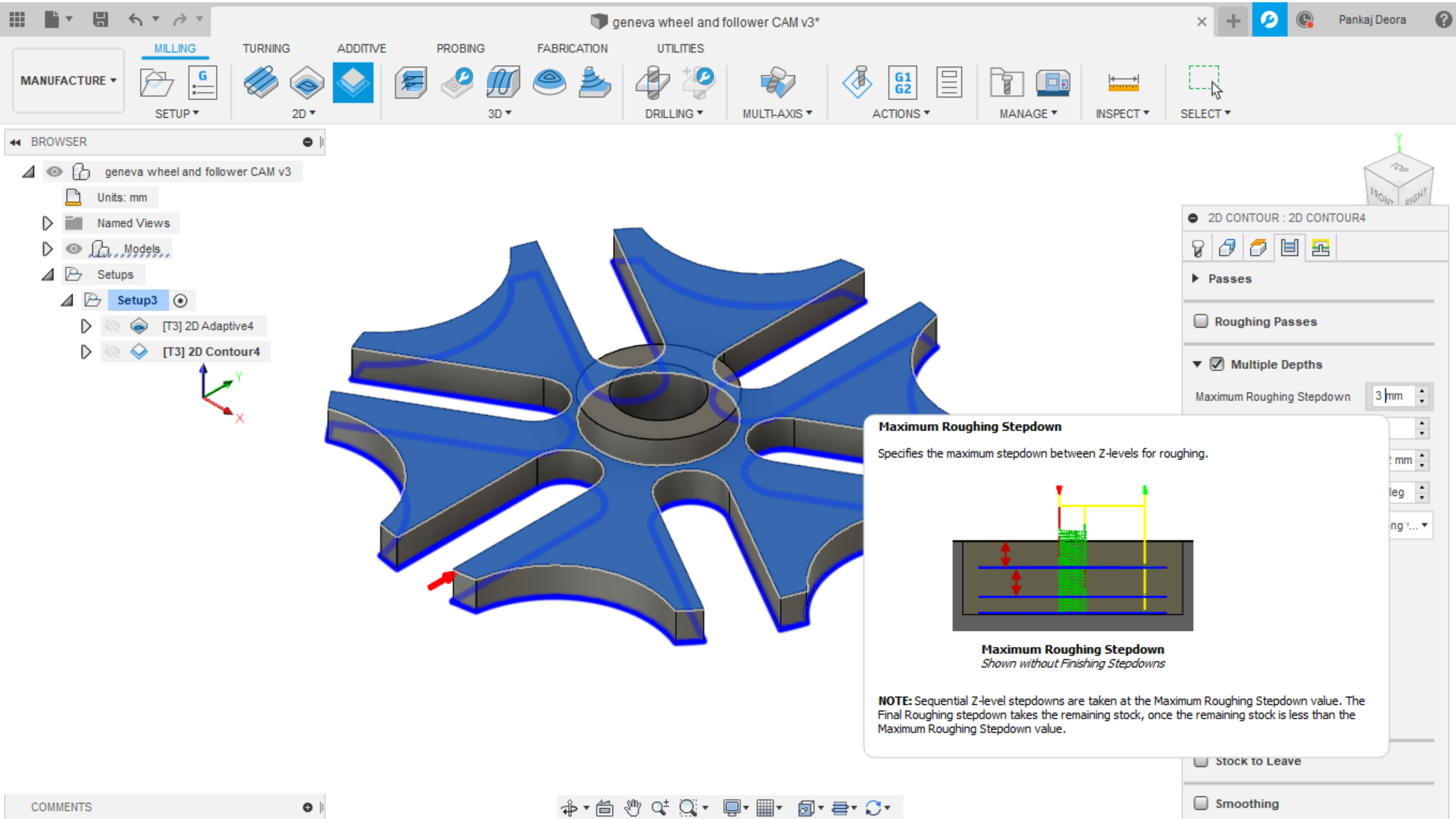
Select the top face of the wheel as shown



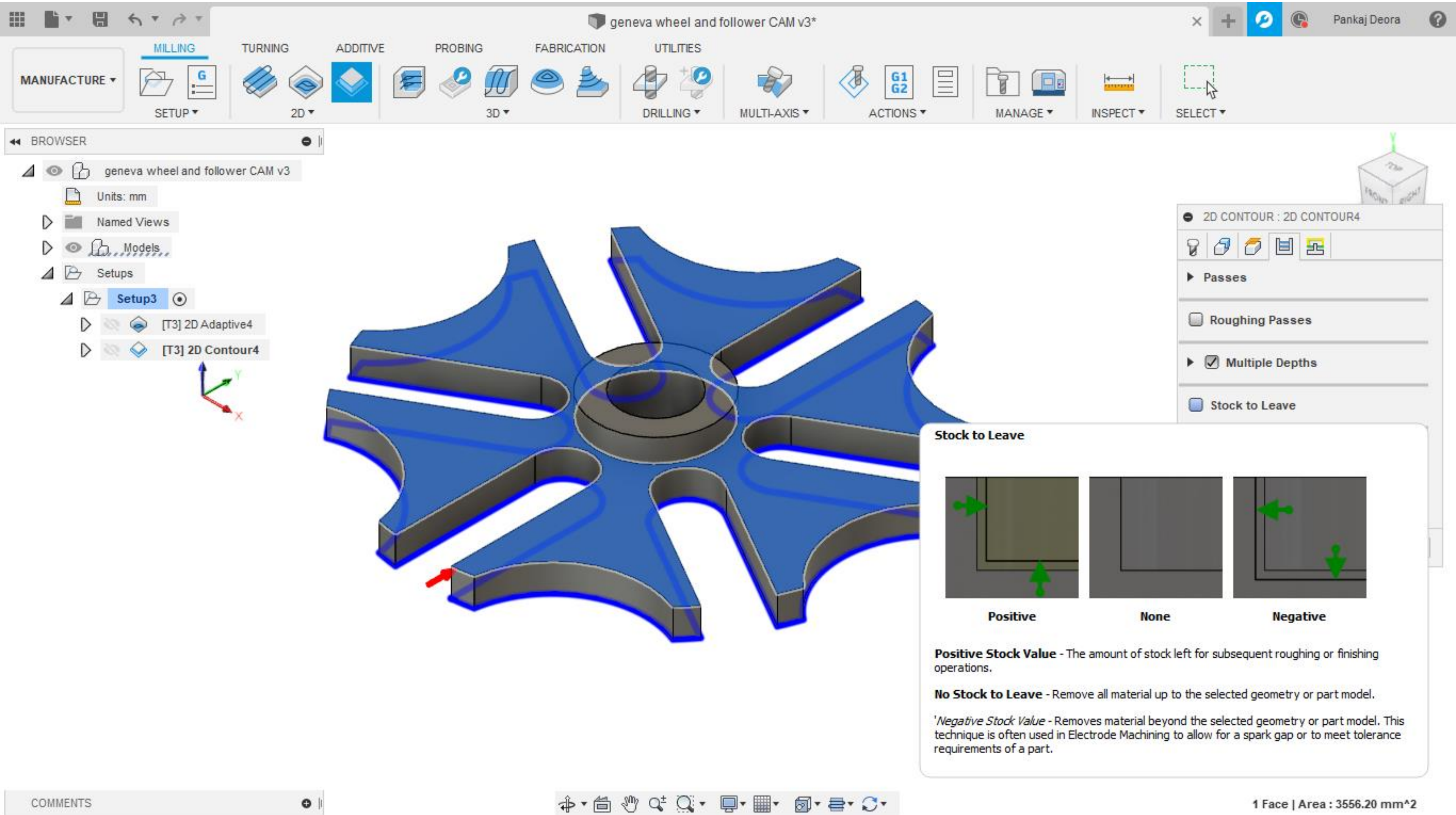
Under “Passes” tab, check “Multiple Depths” box



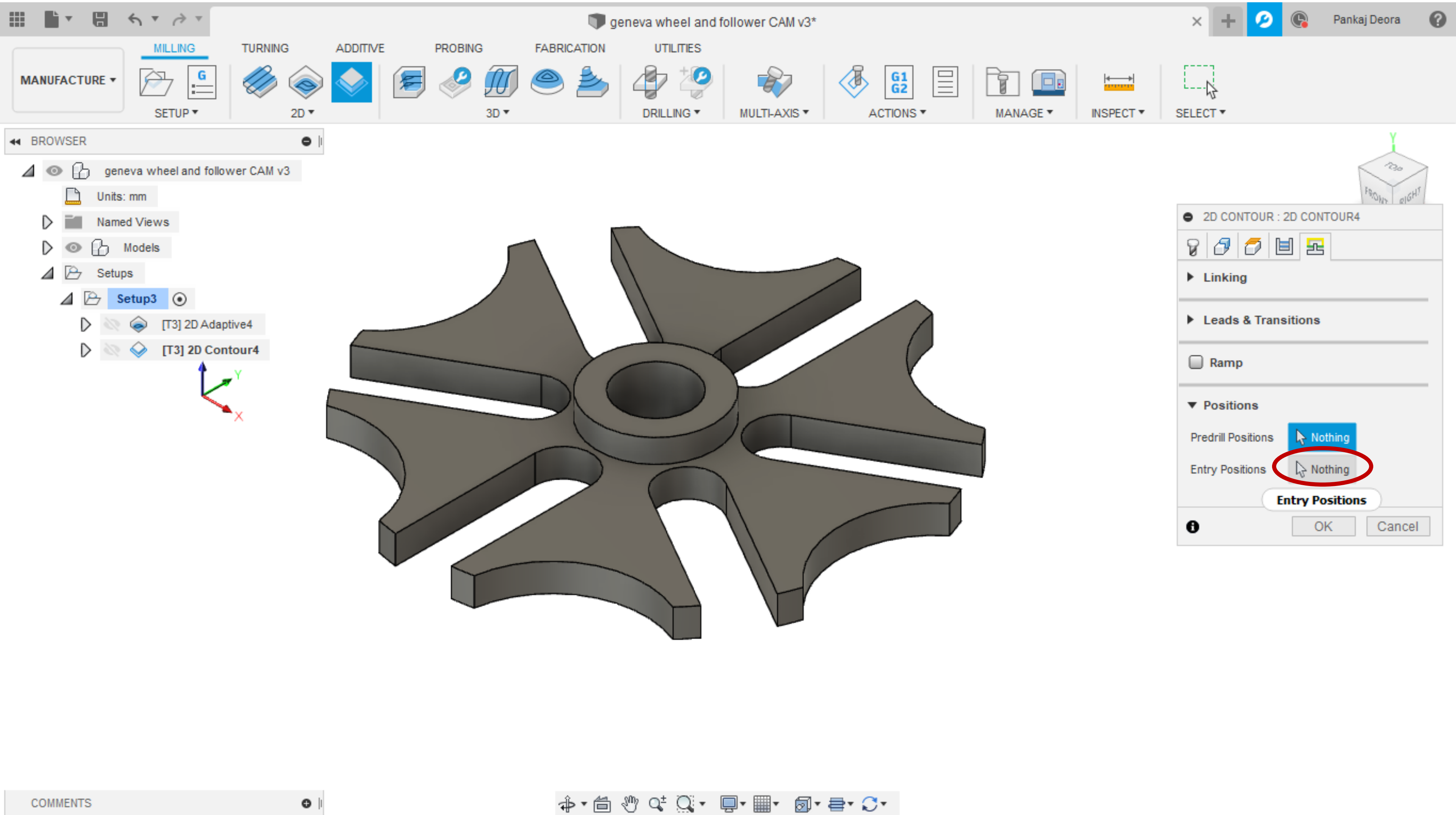
Again enter 3mm value under “Maximum Roughing Stepdown”



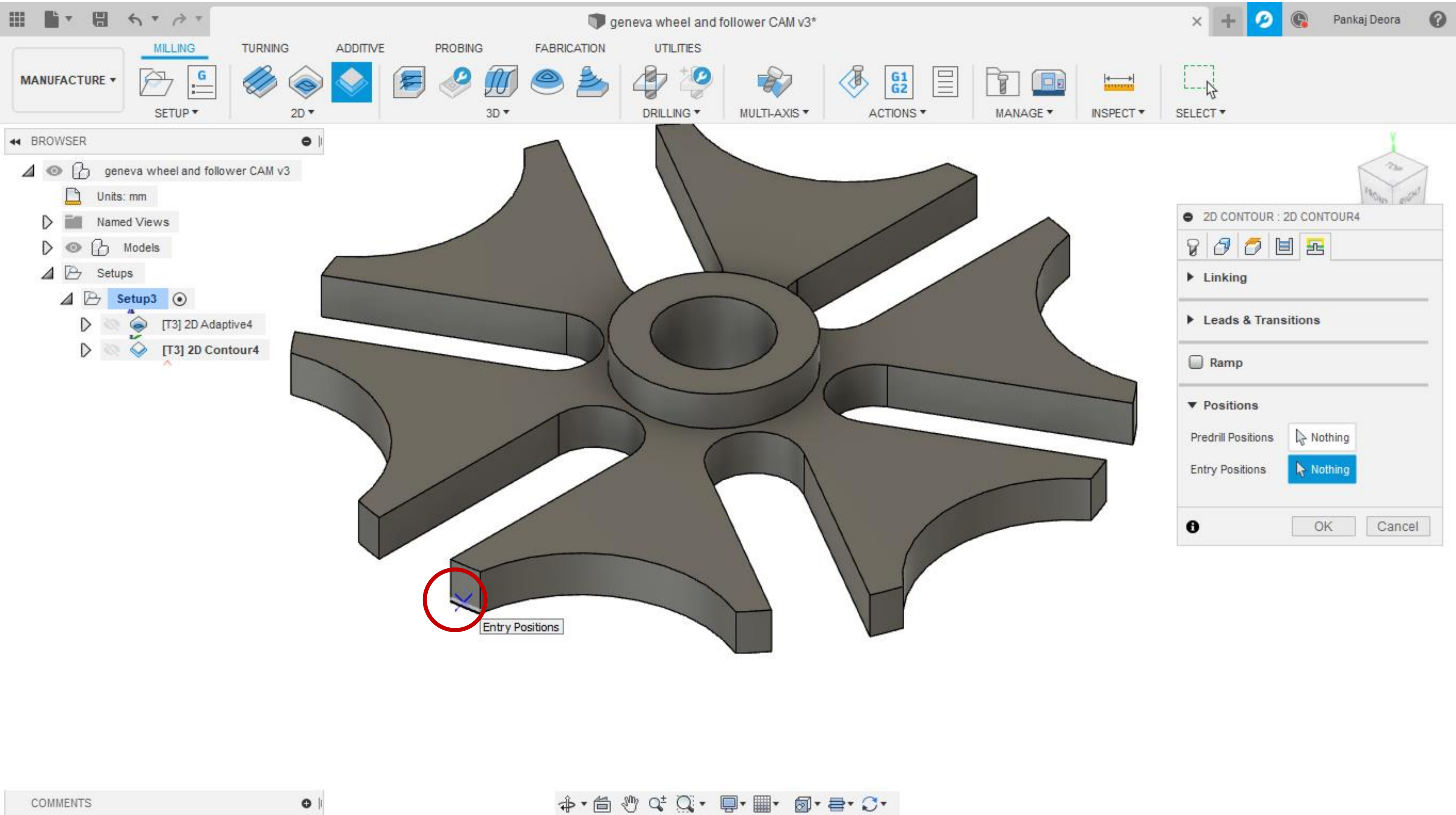
Uncheck “Stock to leave” box



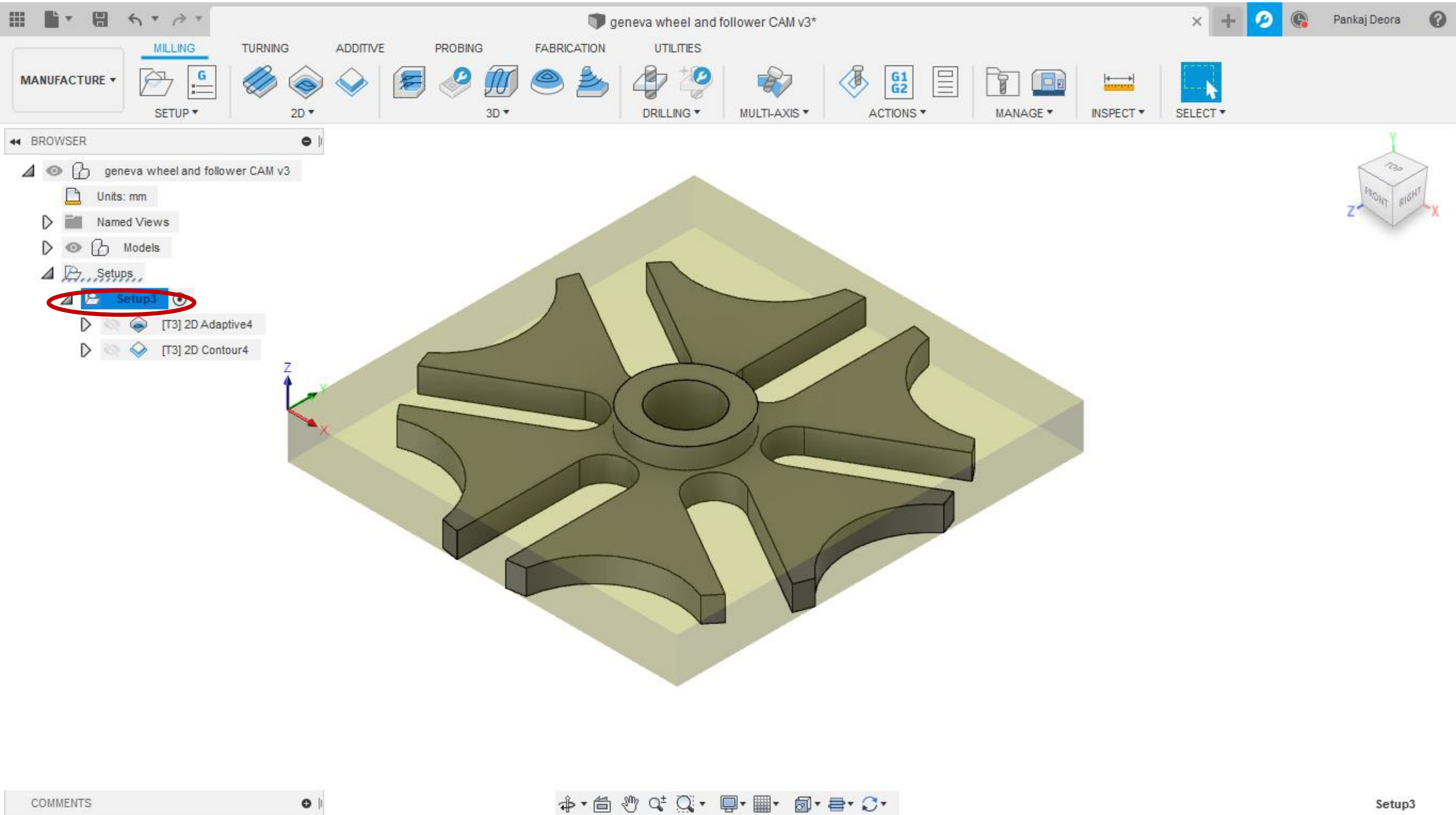
Go to “Linking” tab and click on “Entry positions” as shown



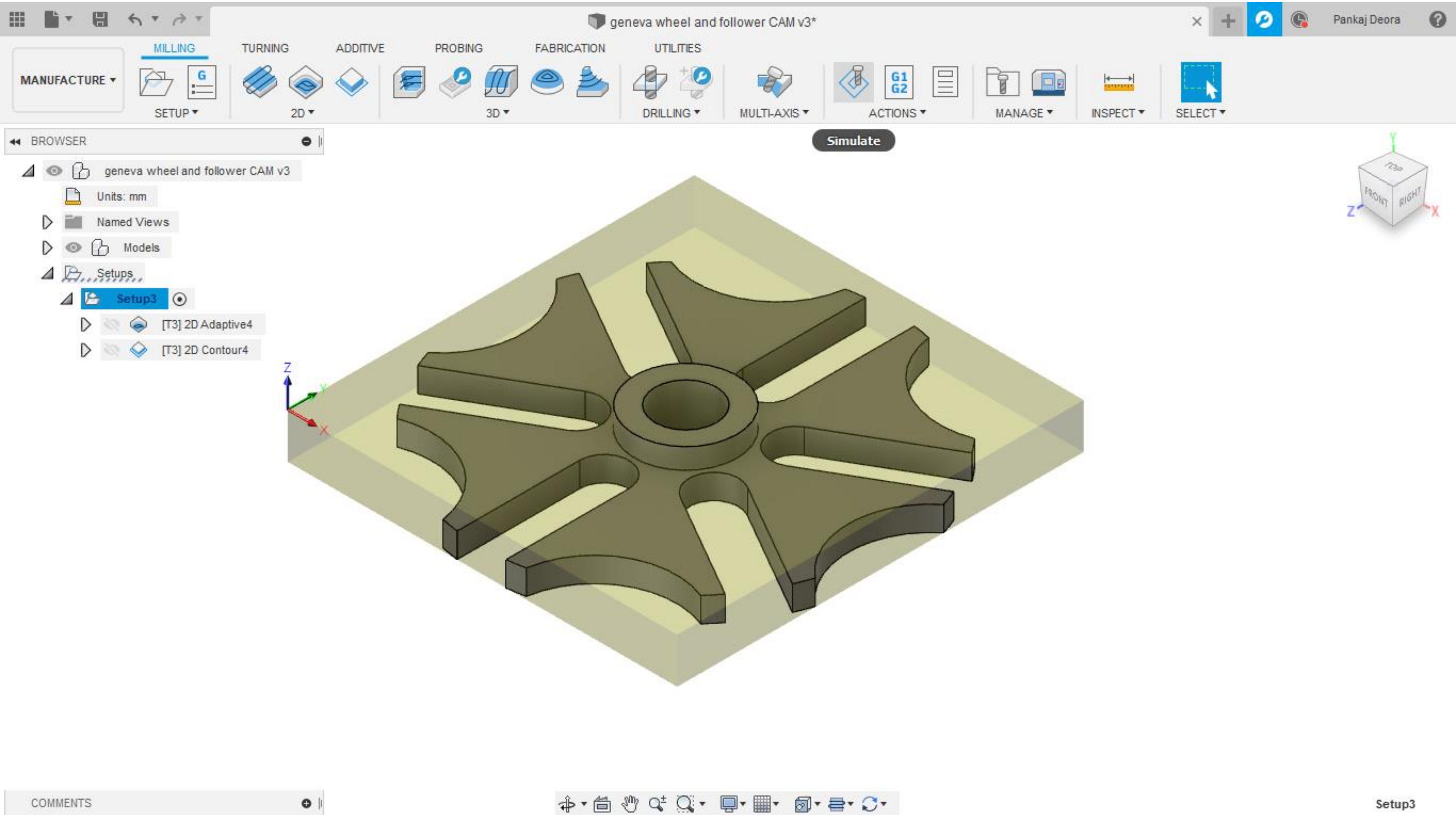
Select a point on the wheel as shown. This is the point from where the tool starts entering into the workpiece. Click “OK”



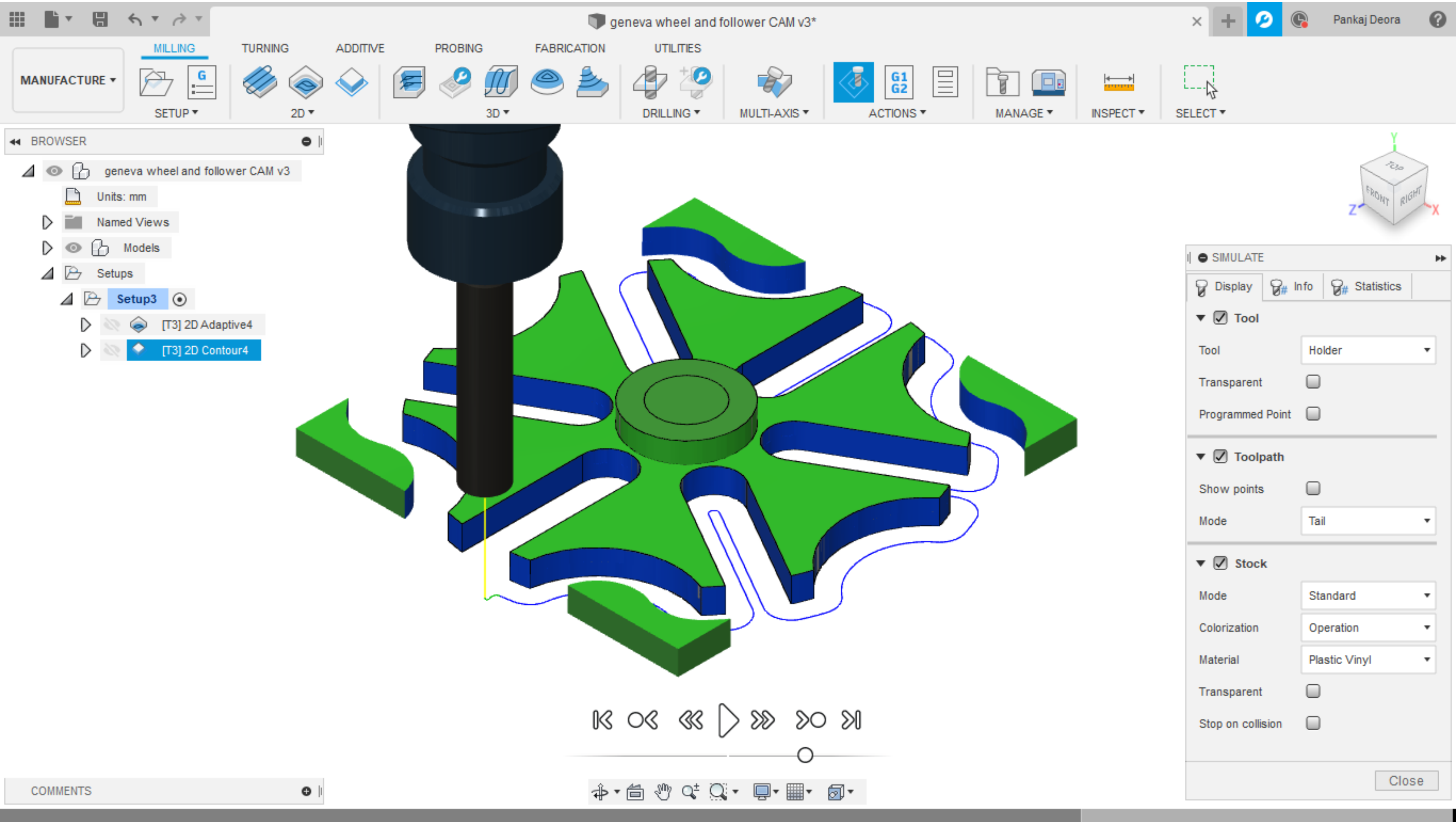
Click on the Setup as shown



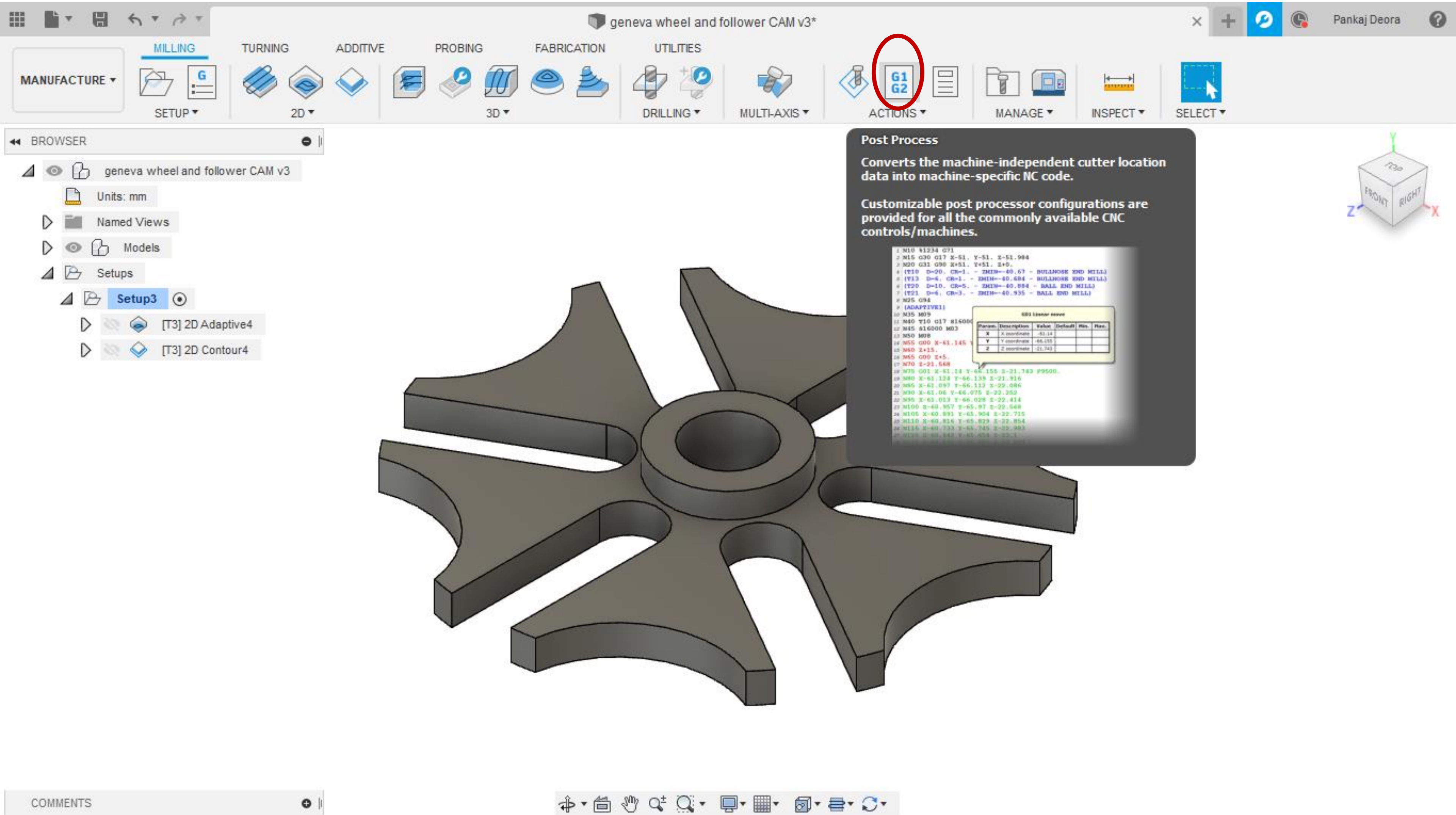
Start the simulation



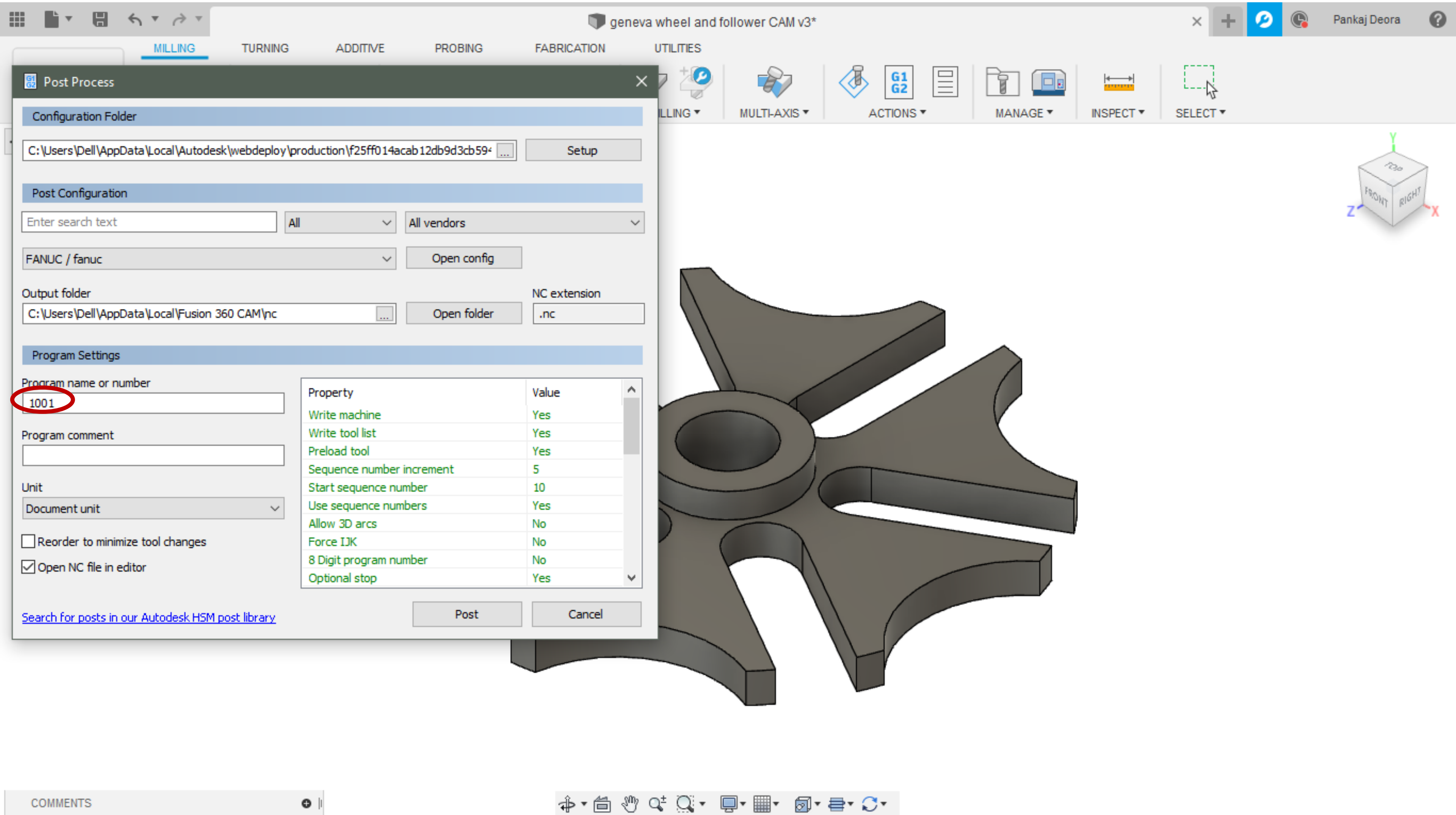
You can see the final part generated after cut. Now generate the G Code for the part manufactured.



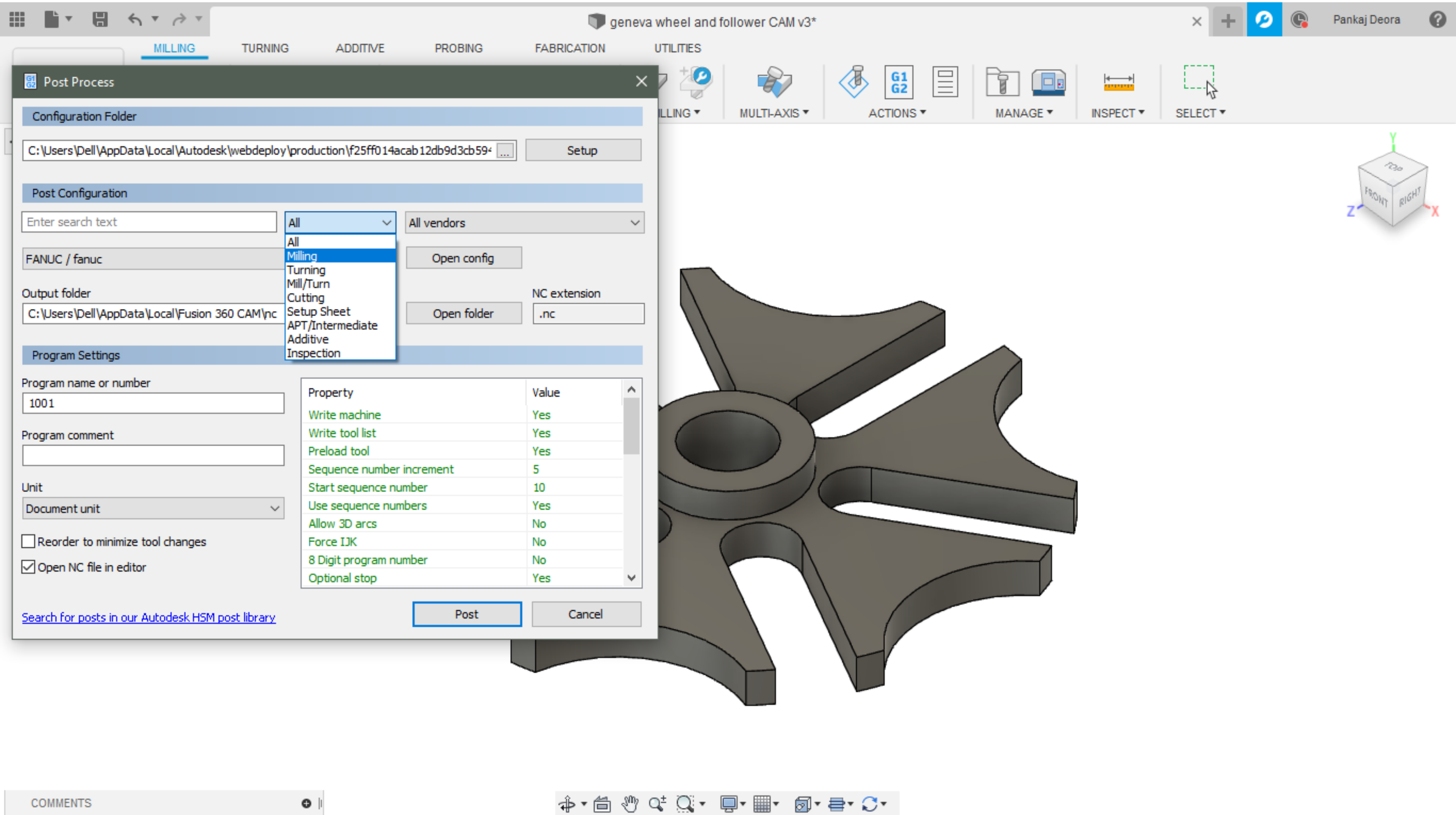
Click on "Post Process" tab as shown



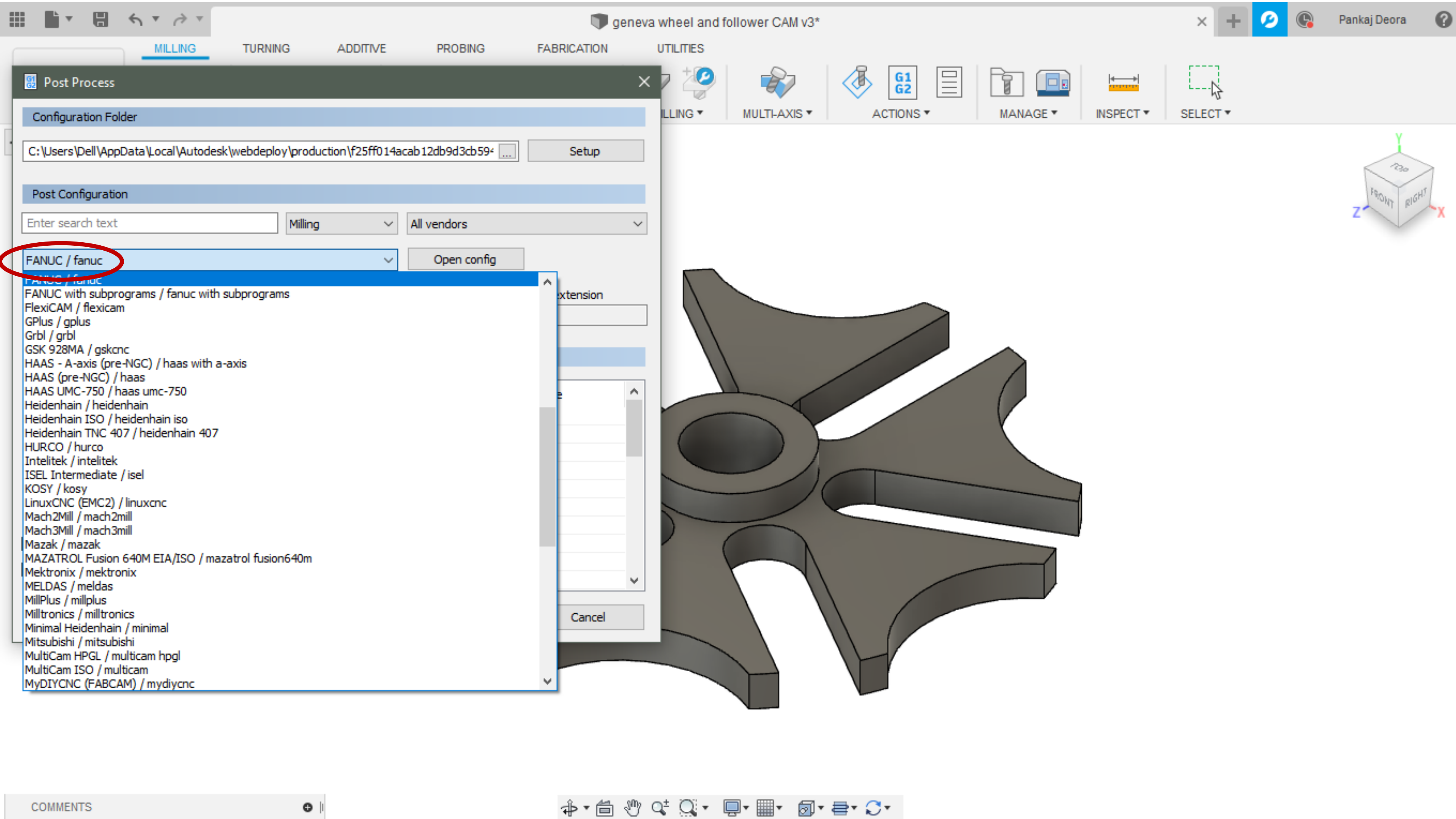
In the new window that follow, give a program number as shown. Here it is given ‘1001’



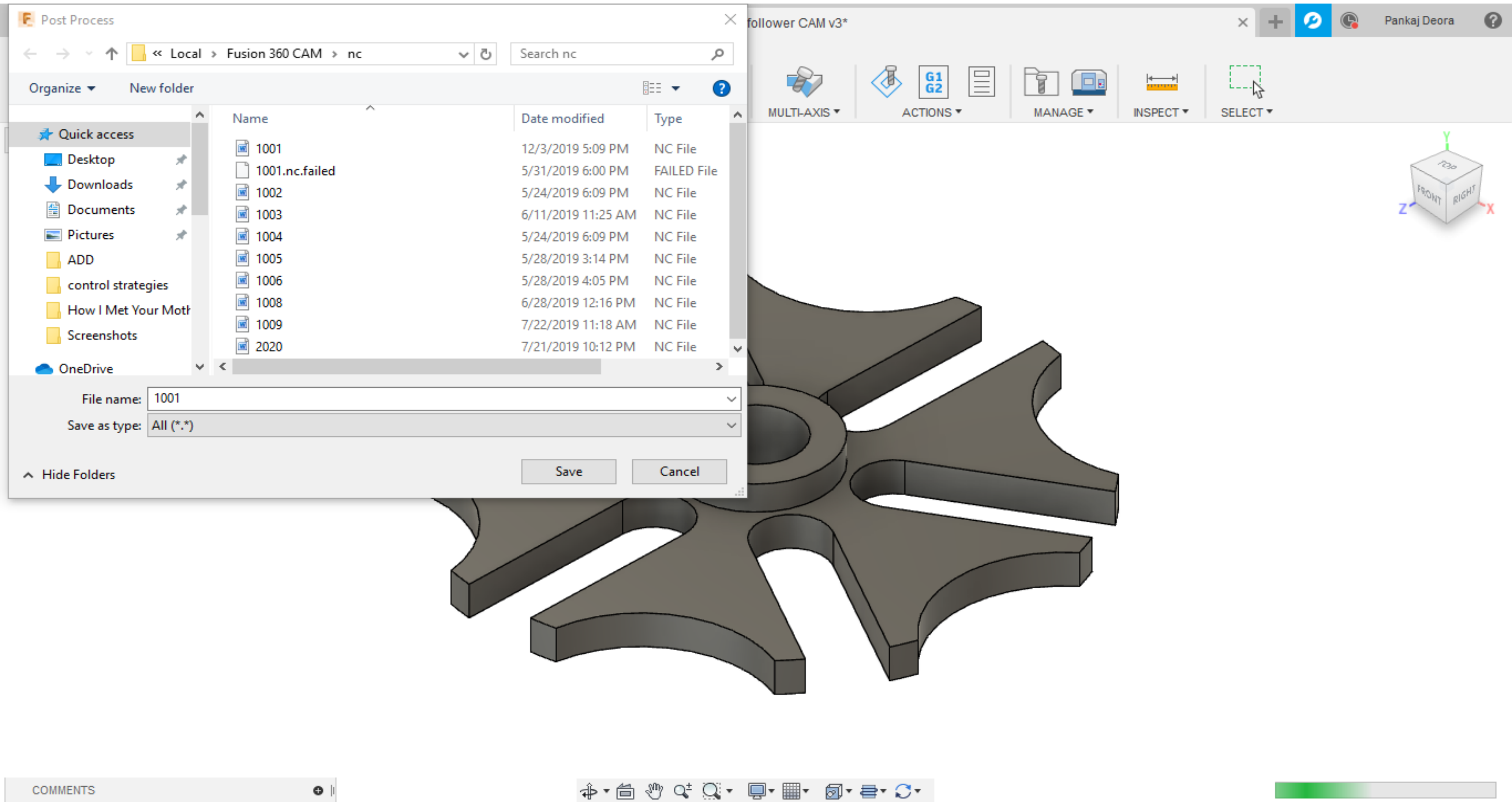
Select “Milling ” as shown



Select a post processor. For our purpose, we select "Fanuc" as shown. Click "Post"



Save the code. To open the code generated, go to the folder where you saved the code and right click on the file and click on ‘open with’ and select Notepad or MS Word.



Show the G code you have generated to your guide/TA/Tutor. If you have generated the G code successfully, your exercise is complete.

Note that you will be expected to do similar things for your project, in which you must make one part on a CNC machine using a G code that you will generate using CAM

CAM Task B

Generate a G-Code for a part to be 3D Printed

Contributed by:

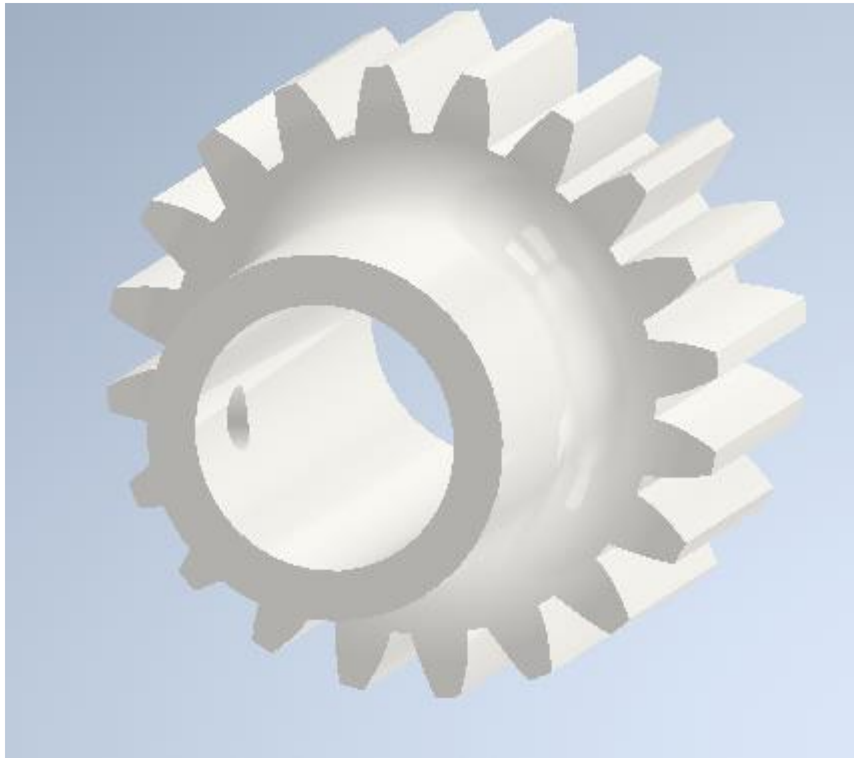
Srijan Bharati, Govind N Sahu, Pulkit Jain

Updated by: Harsh Singh Rajput and Yalamati Hari Charan

MadLab, IITK

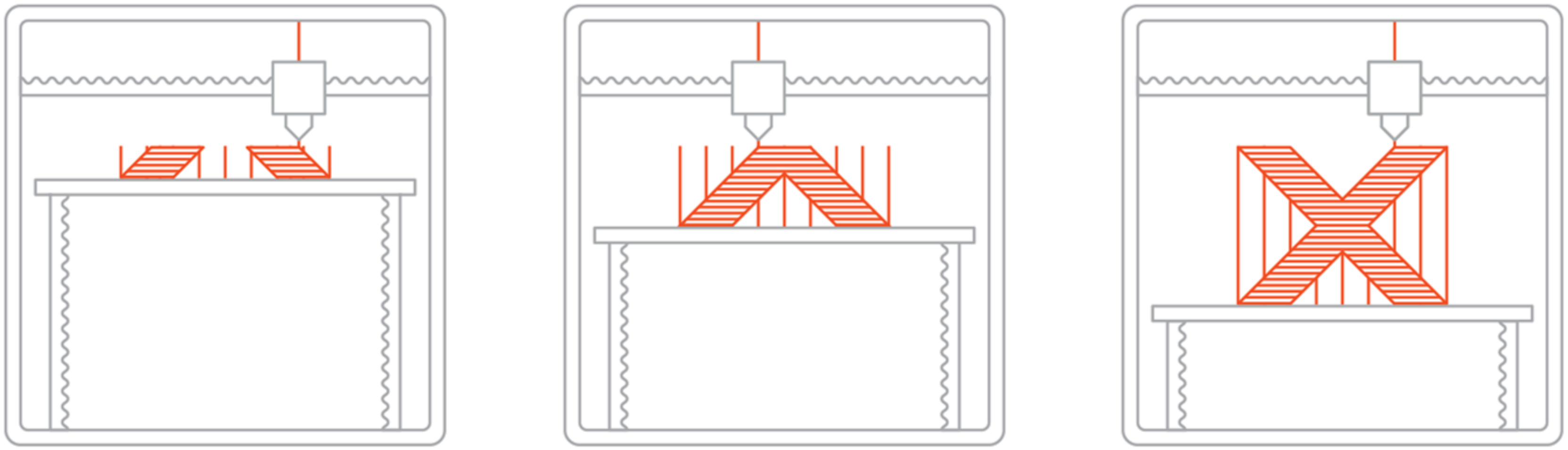
Objective

Generate a G-code to manufacture a gear using the 3D printing machine. CAD model for the part will be provided to you.

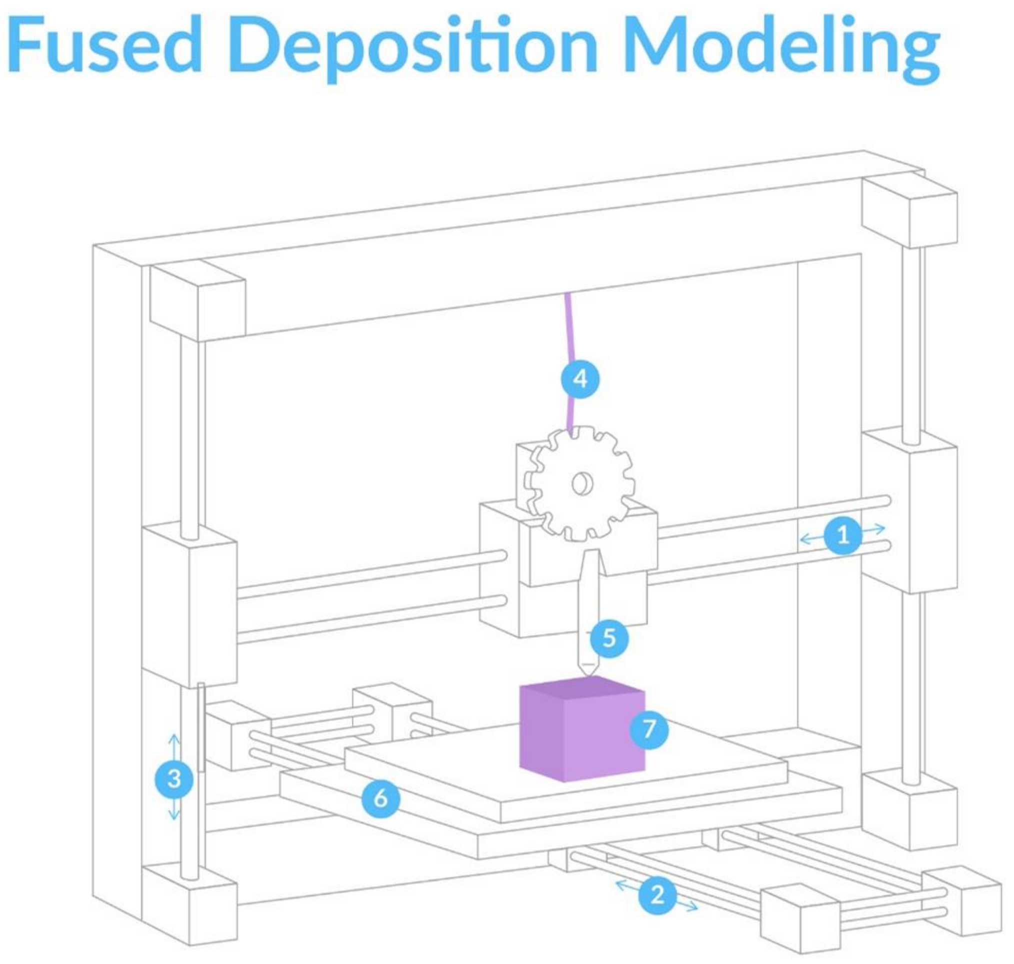


About the Printer and method of 3D printing

- We will use the Fused Deposition Modeling (FDM) method of rapid prototyping, i.e. 3D printing. There are other methods too. Those will be discussed in class.
- The printer is a cartesian Printer (x-y-z)



About the printer

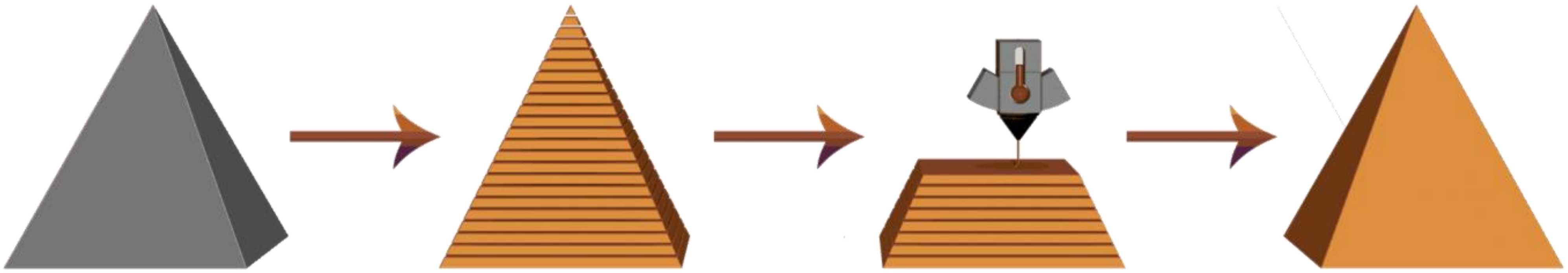


Cartesian Printer

1. X Direction
2. Y Direction
3. Z Direction
4. Filament
5. Extrusion Nozzle
6. Heated Platform
7. Printed Part

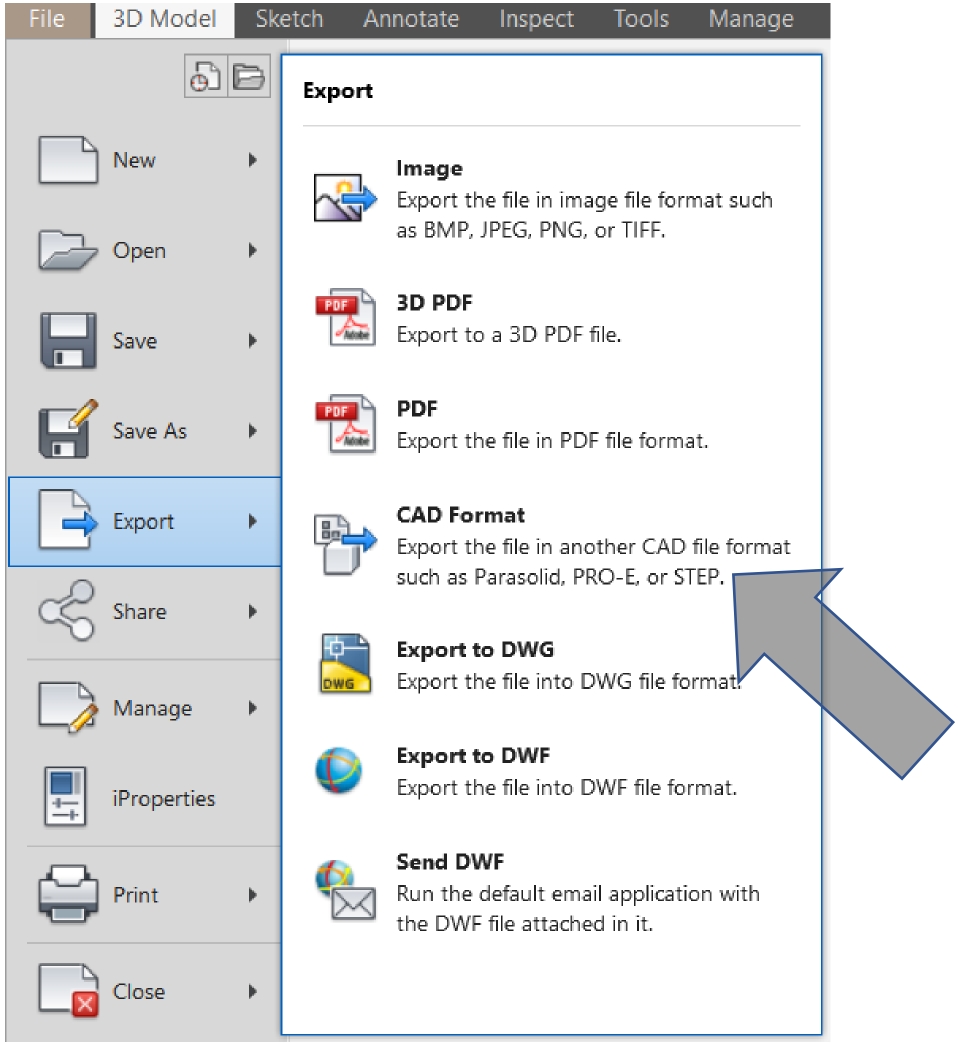
Printing Procedure

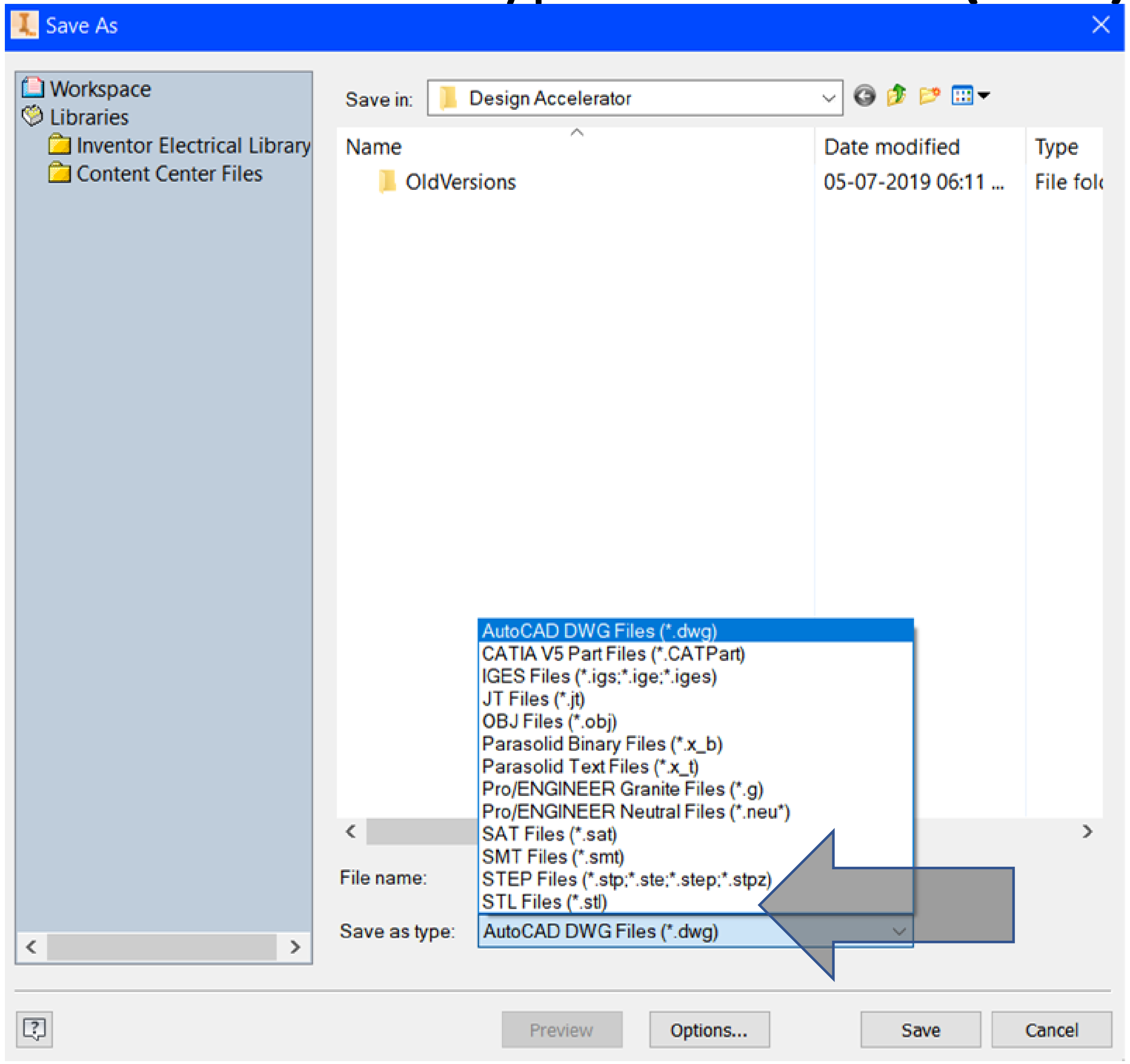
CAD file	Slicing to G code	3D Printing	Product



At first, you must export the CAD model in a neutral format

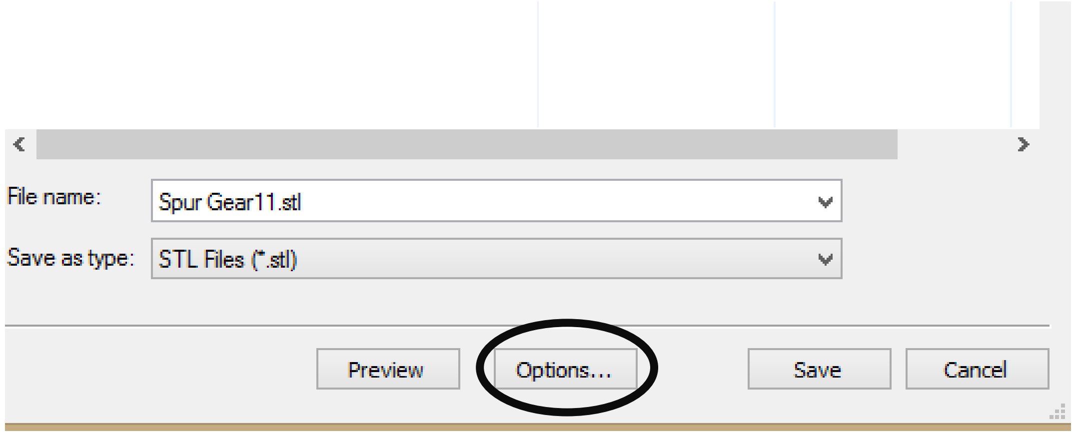
- Go to file□ **Export □ CAD Format** and select file type as **STL files (*.stl)**

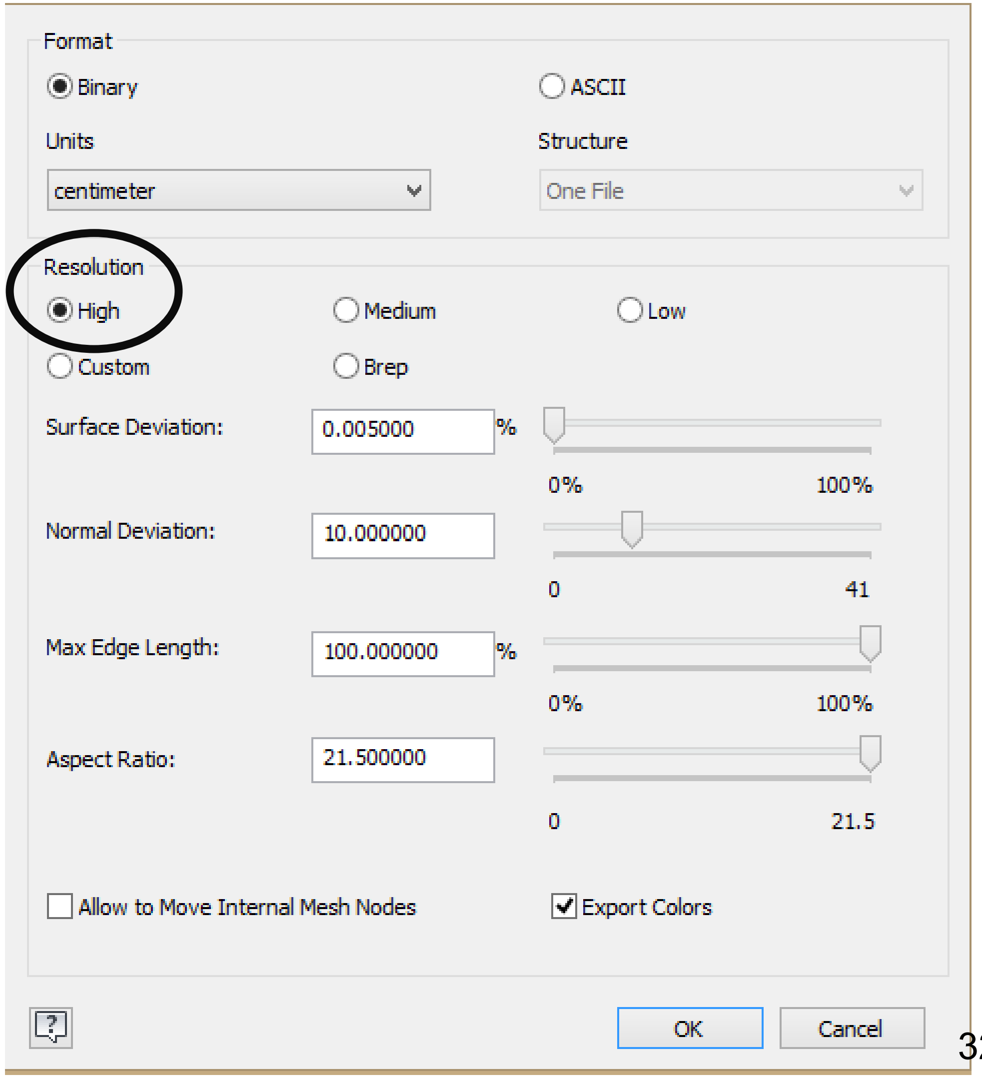




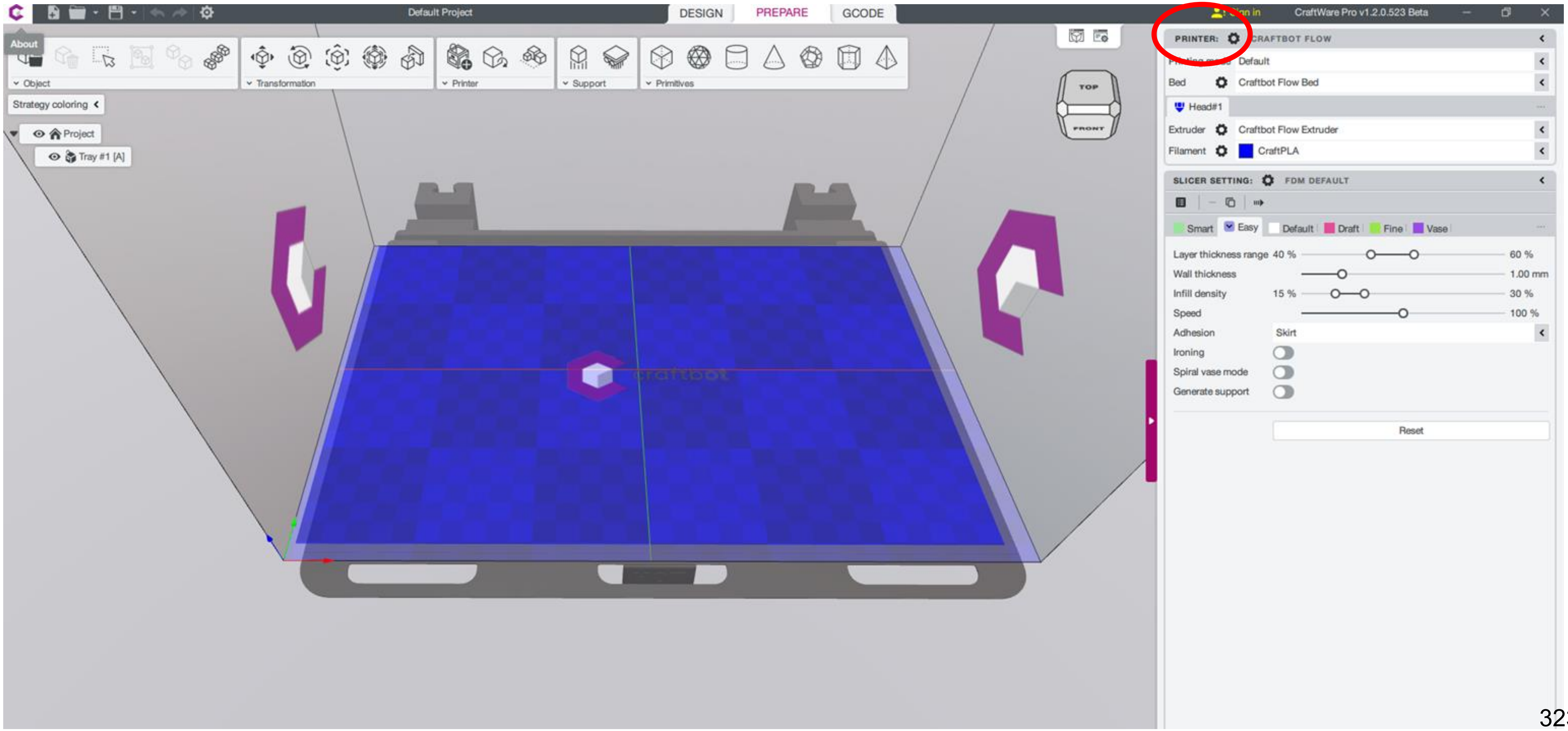
Exporting the CAD model

- After selecting **file** type click on **options** to save the file with high resolution.
- Select resolution as **high** and save the file
- A high resolution file makes for a better printed product



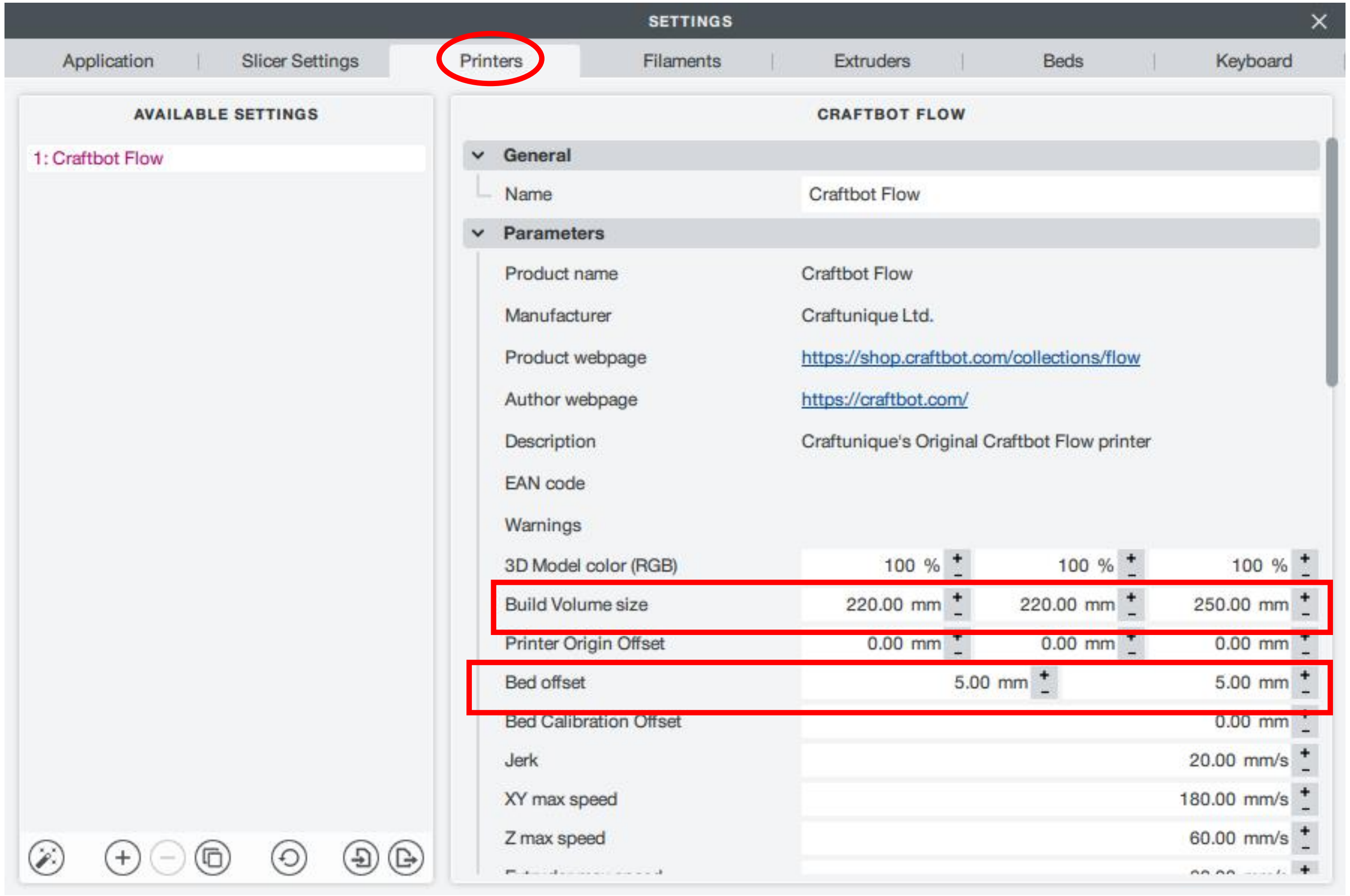


Open the Craftware software and click on ‘Printer’ settings to set the printer properties in the software.

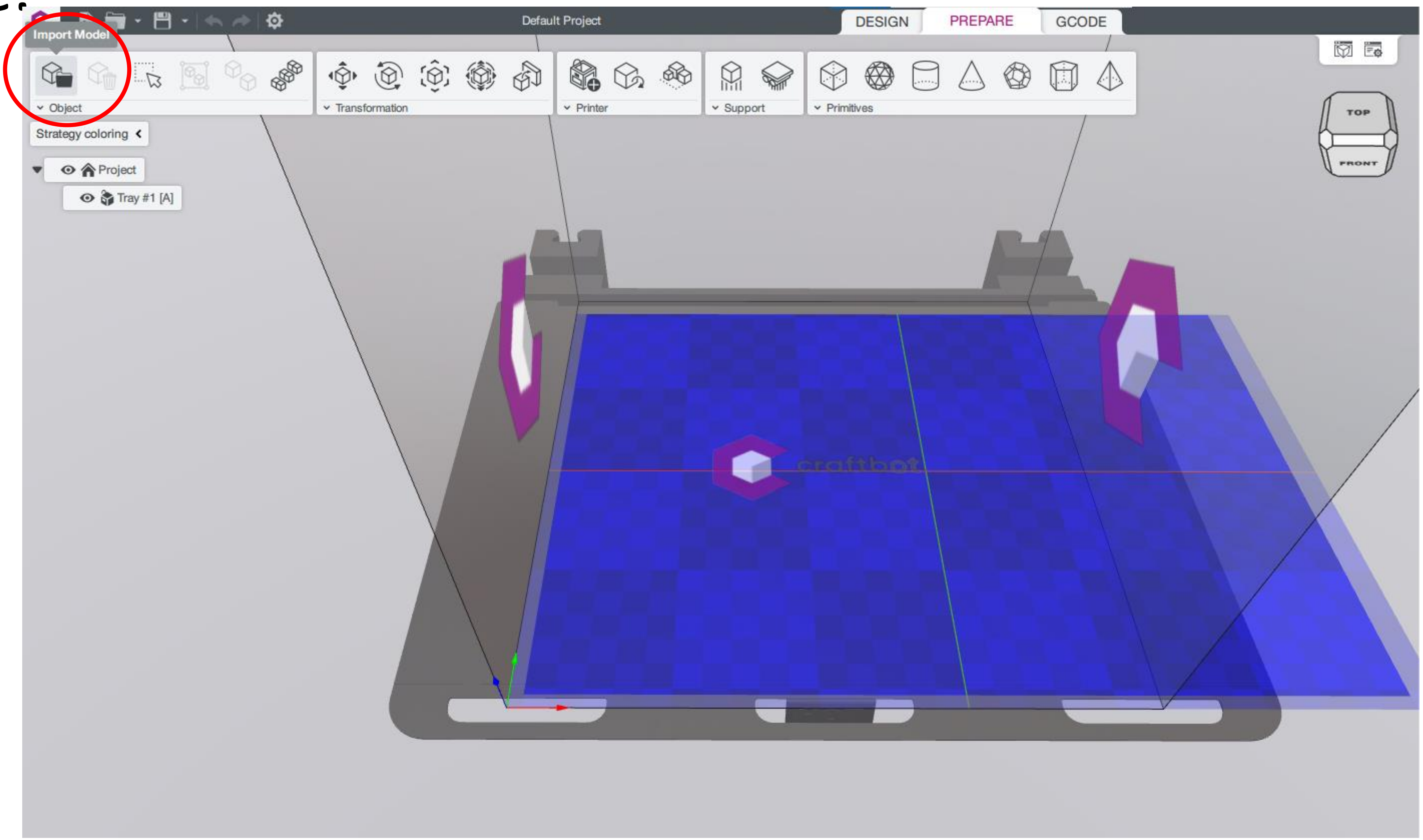


The procedure to slice to generate a G-code involves many steps. Start with:

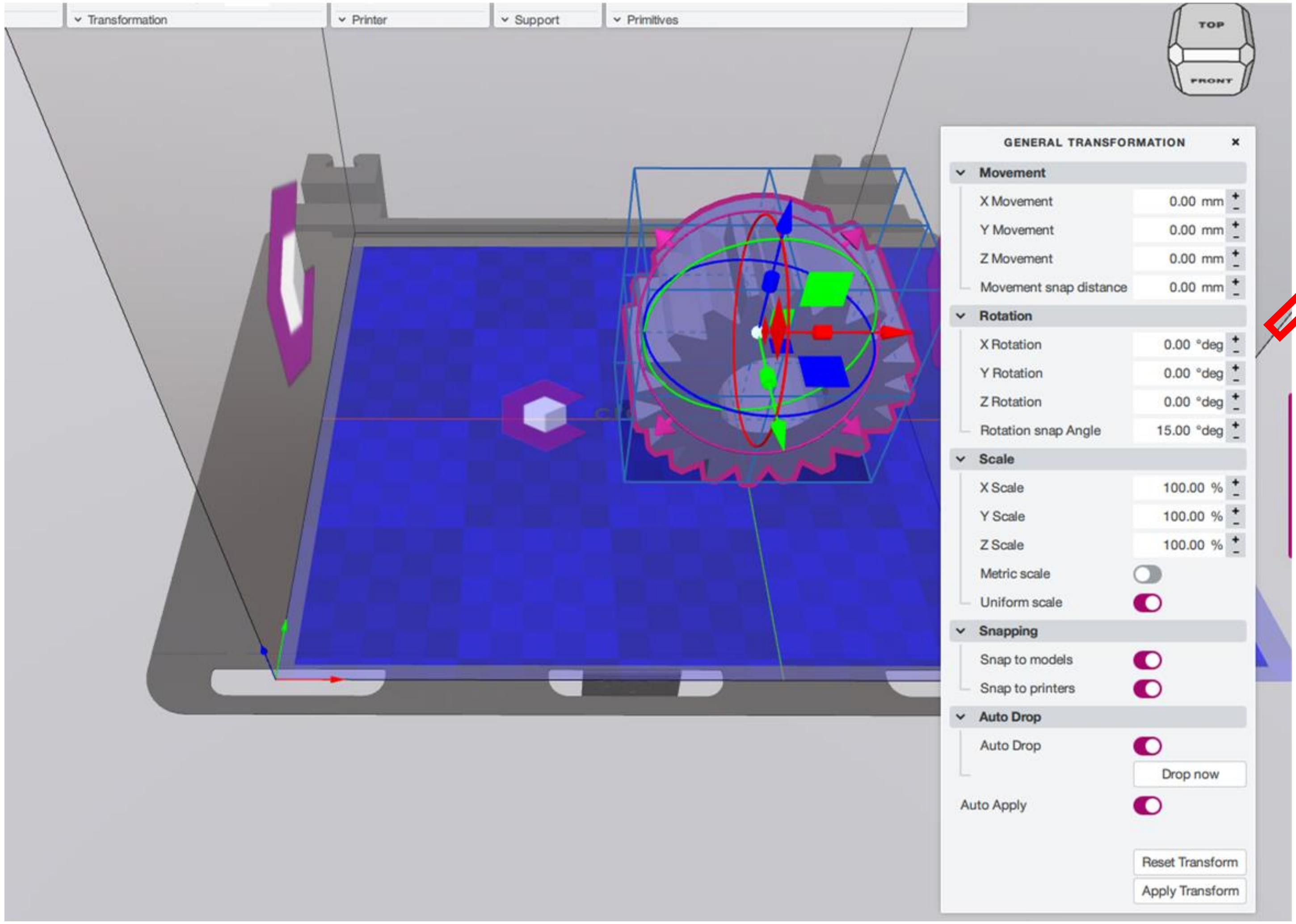
- Go to the **Printer** in the popup window and put the following properties of printer we will be using – the same one you assembled.
- Width (X) = 220 mm;
- Depth (Y) = 220 mm;
- Height (Z) = 250 mm;
- Offset (X) = 5mm;
- Offset (Y) = 5mm;
- Close this pop-up box



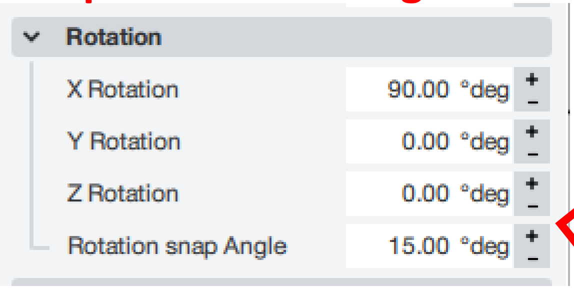
Next, click on ‘Import Model’ to import the saved .stl file.

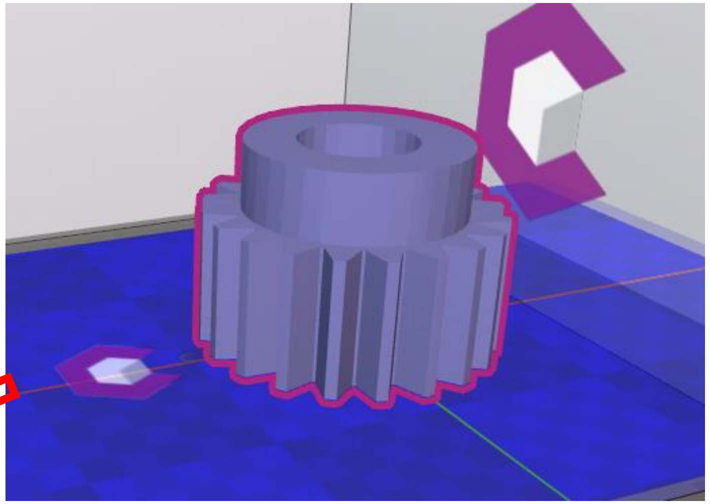


Next, Orient the gear using the ‘Rotation’ settings. We would like that we build with the base down (See preferred orientation). Then click ‘Drop now’ and ‘ Apply Transform’.



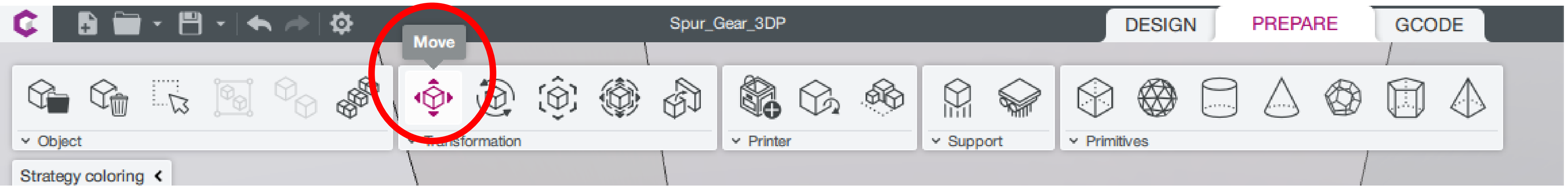
Step 1 : Rotate the gear to bring base down

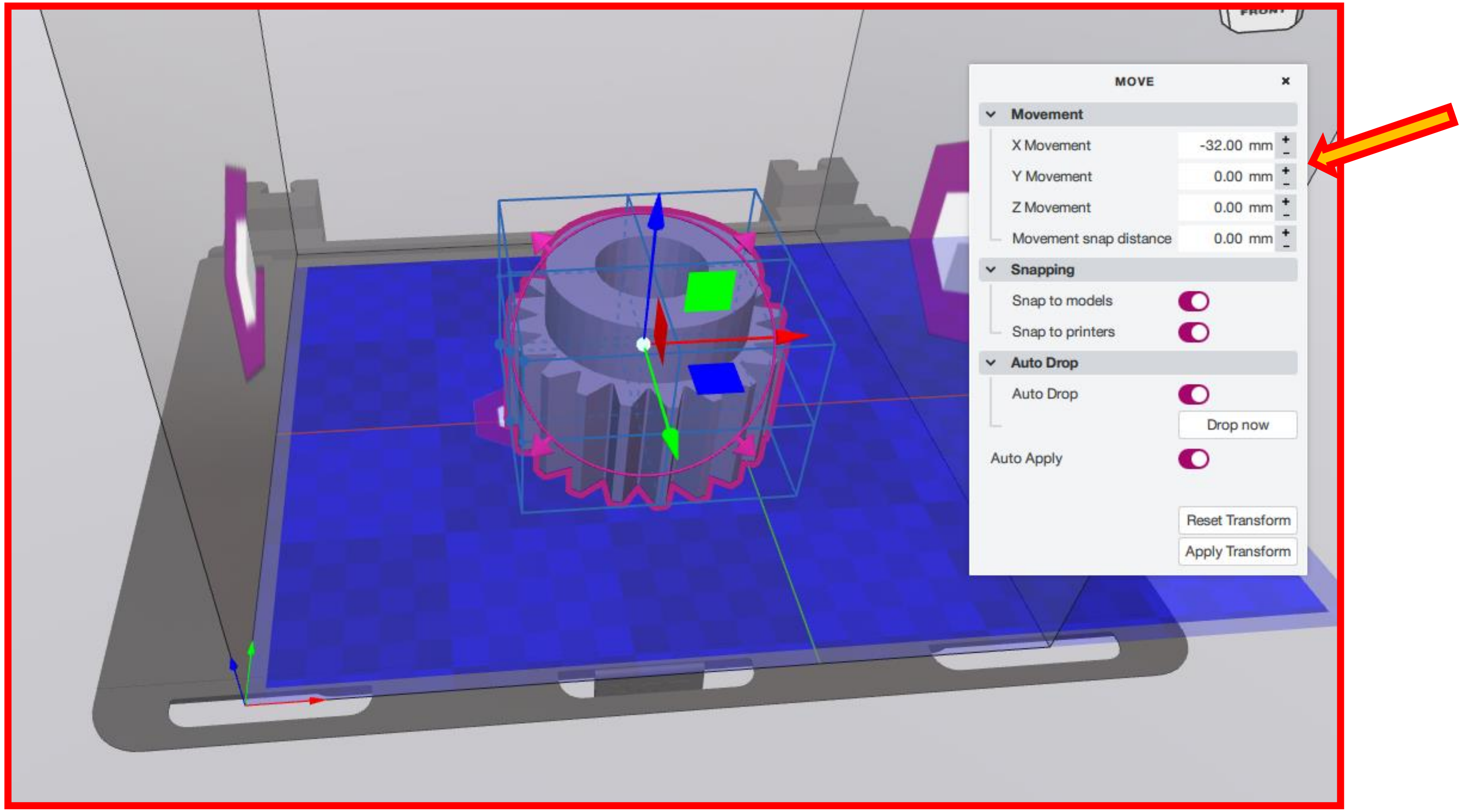




Step 2 : Click ‘Drop now’ and ‘Apply Transform’

Next, Click on the ‘Move’ button to move the part in the middle (approximately). To do this, you can select any axis and move the part in that particular direction. Finally click ‘Apply Transform’

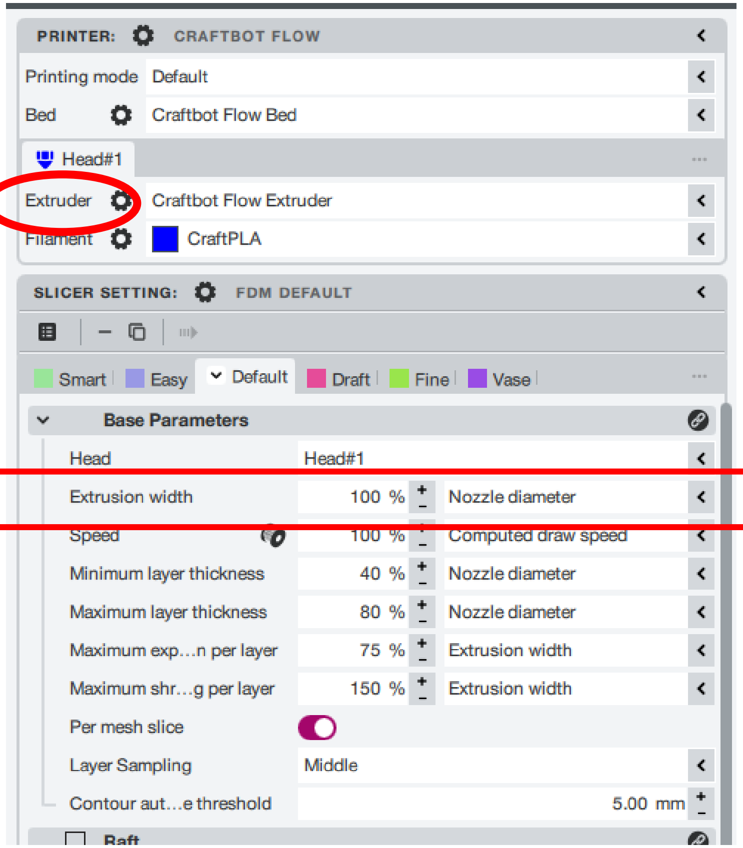


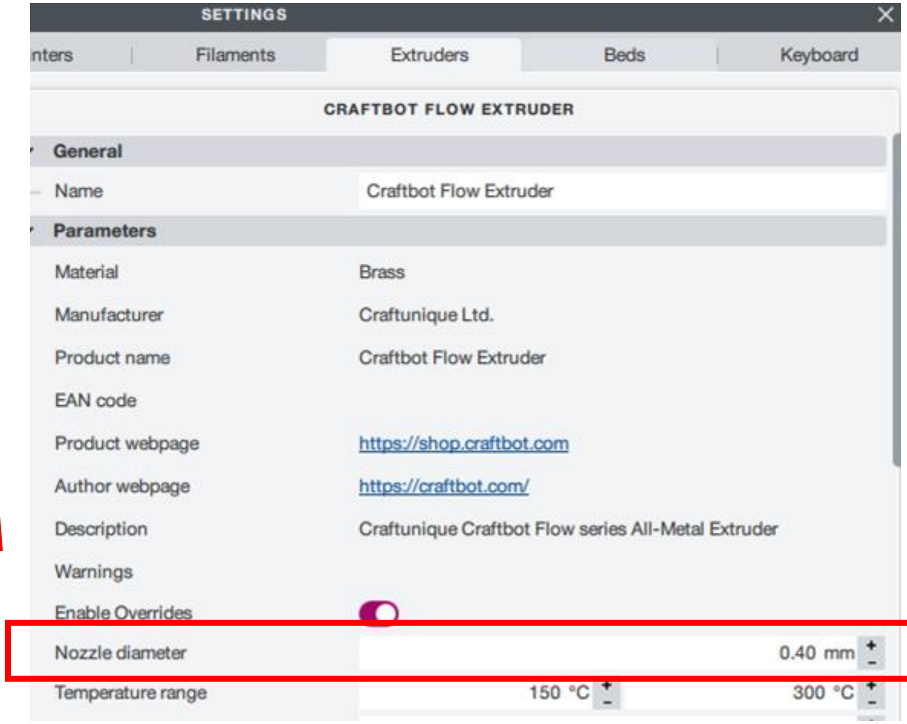


Step-1 : Go to “Extruder” settings. Set the nozzle diameter to 0.4 mm (default case)

Step-1

Step-2

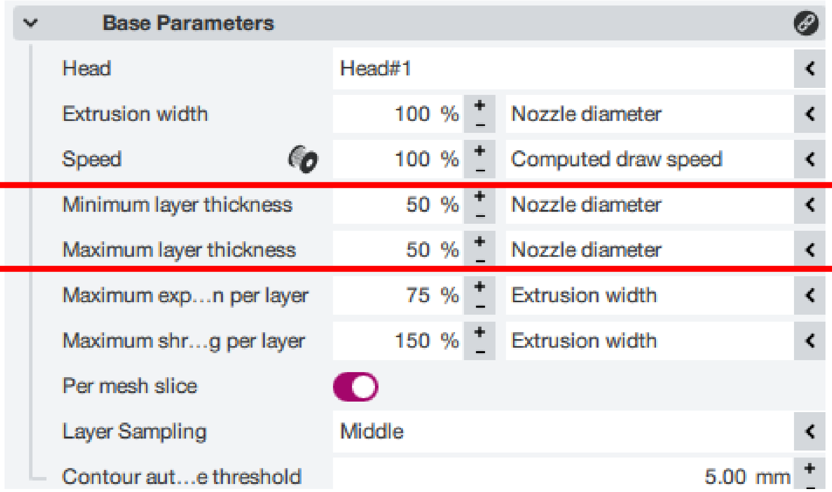




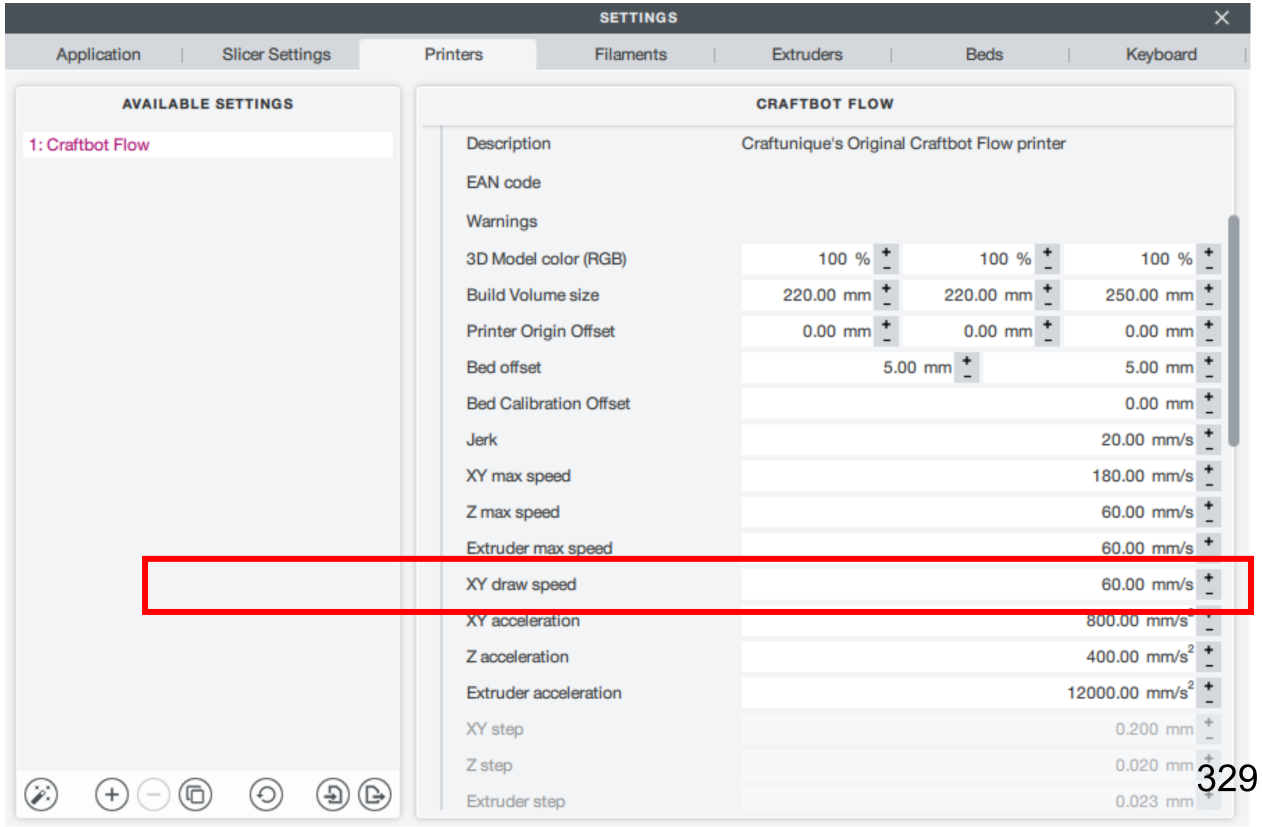
Step-1

Step-2 : In the “Slicer Settings”, go to “Default” tab. Set the ‘Extrusion width’ to 100%. This is determined by the nozzle diameter on the machine you will use, which has a 0.4 mm opening.

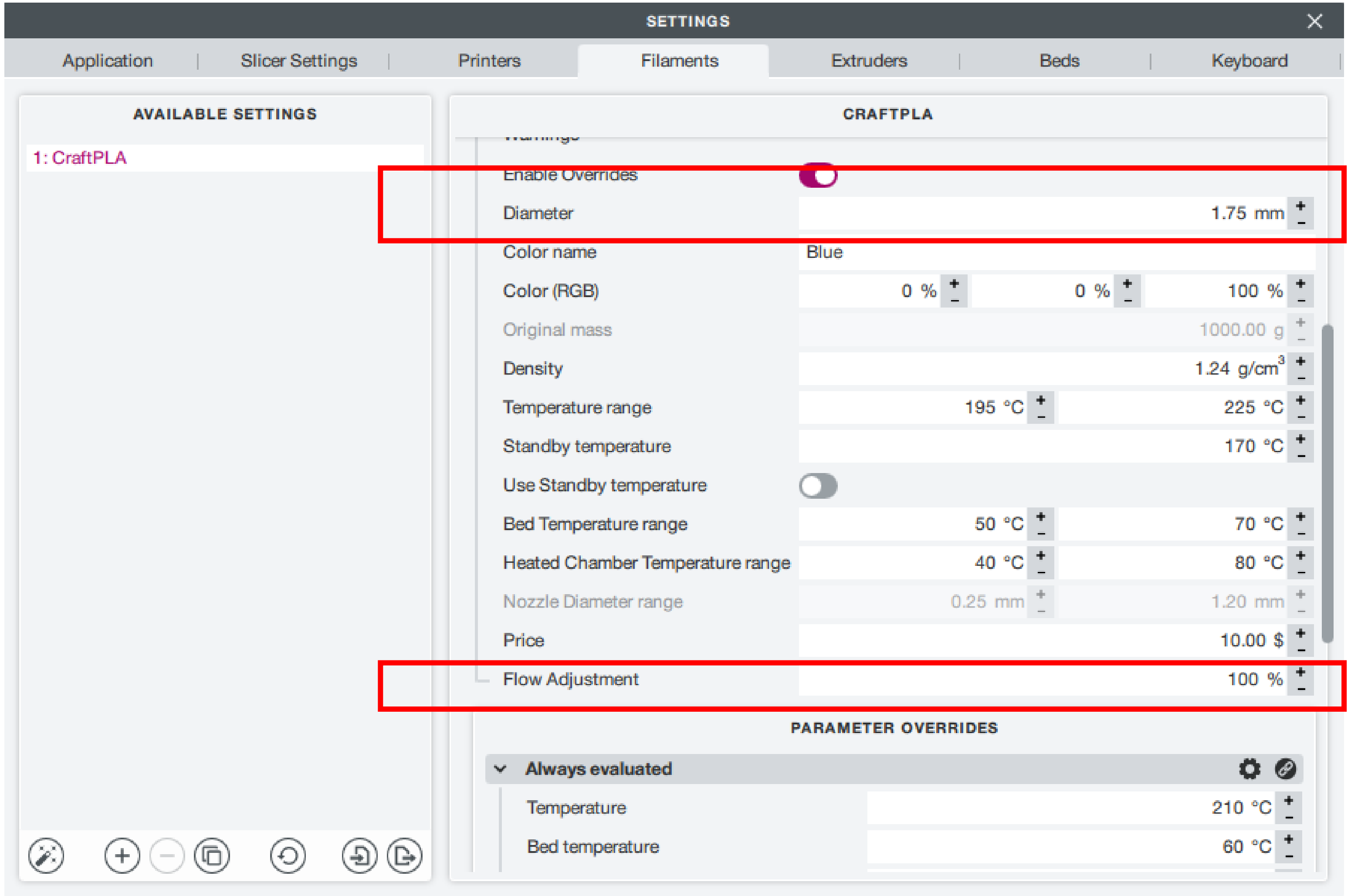
- Layer Thickness is the Z axis resolution and defines thickness of each layer. The finer this layer thickness, the better the printed product, but more the time it will take. We need to set this value to be 0.2 mm. To do this, set 'Minimum layer thickness' and 'Maximum layer thickness' as 50% of Nozzle diameter.



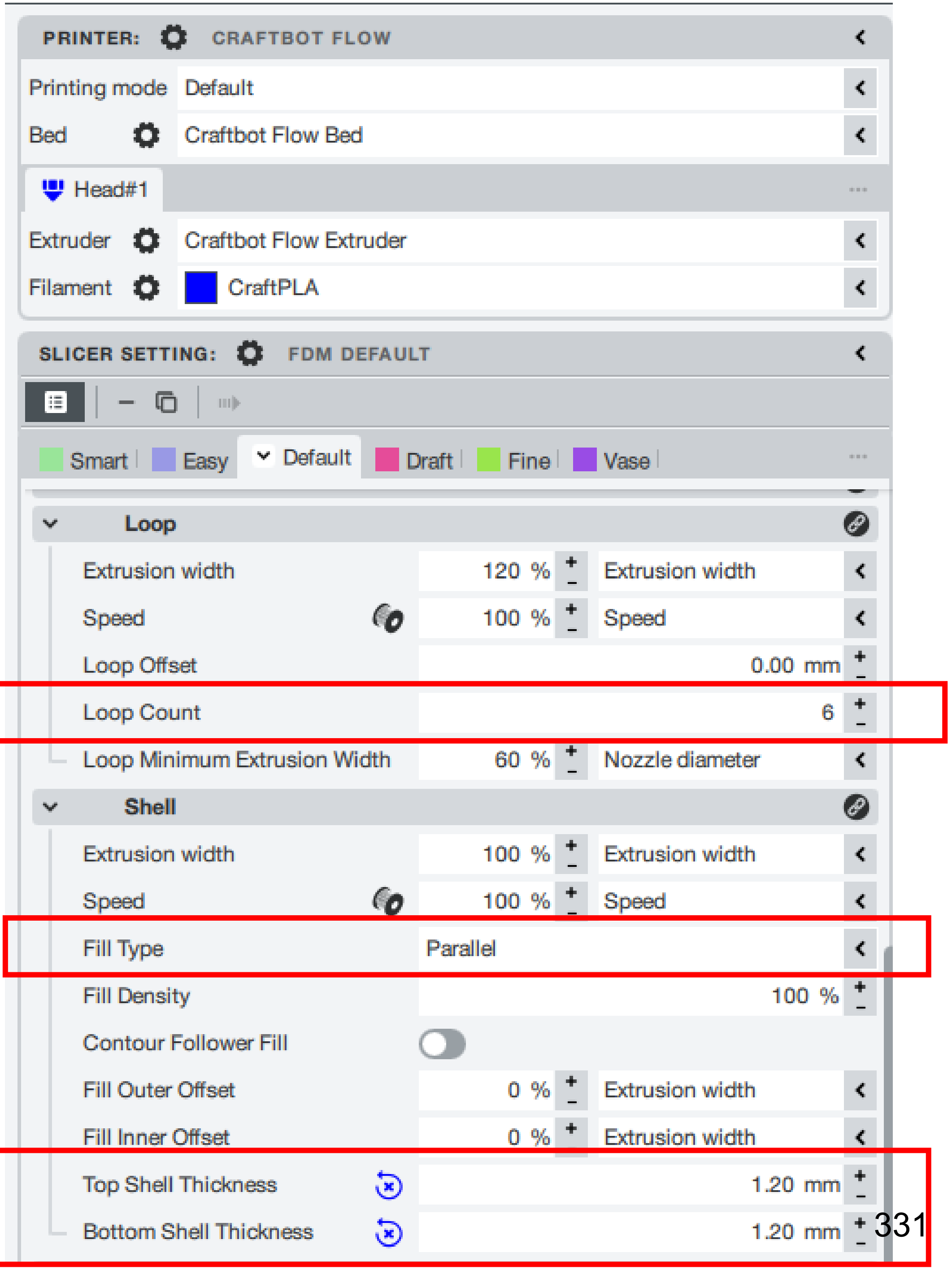
- Again go to 'Printer' settings to set draw speed. Draw speed controls the speed at which the filament is drawn from the spool. This also governs the printing speed and quality. Set XY draw speed value to be 60 mm/s.



- Now go to ‘Filaments’ tab.
- Filament diameter helps software to calculate material flow. We will be using a filament of diameter 1.75 mm. Set this value.
- Keep the ‘flow adjustment’ at 100 %. If underflow or over flow occurs, flow adjust is used to correct it



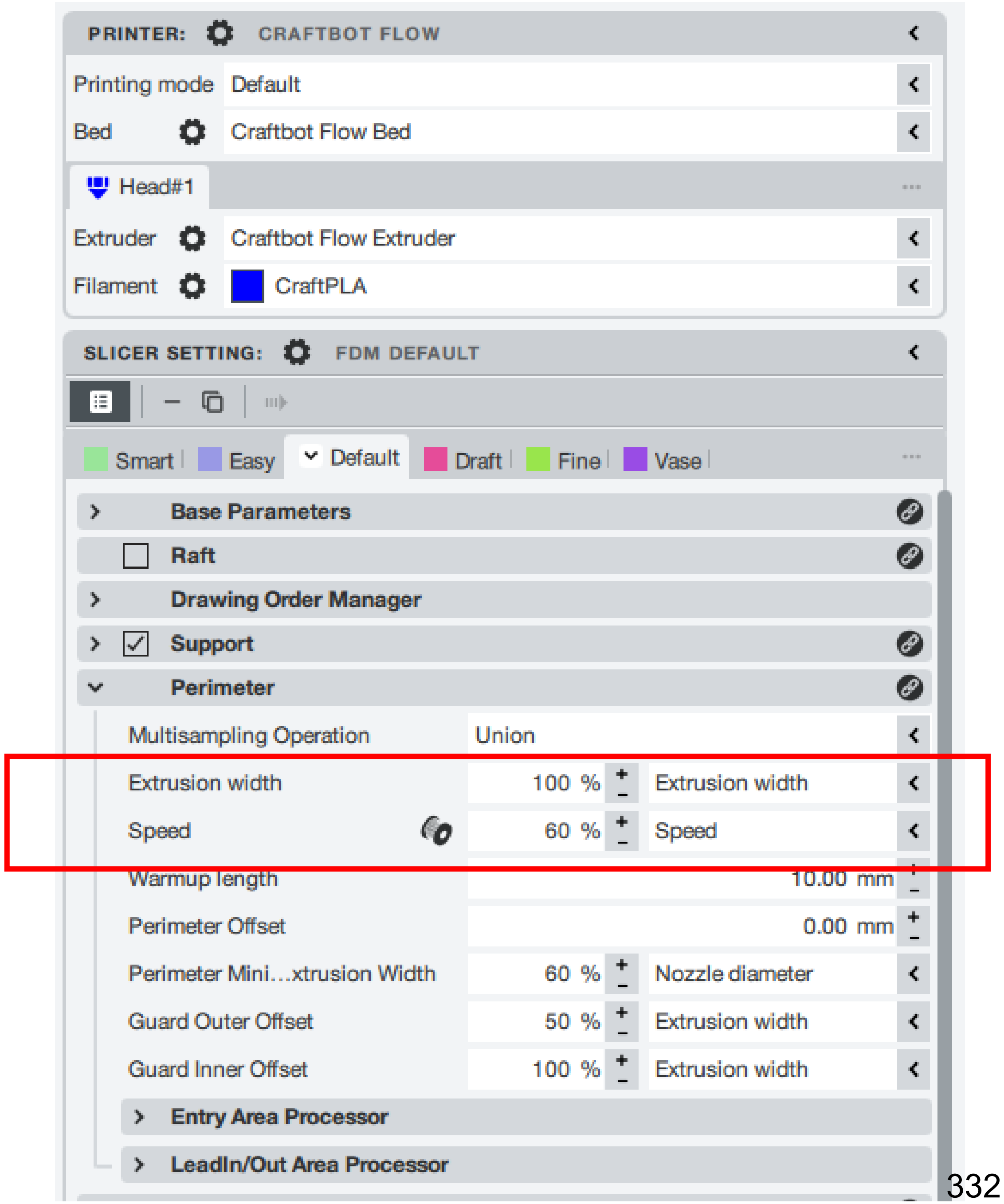
Now we have to set the properties of the vertical layer at the boundary of the part and other settings. Go to 'Slicer Settings' and then 'Default' tab.



- Go to 'Loop' section and set the 'Loop count = 6 loops'
- In the 'Shell' section, set 'Fill Type' as "Parallel"
- In the 'Shell' section, Top Shell Thickness = 1.20 mm, Bottom Shell Thickness = 1.20 mm

In the ‘Slicer Settings’ and ‘Default’ tab only, go to ‘Perimeter Section’ now.

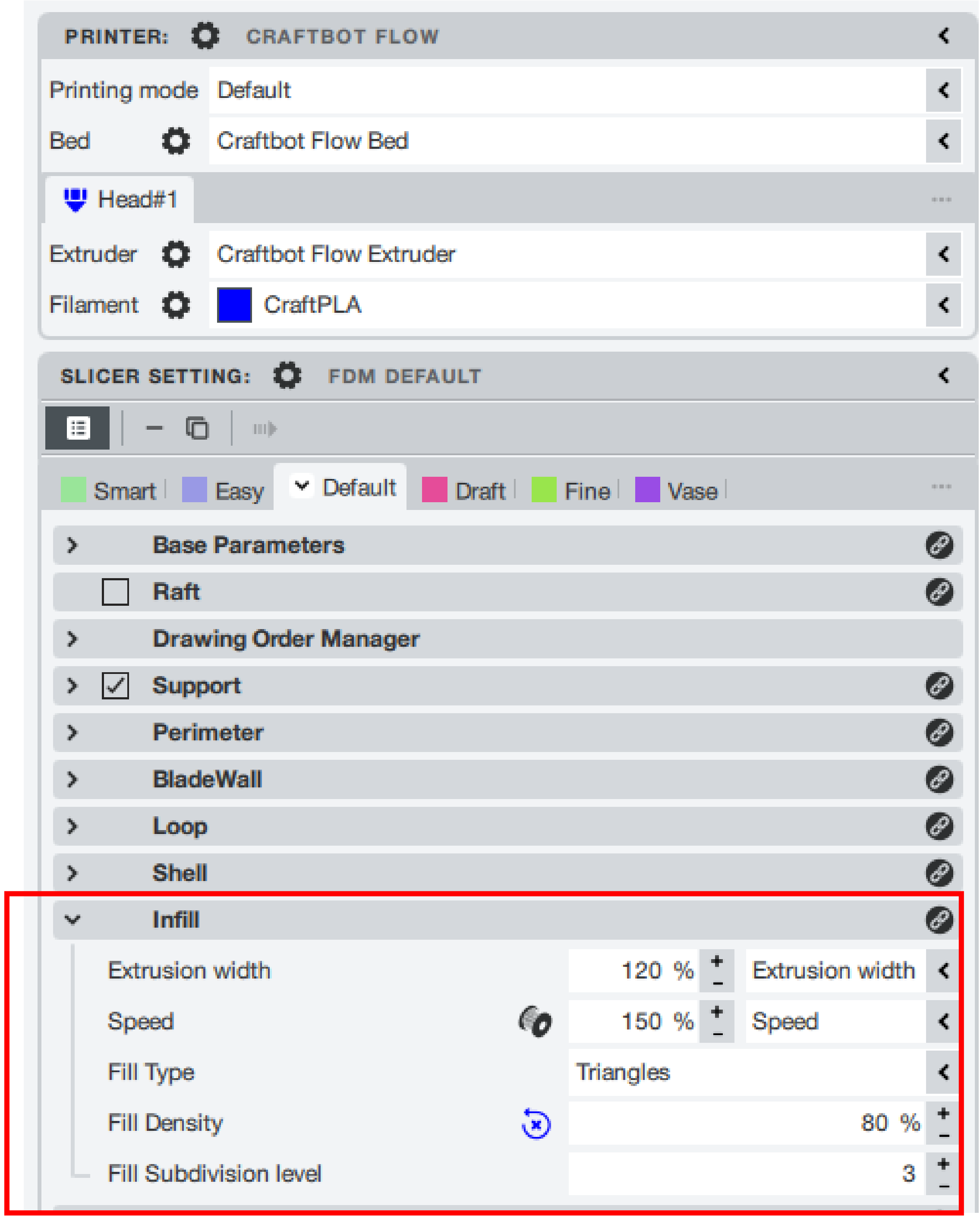
- Set ‘Extrusion Width’ as 100%.
- Set ‘Speed’ as 60%.



In the ‘Slicer Settings’ and ‘Default’ tab only, go to ‘Infill Section’ now.

- Set ‘Extrusion Width’ as 120%.
- Set ‘Speed’ as 150%.
- Set ‘Fill Type’ as ‘Triangles’.
- Set ‘Fill Density’ as 80%.

Leave all the other parameters unchanged.



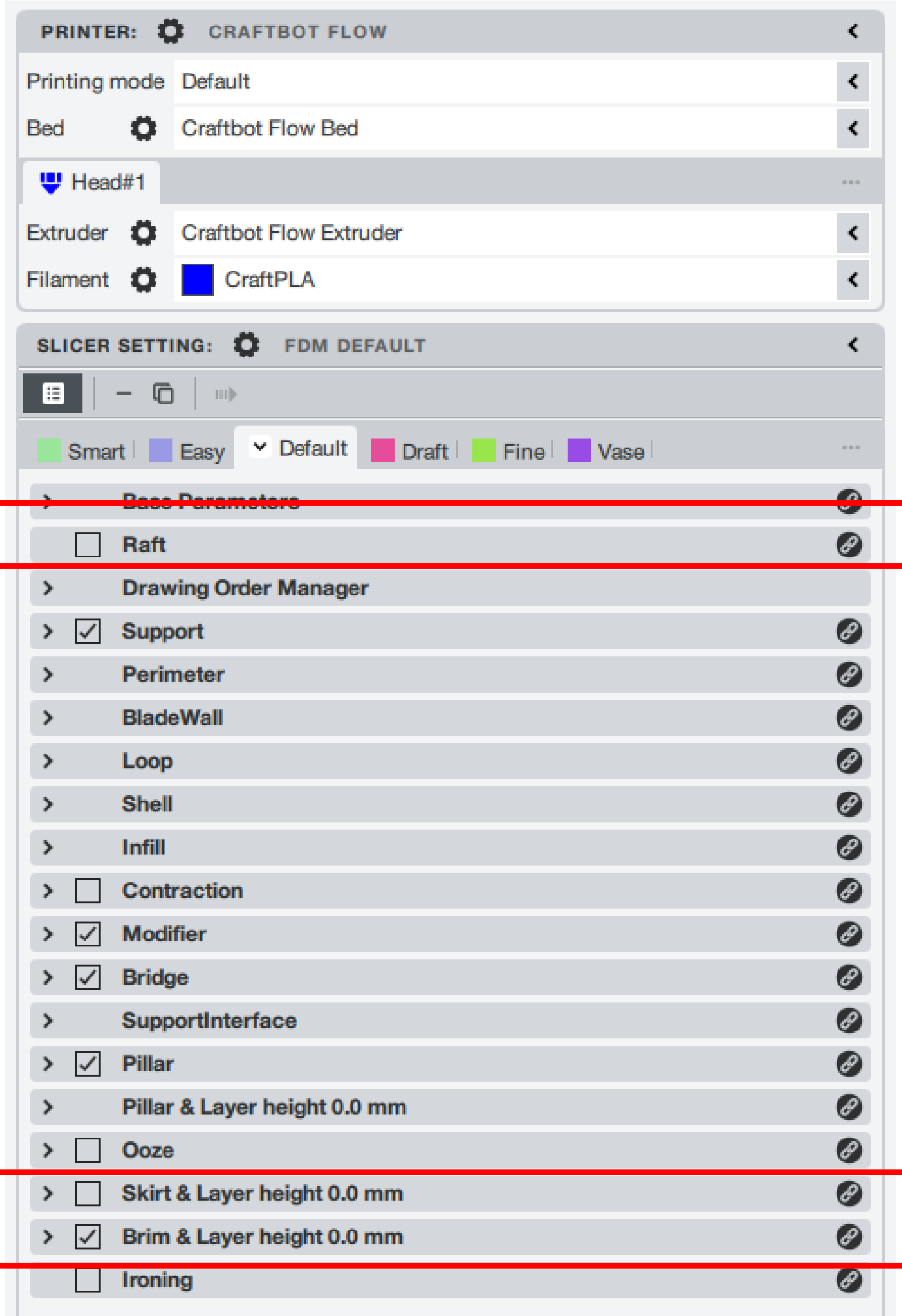
Look at the 'Slicer Settings' -> 'Default' tab sections (as shown in right image)

Select checkbox for 'Brim & Layer height 0.0 mm'.

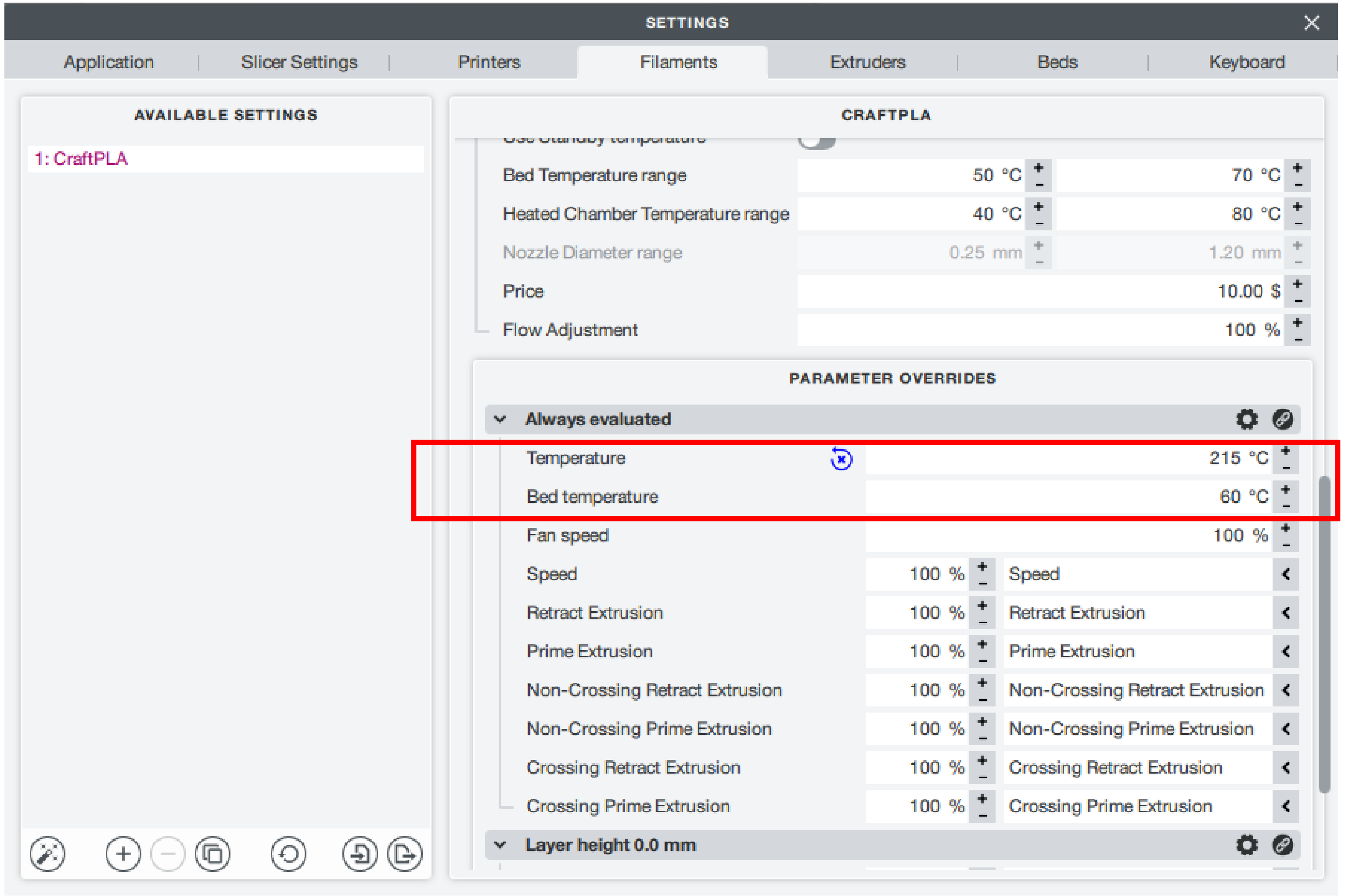
Note that 'Raft' and 'Skirt & Layer height 0.0 mm' checkboxes should NOT be selected.

<u>Types</u>

- Brim – Creates few loops at a distance from the main object and is useful to determine levelling problems.
- Skirt – Creates an extra layer connected around the perimeter of the main object which helps better sticking and prevents warping when print cools down.
- Raft – Creates a support platform on which the main object is printed. Raft is used to prevent warping, levelling problems and unsticking of the print but consumes more material.

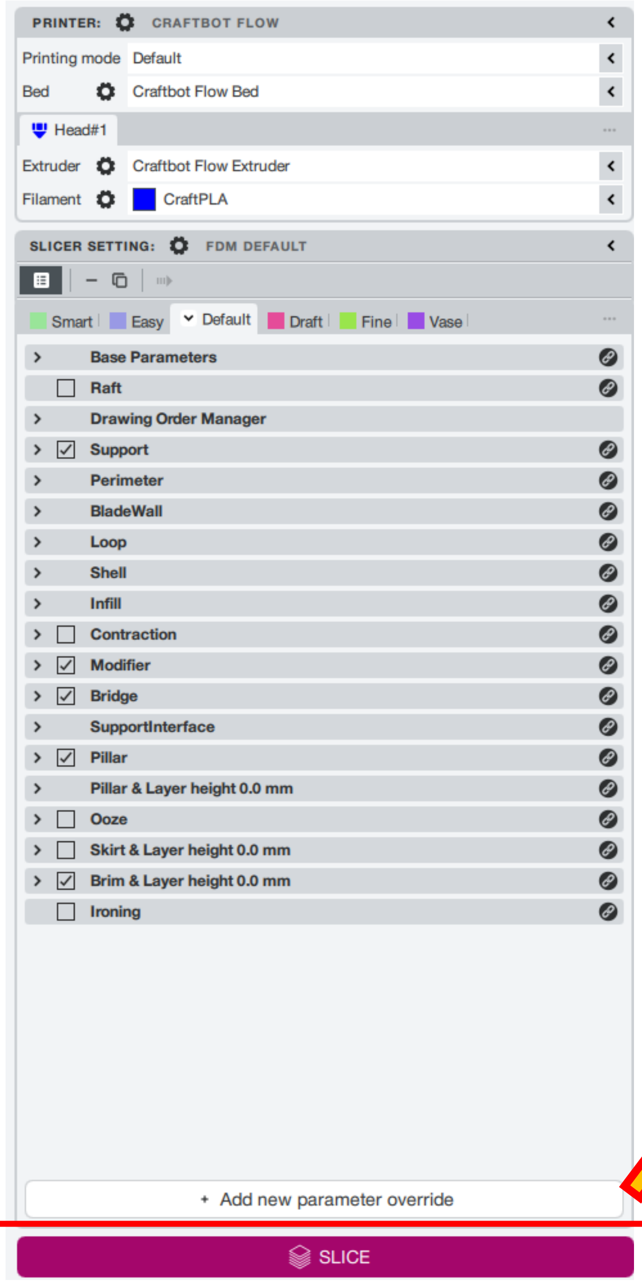


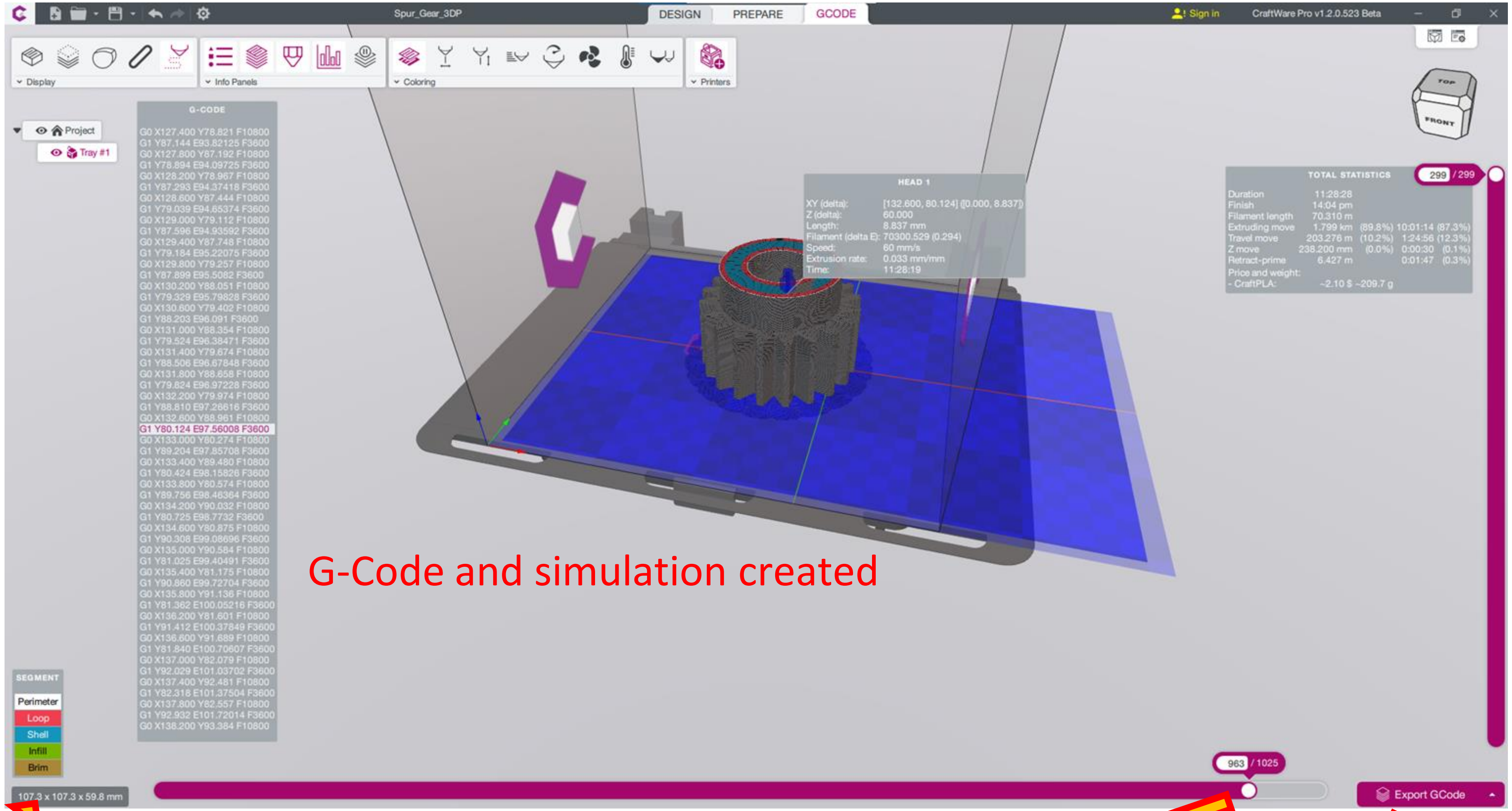
Next, again go to ‘Filament’ settings -> ‘Parameter Overrides’ section



- Set ‘Temperature’ as 215 degree C. (This is the Head Temperature)
- Set ‘Bed Temperature’ as 60 degree C.

If you've done everything correctly, you may now click on 'Slice'. This will create G-Code and printing simulation.

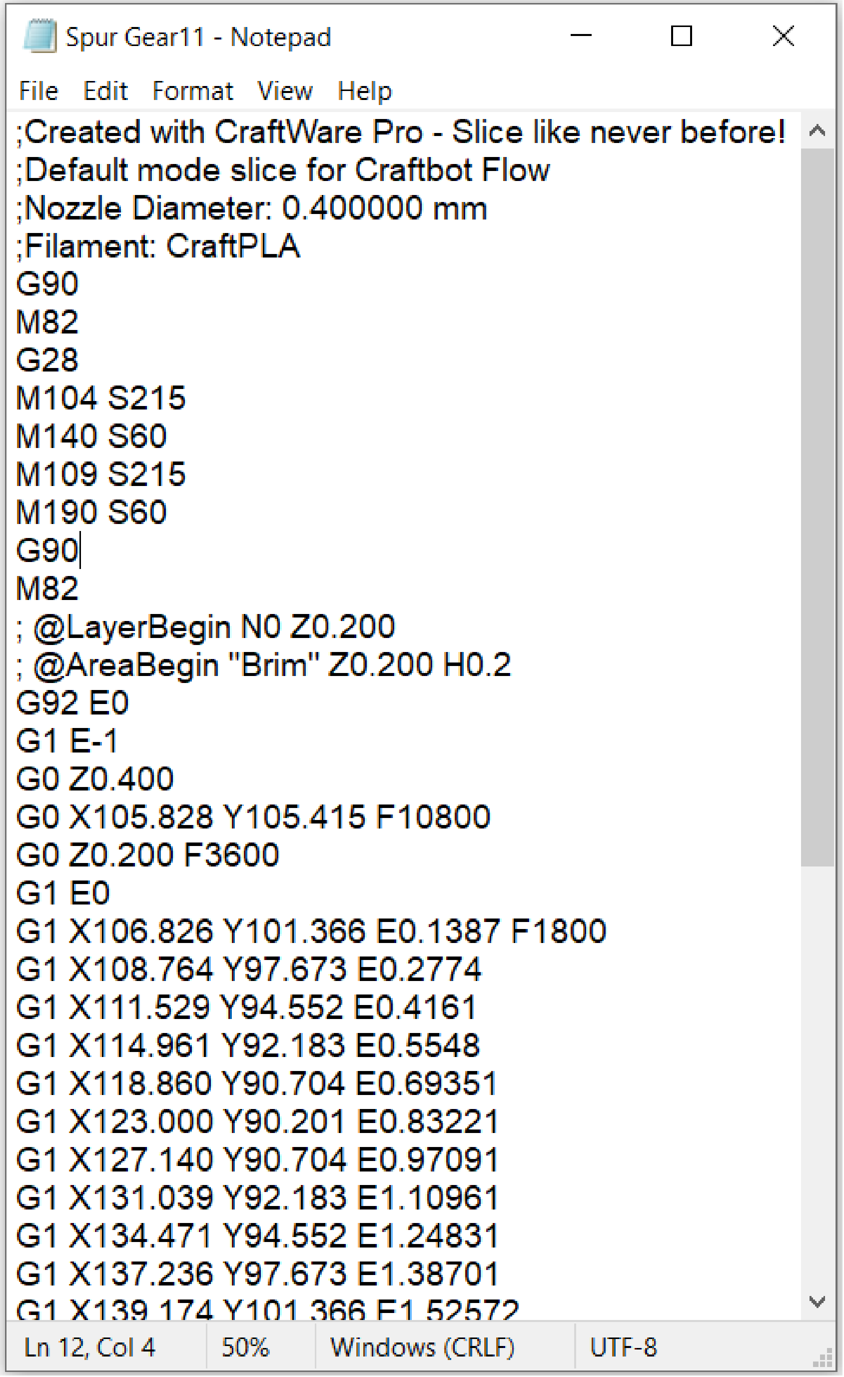




You can use the below and side sliders to view printing simulation.

You can Export the GCode now

- Open the saved G-code in the Notepad
- Done, this is the exercise!
- Congratulations!
- You will need to do something similar when you actually make a G code to print a part that goes into your project.

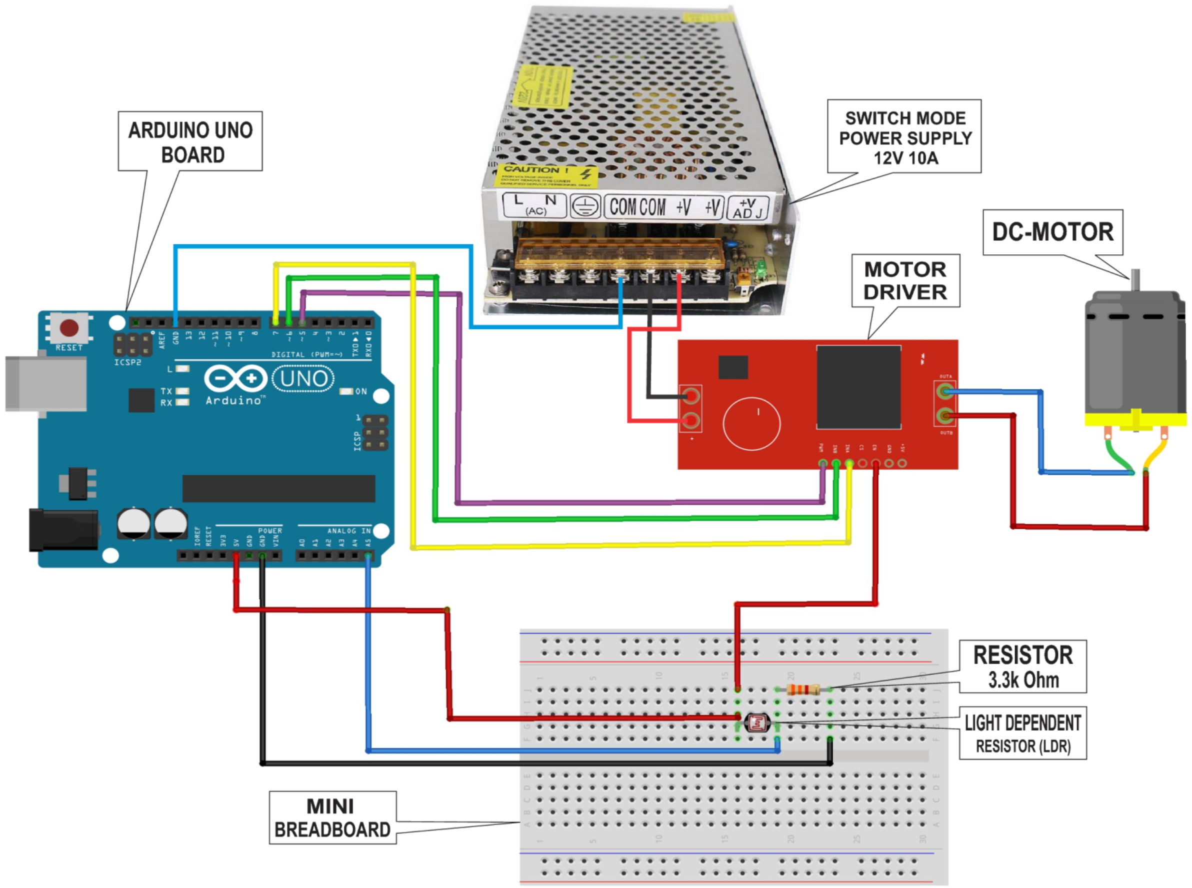


Arduino board-based DC motor control

Contributed by:
Mr. Srijan Bharati, Mr. Anurag Singh and Mr. Arijit Roy, MadLab, IITK
Updated by: Kishan Babu Prajapati and Rakesh Thapliyal, TA202 Labs

Objectives:

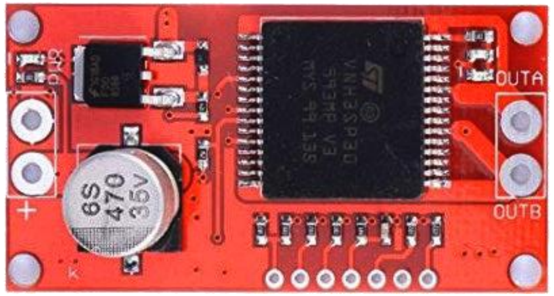
- To be introduced to Arduino programming
- To program Arduino UNO to run a DC motor and control its speed
- To program Arduino UNO to run a DC motor with spin (direction) control
- To use a light dependent resistor (LDR) and turn the motor ON and OFF by interrupting the light falling on the LDR



Your kit includes:

Arduino Uno board

Motor driver

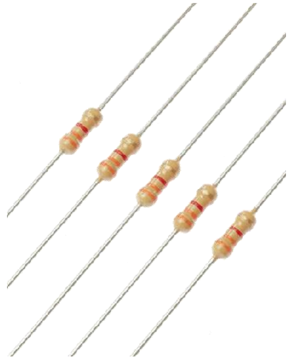


DC motor

Light dependent resistor

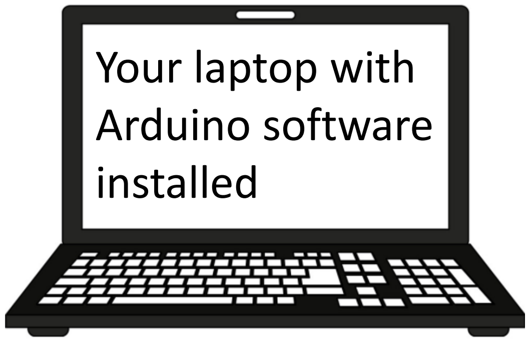
Jumper wires

Resistors



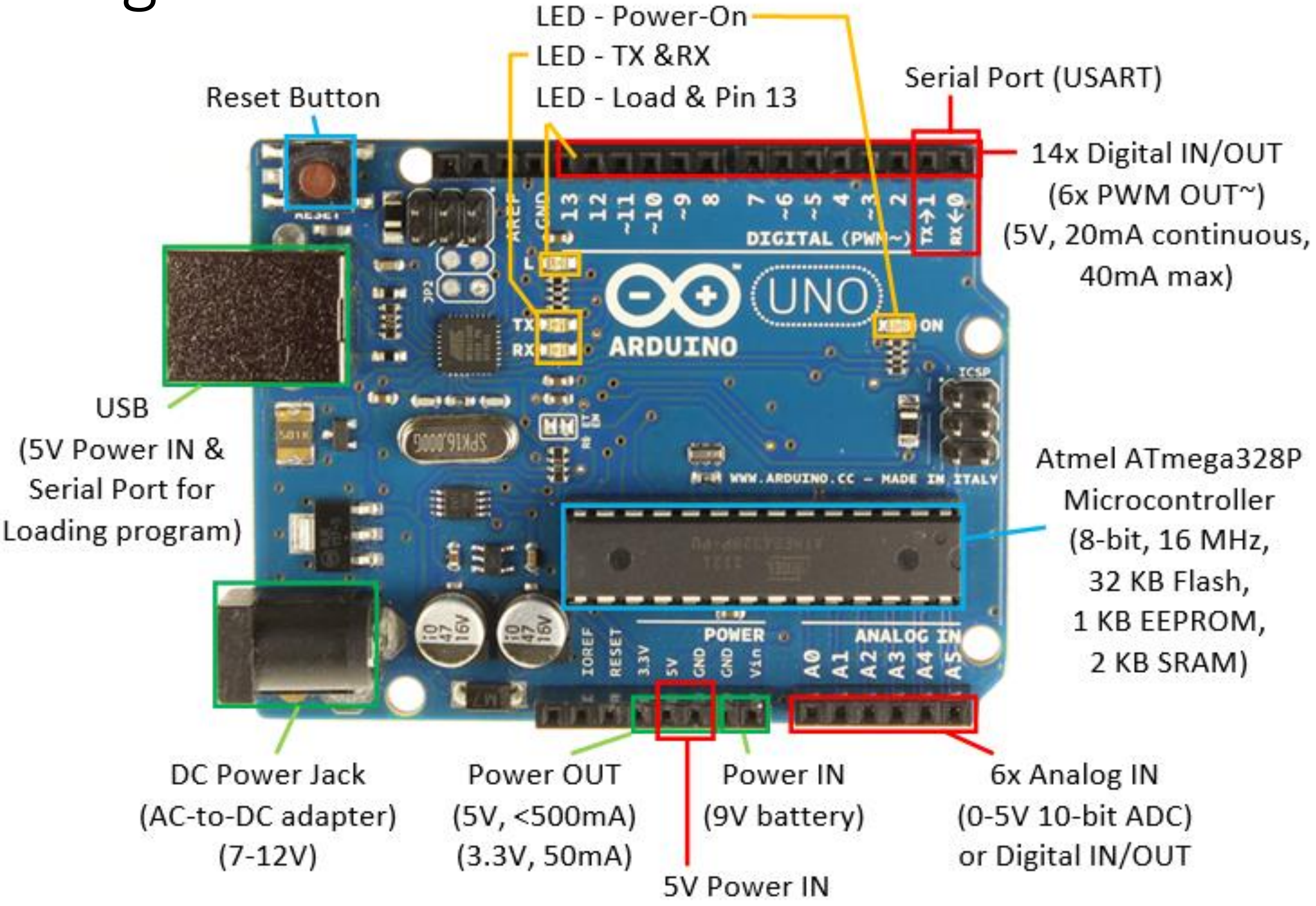
USB AB Cable

12V SMPS



Arduino Uno pin diagram

- You will power the board using the USB cable from the laptop
- You will only control motor in this lab, and as such, only pins 5, 6, 7 and +5V will be used for output and pin A5 will be used for input in this exercise



https://www.ntu.edu.sg/home/ehchua/programming/arduino/Arduino.html

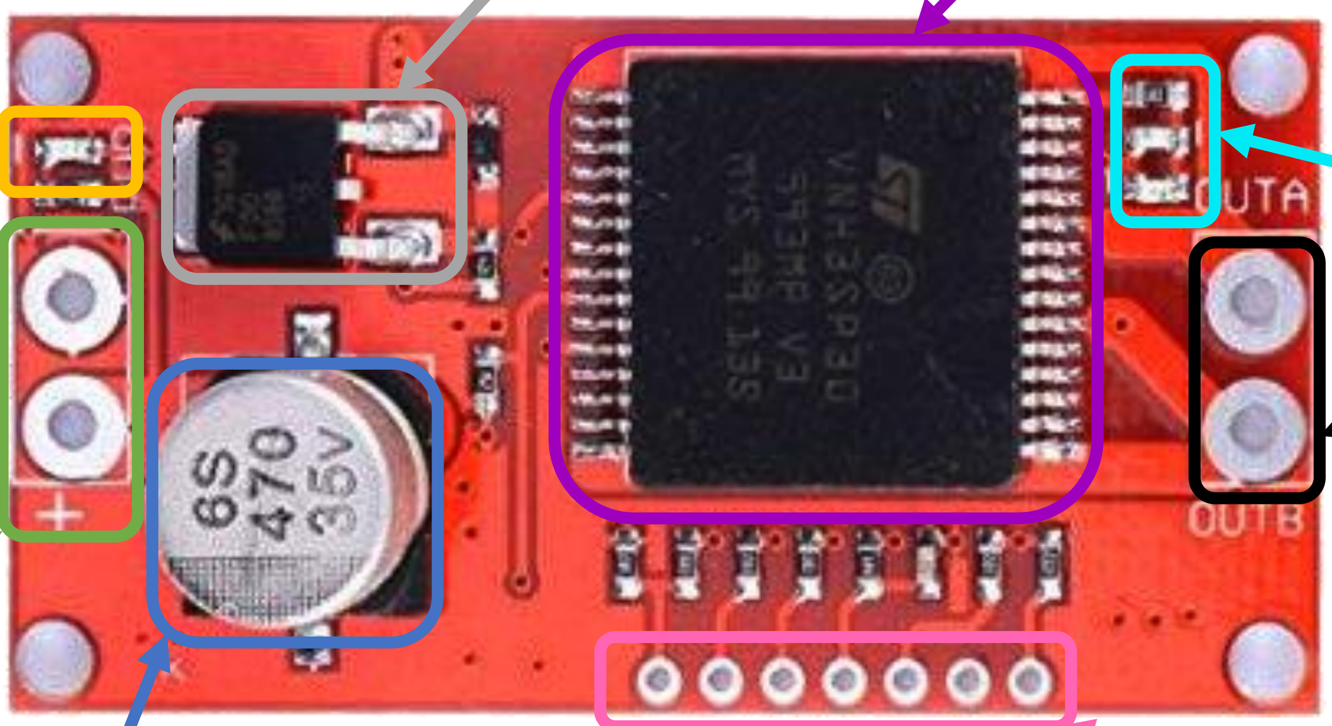
Motor driver pin diagram

- You will power the board using 12V SMPS supplied

5V VOLTAGE REGULATOR

H BRIDGE DRIVER IC

POWER INDICATOR LED



OUTPUT INDICATOR LEDS

OUTPUT TERMINAL
15A MAX CURRENT
USE HEATSINK BEYOND 5A CURRENT

INPUT TERMINAL 12V DC
36V MAX, 18V SAFE LIMIT

STABLIZATION CAPACITOR

CONTROL INPUT PINS
(FROM LEFT TO RIGHT)
PWM : PULSE WIDTH CONTROL (UPTO 10 kHz)
INB : INPUT FOR ONE SIDE OF H BRIDGE
INA : INPUT FOR OTHER SIDE OF H BRIDGE
CS : CURRENT SENSE ANALOG OUTPUT
EN/DIAG : ENABLE ENTIRE DRIVER
GND : GROUND
+5V

https://www.instructables.com/id/Tutorial-for-VNH2SP30-Monster-Motor-Module-Single-/
https://www.st.com/resource/en/datasheet/vnh3sp30-e.pdf

Additional resources

- For more on how the Arduino board works, see: https://www.ntu.edu.sg/home/ehchua/programming/arduino/Arduino.html
- For more on how the driver works, including what a H-bridge is, and what a PWM signal is, please see: **https://howtomechatronics.com/tutorials/arduino/arduino-dc-motor-control-tutorial-l298n-pwm-h-bridge/**
- For more on how to use the motor driver used in the exercise: https://www.instructables.com/id/Tutorial-for-VNH2SP30-Monster-Motor-Module-Single-/
- For more on Light dependent resistors:

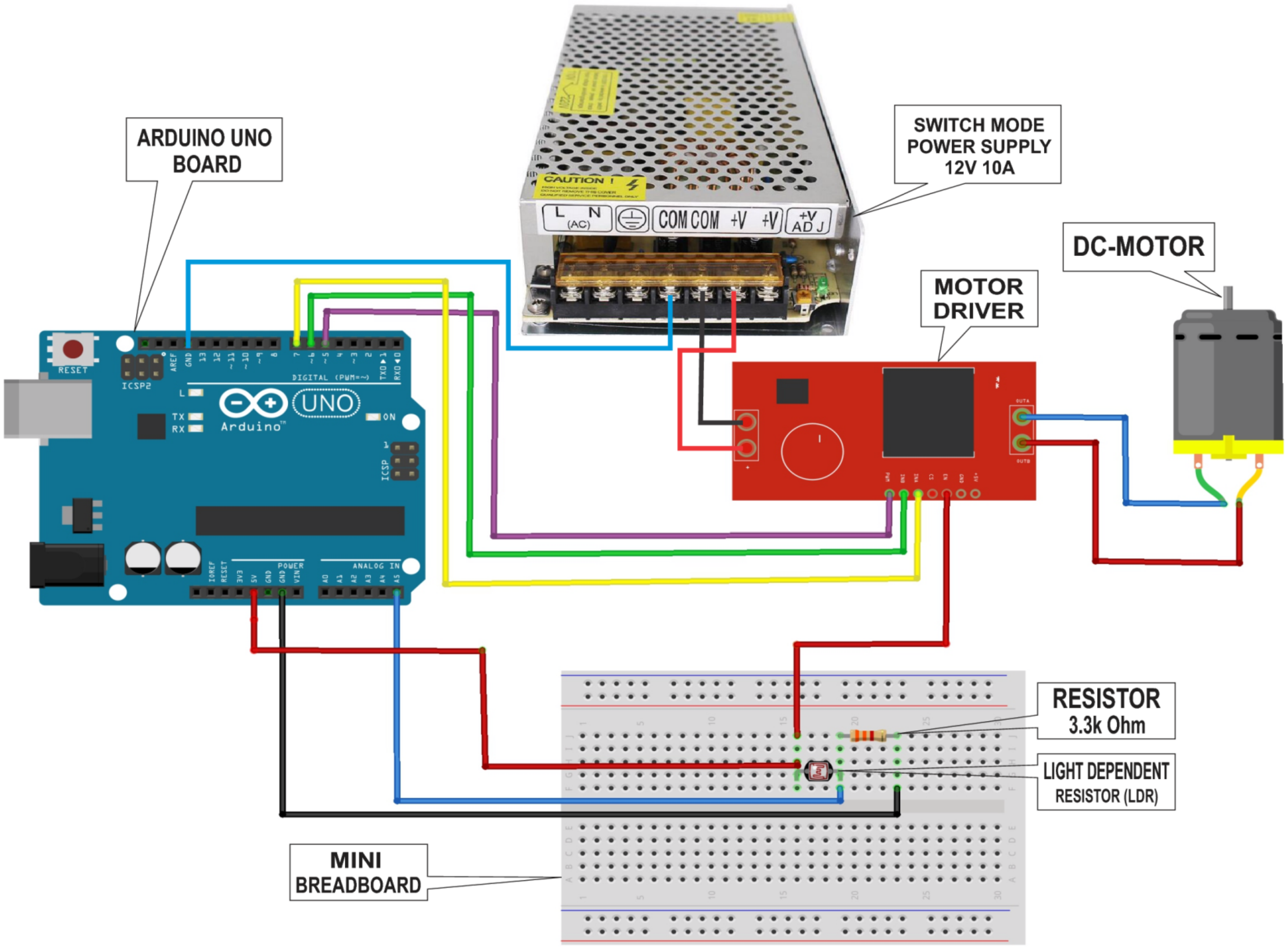
  https://www.electronics-tutorials.ws/io/io_4.html

  http://technologystudent.com/elec1/ldr1.htm

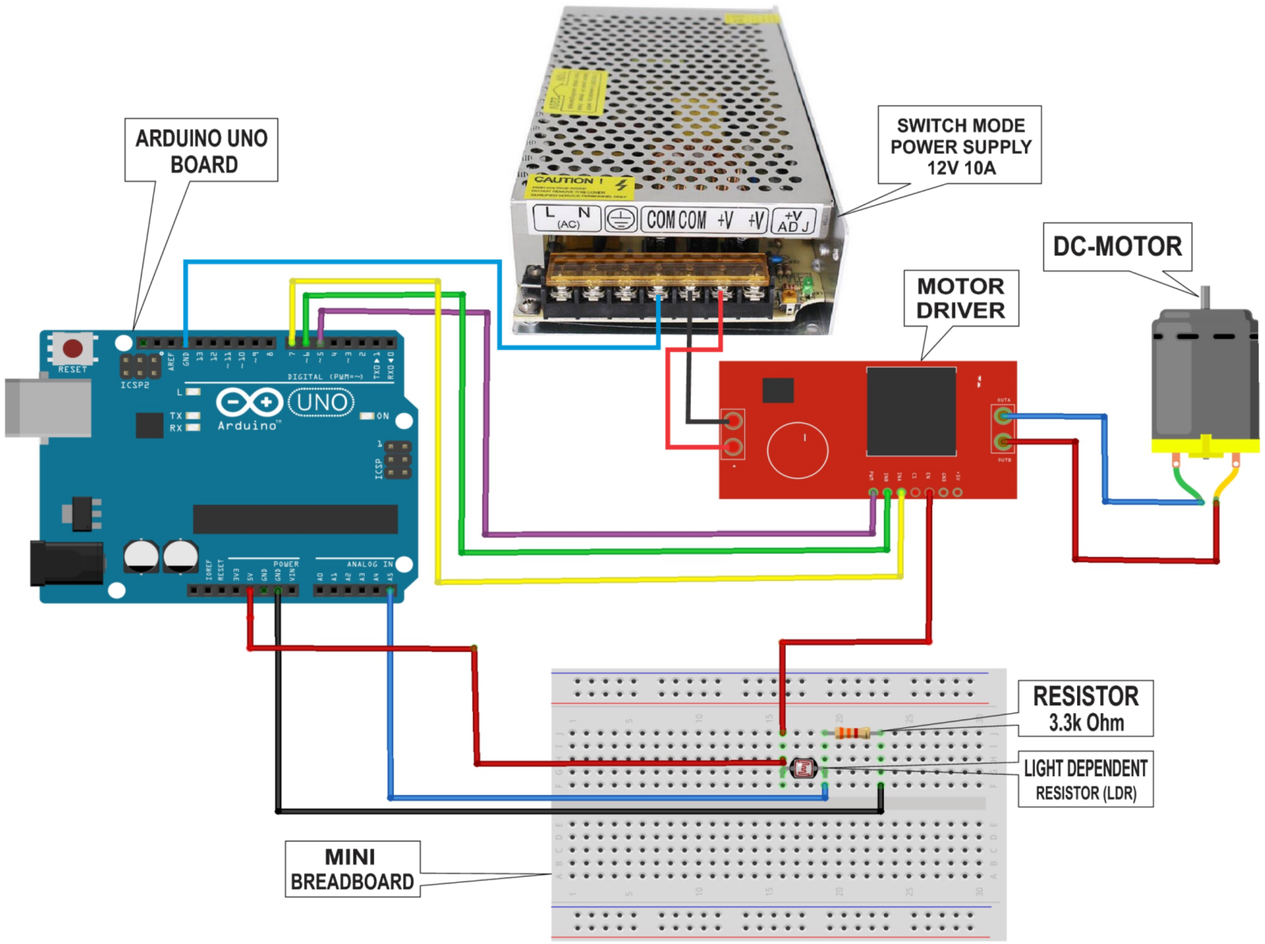
  https://www.electronics-notes.com/articles/electronic_components/resistors/light-dependent-resistor-ldr.php

  http://www.resistorguide.com/photoresistor/

Schematic of connections to be made. Details on next slide.



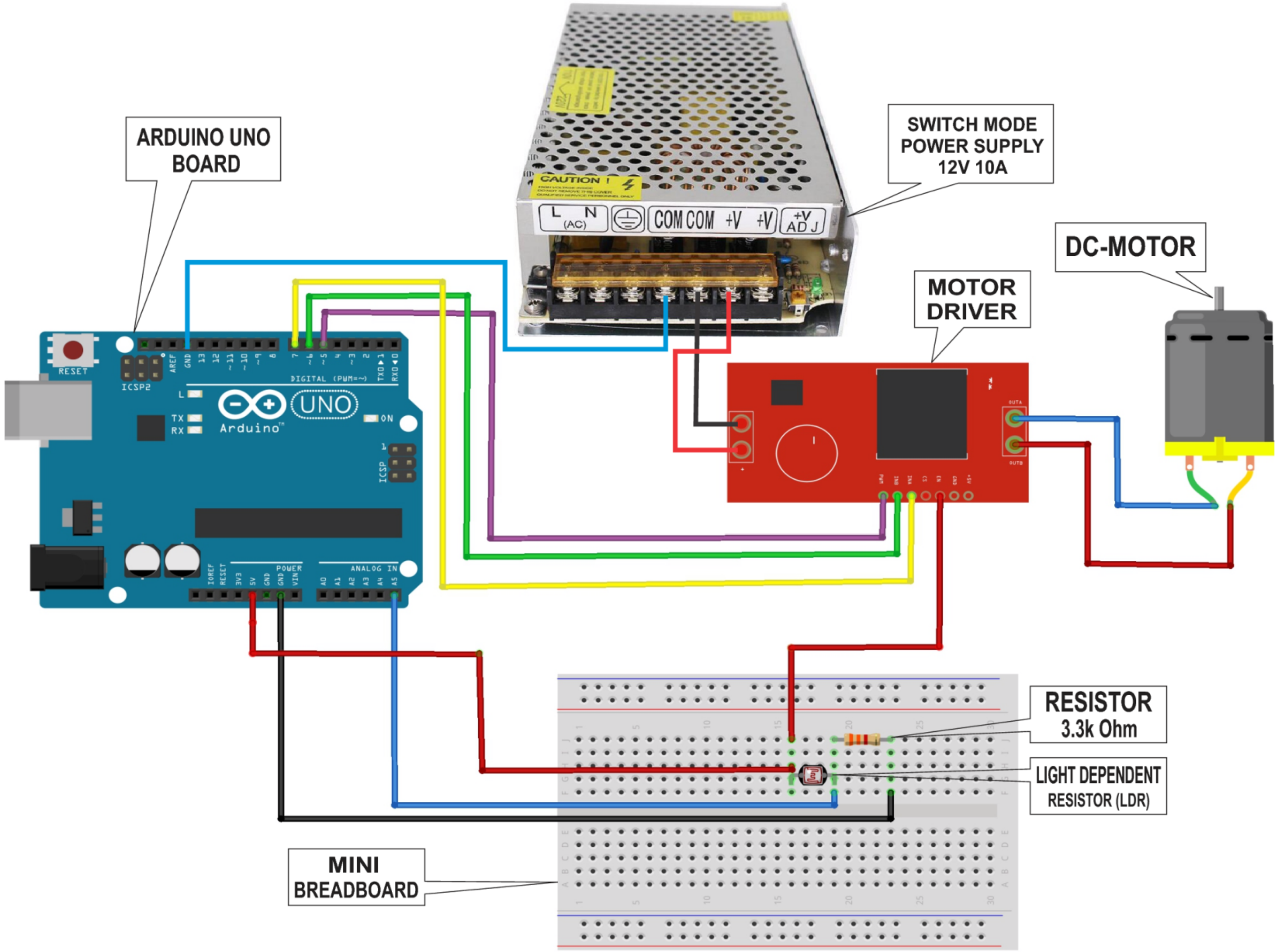
Schematic of connections to be made:



Pin connections :-

1. Connect **pin 5** of Arduino to pin **PWM** of the motor controller (PWM Output)
2. Connect **pin 6** of Arduino to pin **INB** of the motor controller (Digital output)
3. Connect **pin 7** of Arduino to pin **INA** of the motor controller (Digital output)
4. Connect LDR and 3.3kOhm resistor in series and connect the free end of the resistor to any **GND pin** on Arduino as shown in schematic drawing with help of breadboard.
5. Connect the common end of LDR and resistor to **pin A5** of Arduino (Analog input)
6. Connect **+5V** of Arduino to **EN** of the motor controller and free end of the **LDR**
7. Connect V^- of the SMPS to any **GND pin** on the Arduino.

Schematic of connections to be made:



Pin connections :-

8. Connect **DC motor** with **OUTA, OUTB** of the **MOTOR DRIVER**
9. Connect V^+ of SMPS to the **'+'** pin (+5 to +36V DC input) of the motor controller
10. Connect V^- of the SMPS to the **Ground** pin of the motor controller

Note :- Connect SMPS in the last step i.e. after uploading the code to Arduino board
If the circuit doesn't work check for lose connections and faulty jumper wires

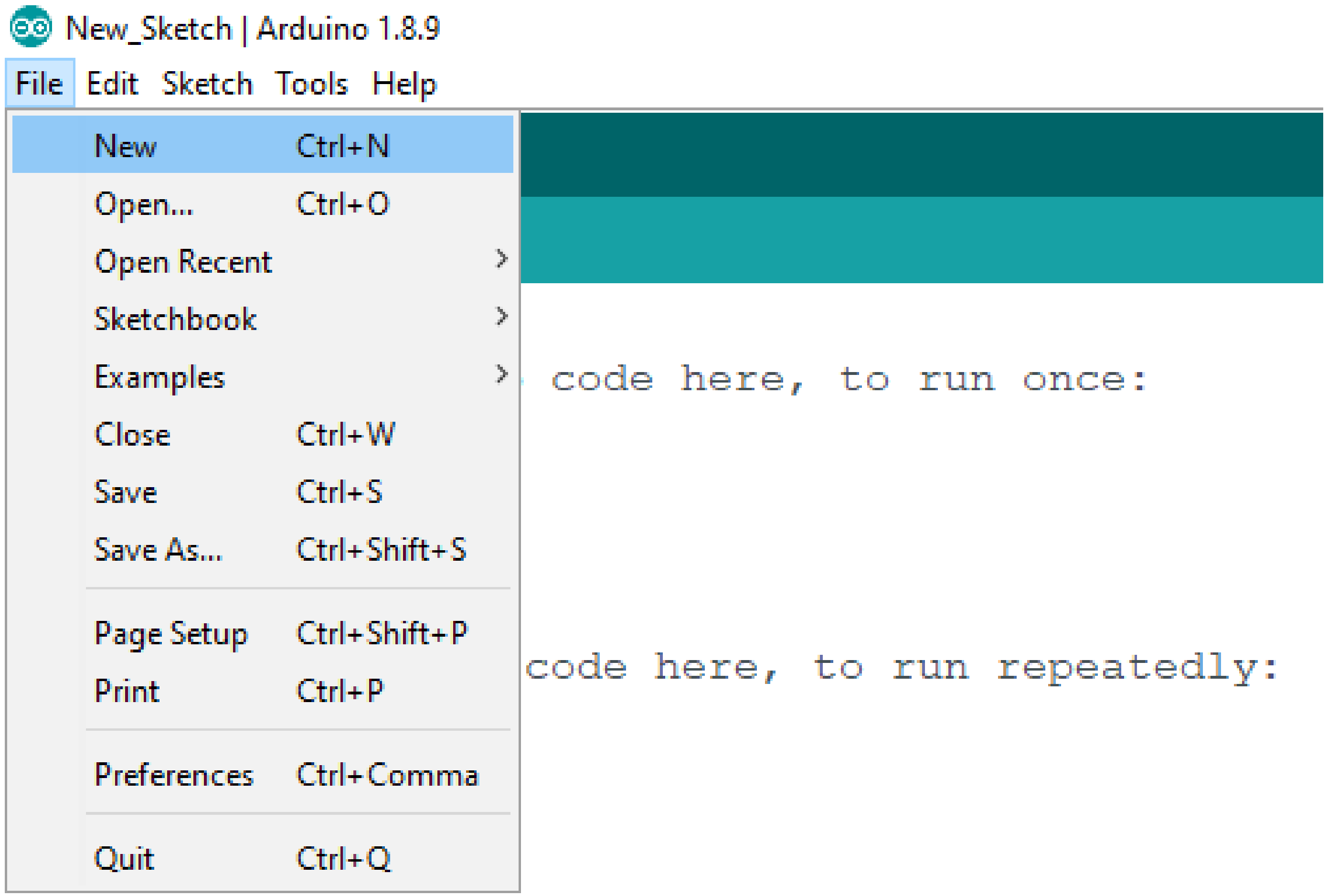
Make sure that the SMPS is not connected to the motor driver!

After successfully making all pin connections as per the schematic, have it assessed by your guide/TA/Tutor before proceeding further.

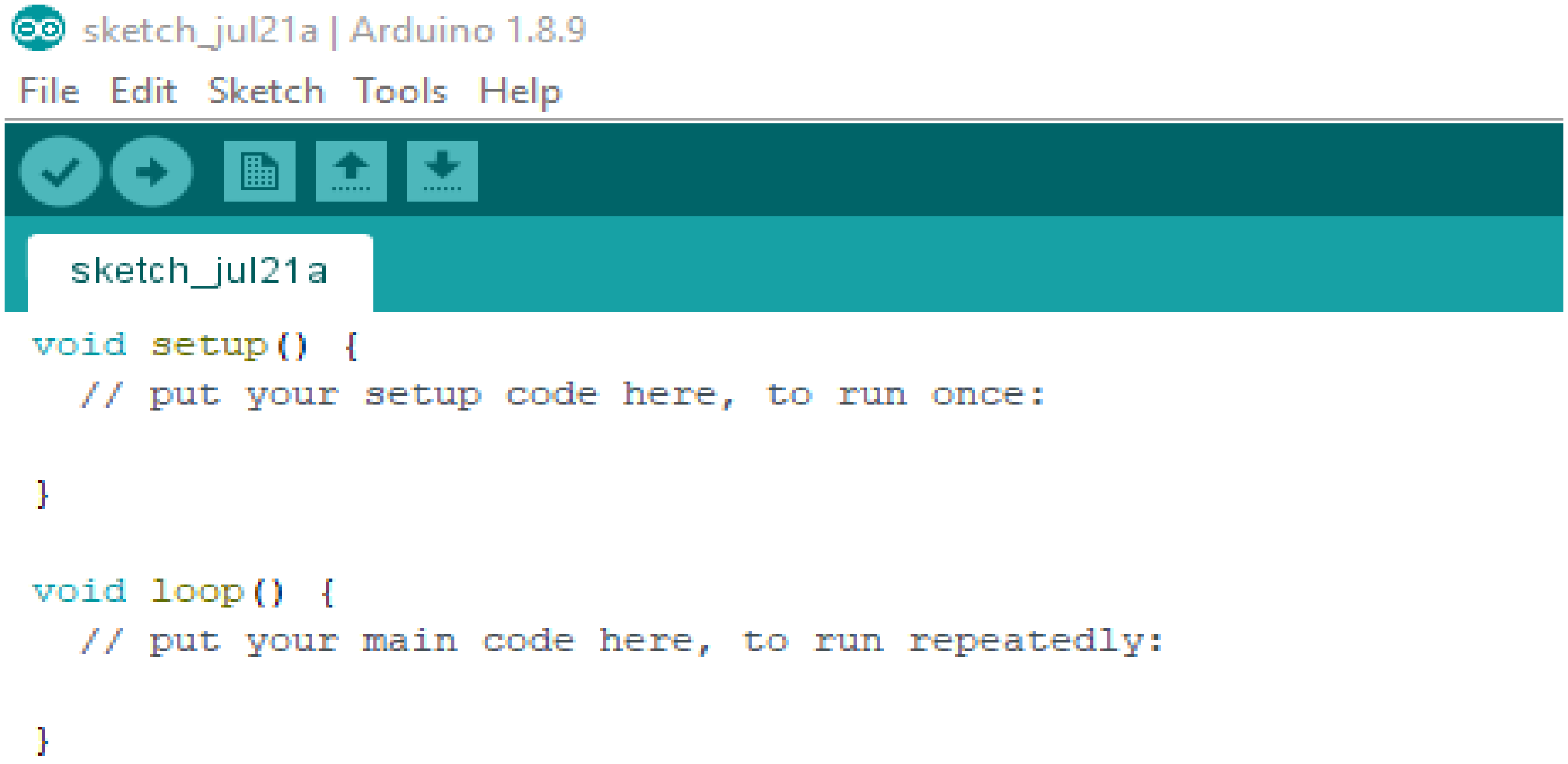
Once assessed, load the Arduino software on your laptop.

On loading the software, you should see something like:

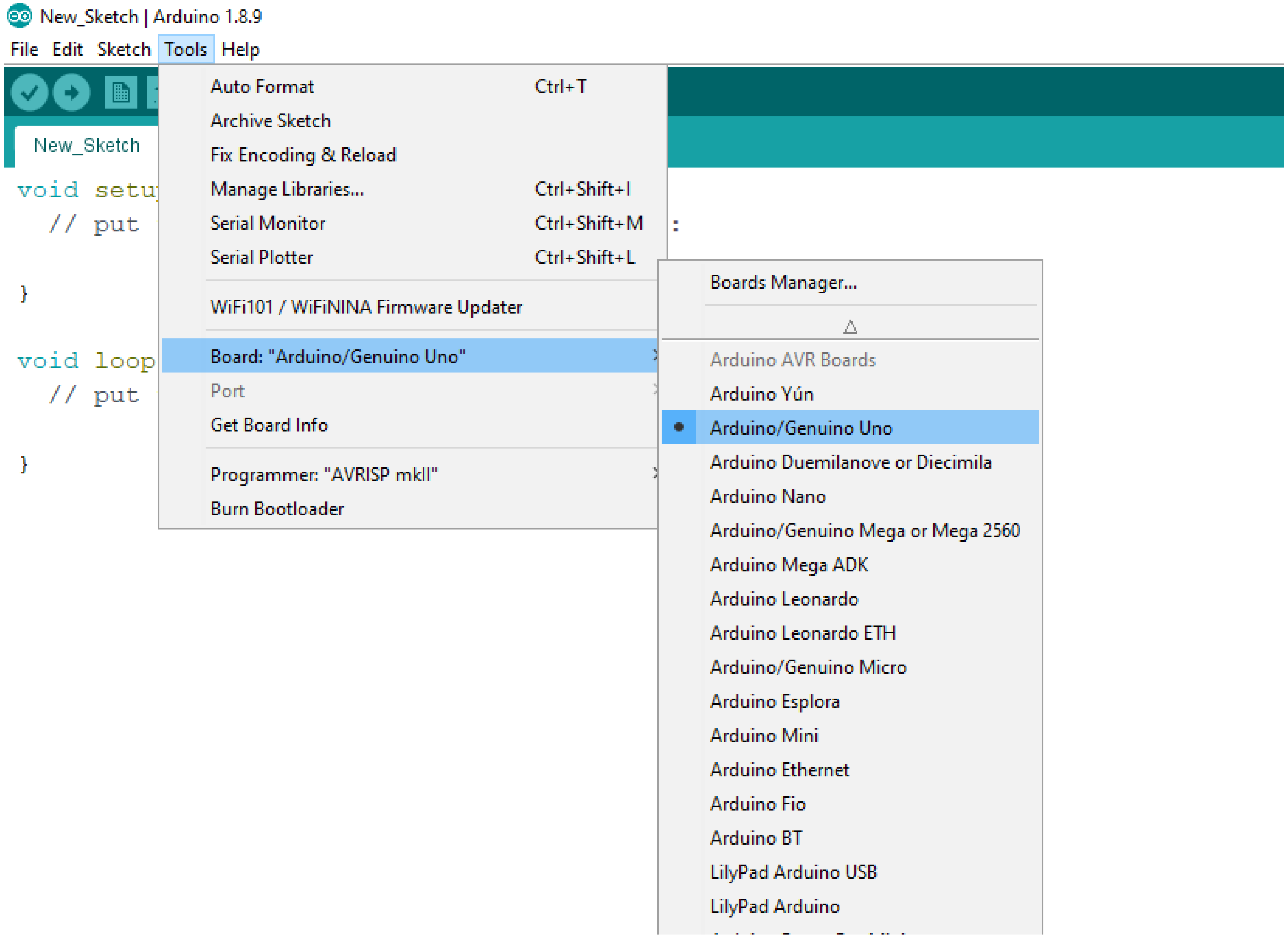
<u>Open a new sketch</u>



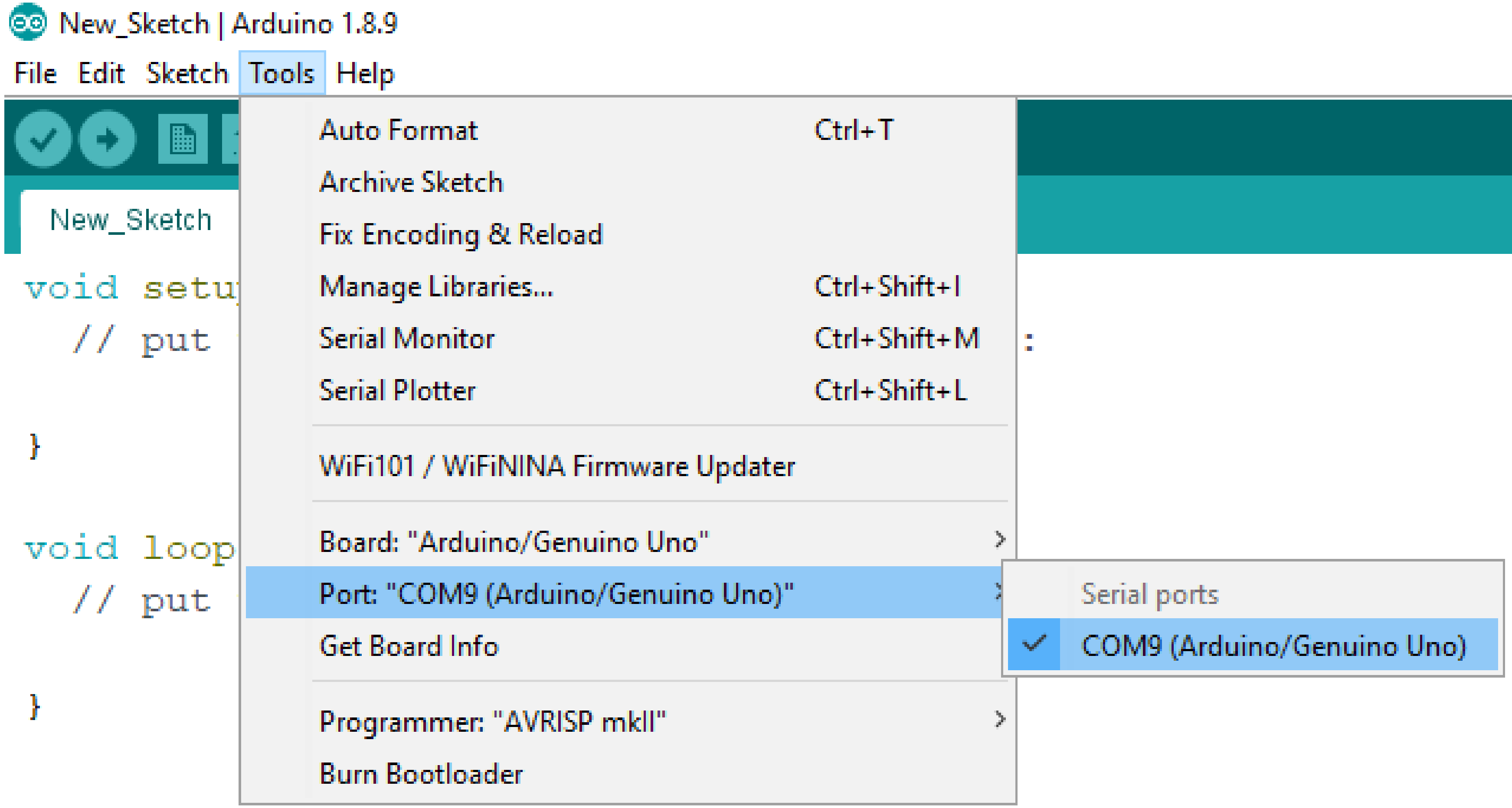
New sketch default window:



Selection of Board: Under ‘Tool’s, select the board correctly – in this case ‘Uno’.



Selection of port: Under ‘Tools’, after connecting the USB AB cable to your computer and Arduino board, check the port connection as shown. Make sure to do this every time you connect the board to the computer



Type out the following code as is. Comments should be self-explanatory.

TA202_Motor_control_with_LDR | Arduino 1.8.5

File Edit Sketch Tools Help

TA202_Motor_control_with_LDR

```
/*Parameters and Constants*/

int PWM = 5;                //connect PWM of Motor Driver to Pin 5 of Arduino UNO Board
int INB = 6;                //connect INB of Motor Driver to Pin 6 of Arduino UNO Board
int INA = 7;                //connect INA of Motor Driver to Pin 7 of Arduino UNO Board

int SENSOR = A5;            //Used to take analog input. In this case sense the resistance change of the Light Dependent Resistor

int SPEED = 120;            //PWM (Pulse Width Modulation) value used to control the speed of the motor. The value can be varied from 0(Lowest) to 255(Highest)
                            //Note! Motor doesn't start untill a significant voltage/PWM is applied due to internal losses and static friction. If PWM is set 0 motor is idle.
int THRESHOLD = 700;        //Arduino will recieve value from 0(dark) to 1023(bright). This value is to set a cutoff. Change this to tune it to your ambient illumination.


/*The code has two functions void setup() and void loop() void setup() runs only once
  in the beginning and after pressing reset. void loop() runs in never ending loop
  once void setup() is completed*/


void setup() {

                            //pinMode(PIN NUMBER,MODE) is used to pre define the character of characteristic of the pin
  pinMode(PWM,OUTPUT);      //PWM (Pulse Width Modulation) pin is used to control the speed of the motor, the value can be varied from 0(Lowest) to 255(Highest)
  pinMode(INA,OUTPUT);      //INA and INB control the direction of rotation of the motor        | INA  | INB  |  State  |
  pinMode(INB,OUTPUT);      //                                                                  | HIGH | LOW  | Forward |
                            //                                                                  | LOW  | HIGH | Reverse |
                            //                                                                  | HIGH | HIGH |  Brake  |
                            //                                                                  | LOW  | LOW  |  Brake  |
  pinMode(SENSOR,INPUT);    //LDR will be used as a potentiometer and the varying voltage will be measured through this pin.

  analogWrite(PWM,0);         //Turn the motor OFF by default
}
```

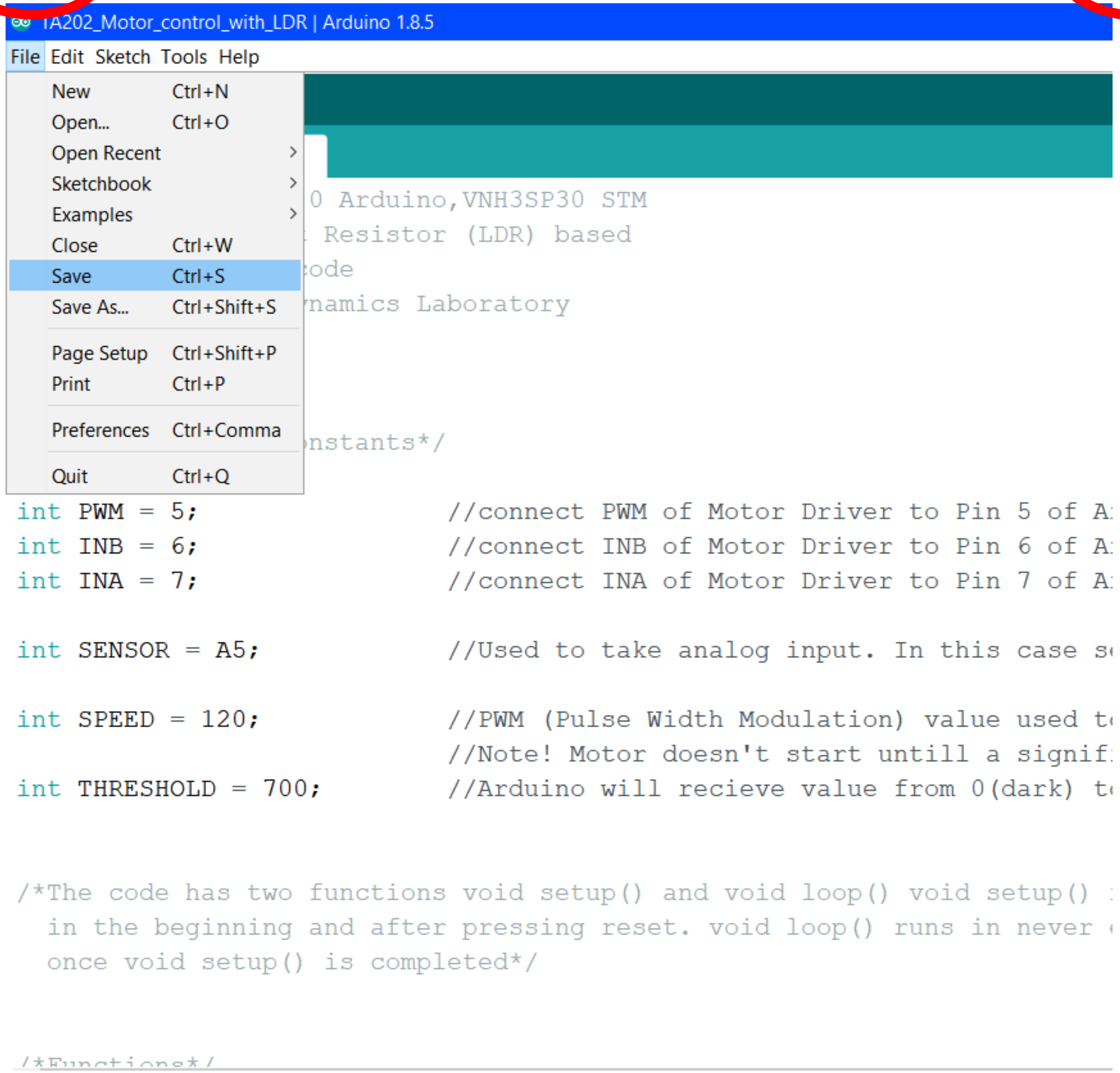
Type out the following code as is. Comments should be self-explanatory.

```
void loop() {

  digitalWrite(INA,HIGH);     //These two lines define the default direction of rotation, interchange INA and INB to flip the direction of rotation.
  digitalWrite(INB,LOW);


  if( analogRead(SENSOR) <= THRESHOLD ) {
    analogWrite(PWM,SPEED);
  }                            //When the sensor output goes below the set threshold value the motor turns ON.
  else {
    analogWrite(PWM,0);
  }
  delay(100);                  //Add delay to wait before loop runs again. Delay is entered in Milliseconds.

}
```

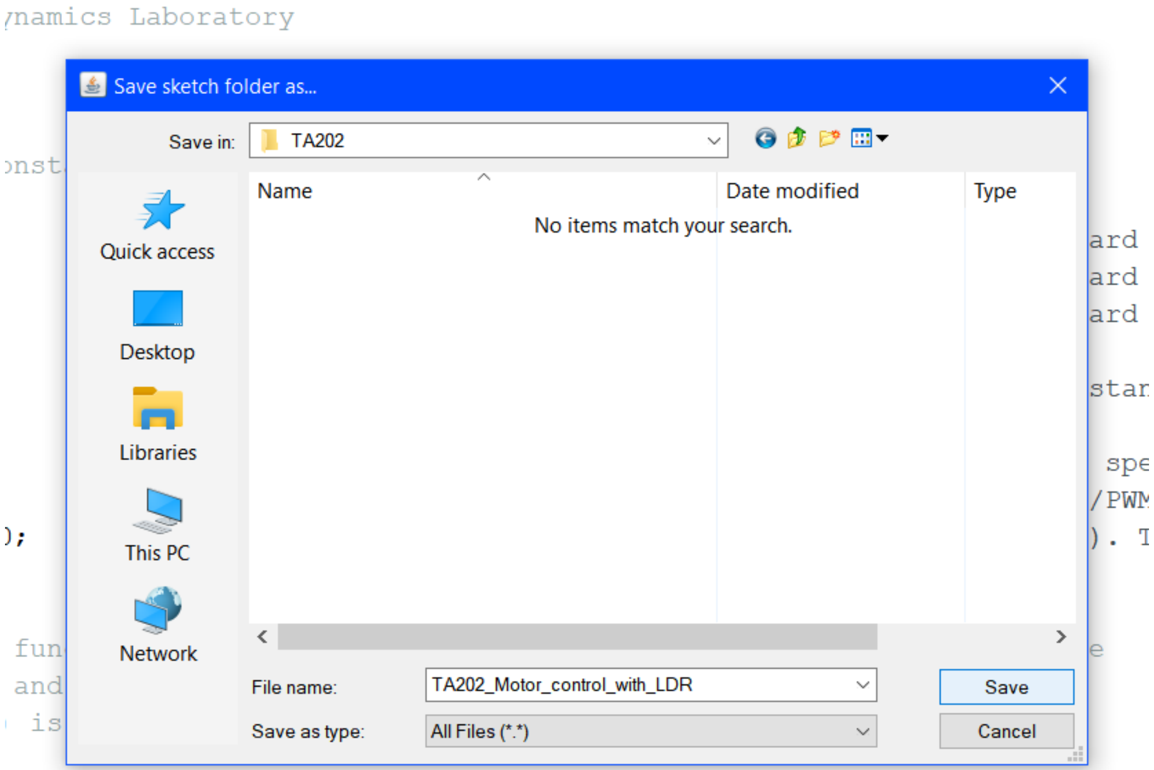
Note: You’ll have to play with the variables ‘SPEED’ and ‘THRESHOLD’ to get the desired results

Save the code

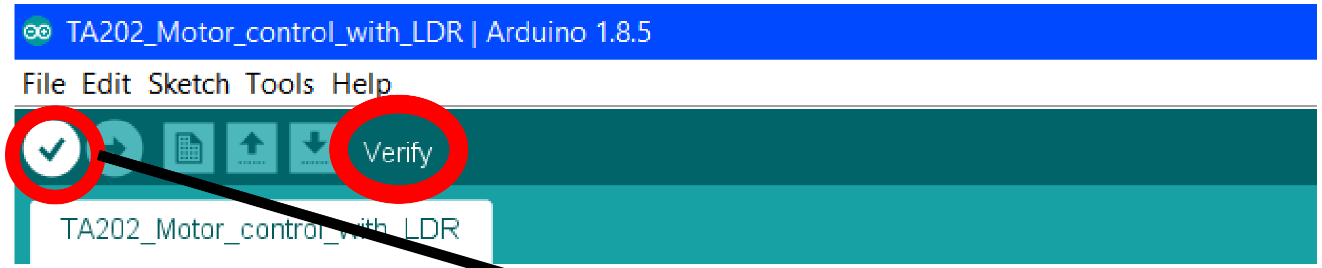
1



2



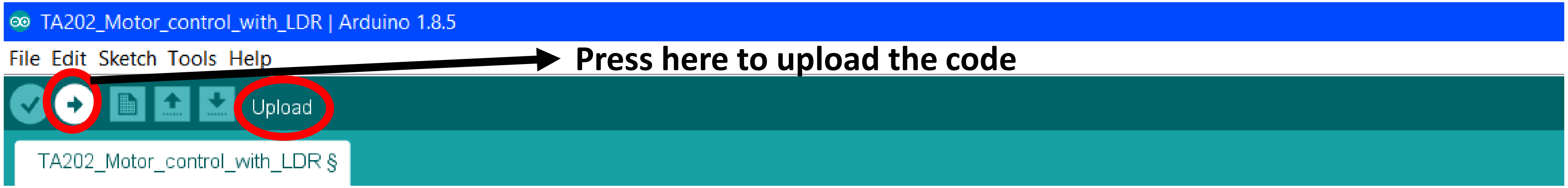
Verify the code as shown. If you find any errors in the code, correct them.



Press here to verify the code

```
/*The code has two functions void setup() and void loop(
  in the beginning and after pressing reset. void loop()
  once void setup() is completed*/


/*Functions*/

void setup() {

                              //pinMode(PIN NUMBER,MODE) i
  pinMode(PWM,OUTPUT);        //PWM (Pulse Width Modulatio
  pinMode(INA,OUTPUT);        //INA and INB control the di
  pinMode(INB,OUTPUT);        //
                              //
```

After verifying the code and connecting the USB AB cable to Arduino and computer, upload the code into Arduino board : -

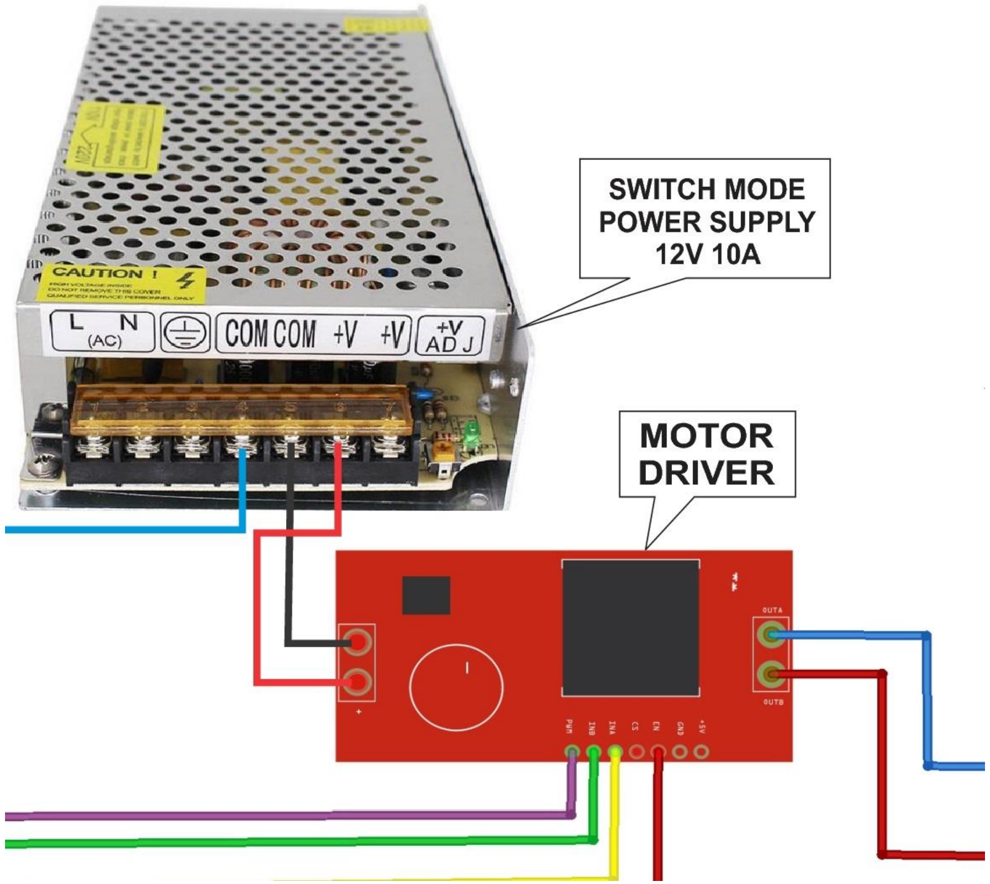



```
/*The code has two functions void setup() and void loop(), void setup() runs only once
  in the beginning and after pressing reset. void loop() runs in never ending loop
  once void setup() is completed*/


/*Functions*/

void setup() {

                             //pinMode(PIN NUMBER,MODE) is used to pre define the character of cl
  pinMode(PWM,OUTPUT);       //PWM (Pulse Width Modulation) pin is used to control the speed of t
  pinMode(INA,OUTPUT);       //INA and INB control the direction of rotation of the motor
  pinMode(INB,OUTPUT);       //
                             //
                             //
```

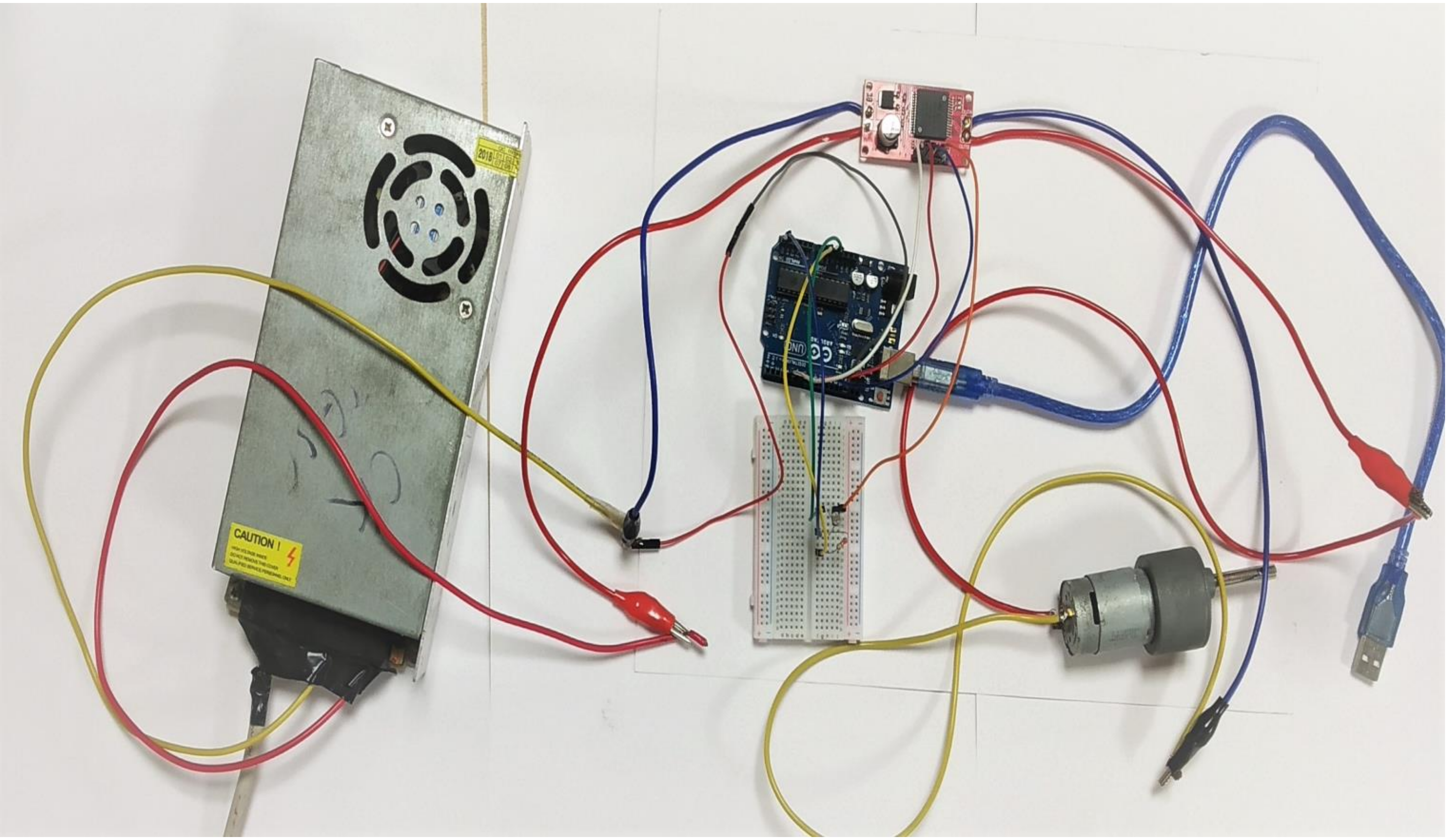
If you are getting any error in code correct it. Sometimes the error may also be due to not selection of **'port'** option.

Now, after uploading the program into the Arduino board.

Connect SMPS to motor controller as shown in figure below.



Your setup should look like this, only that the shaft on your motor should be spinning with the speed you have programmed when you cover the LDR:



Have this assessed by your guide/TA/Tutor before proceeding to the next step.

Now, load a new sketch – as you did earlier. And type out the code below – as is. This code is for spin direction control. Save the code. Verify it. Upload it. If all is well, you can now change the spin of the motor by changing levels form 'HIGH' to 'LOW' as shown in the table

File Edit Sketch Tools Help

TA202_Motor_control_with_LDR §

```
void setup() {

                              //pinMode(PIN NUMBER,MODE) is used to pre define the character of characteristic of the pin
  pinMode(PWM,OUTPUT);        //PWM (Pulse Width Modulation) pin is used to control the speed of the motor, the value can be varied from 0(Lowest) to 255(Highe:
  pinMode(INA,OUTPUT);        //INA and INB control the direction of rotation of the motor        | INA  | INB  |  State  |
  pinMode(INB,OUTPUT);        //                                                                   | HIGH | LOW  | Forward |
                              //                                                                   | LOW  | HIGH | Reverse |
                              //                                                                   | HIGH | HIGH |  Brake  |
                              //                                                                   | LOW  | LOW  |  Brake  |
  pinMode(SENSOR,INPUT);      //LDR will be used as a potentiometer and the varying voltage will be measured through this pin.

  analogWrite(PWM,0);           //Turn the motor OFF by default
}

void loop() {

  digitalWrite(INA,HIGH);       //These two lines define the default direction of rotat
  digitalWrite(INB,LOW);

  if( analogRead(SENSOR) <= THRESHOLD ) {
    analogWrite(PWM,SPEED);
  }                             //When the sensor output goes below the set threshold v
  else {
    analogWrite(PWM,0);
```

INA Input	INB Input	Motor State
HIGH	HIGH	Brake
HIGH	LOW	Forward direction motion of motor
LOW	LOW	Brake
LOW	HIGH	Reverse direction motion of motor

Done!

If you’ve been able to control the speed and the spin direction, your exercise is complete.

Arduino has some lovely tutorials for those interested.

You will have to use this kit in your project to drive and control whatever it is that you like.

Some references:

- https://lastminuteengineers.com/l298n-dc-stepper-driver-arduino-tutorial/
- https://www.arduino.cc/
- https://slideplayer.com/slide/3255631/
- https://www.theengineeringprojects.com/2018/06/introduction-to-arduino-uno.html
- https://www.teachmemicro.com/use-l298n-motor-driver/
- https://www.tutorialspoint.com/arduino/arduino_dc_motor
- https://youtu.be/m9J3mDYy0Sg

Software:

- Arduino - https://www.arduino.cc/en/Main/Software
- Electrical circuit schematic diagram - http://fritzing.org/home/