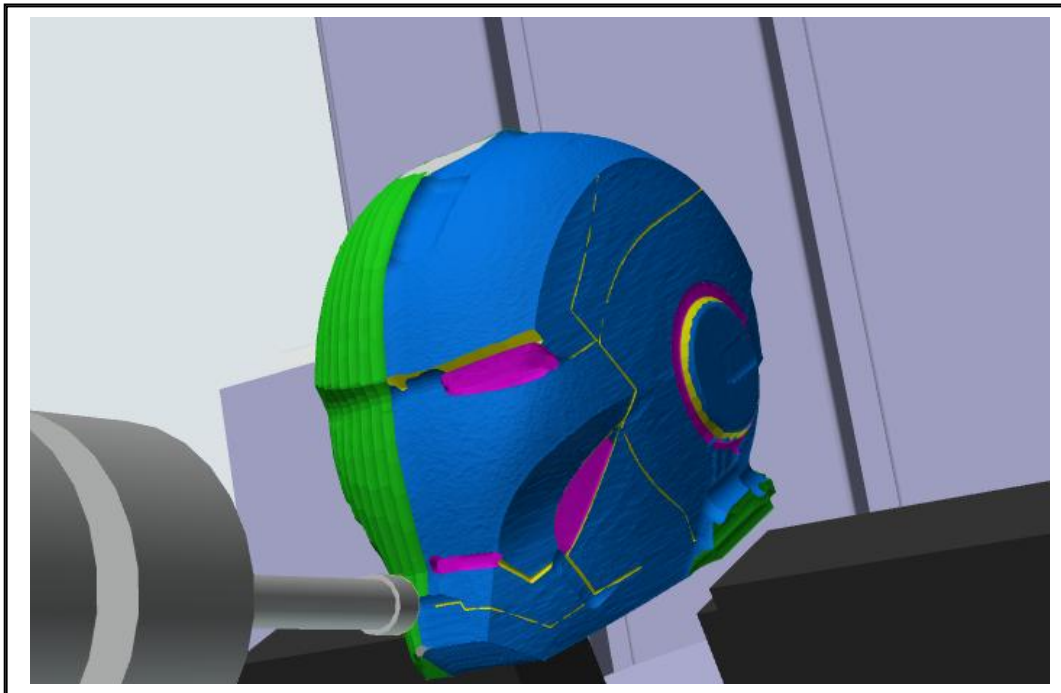


CHALMERS



DELMIA V6 Machining

Bachelor thesis for mechanical engineering

VALJON HASANI
ADAM NILSSON

Department of Product and Production Development
Division of production systems
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden, 2012
Examiner: Torbjörn Ylipää

PREFACE

After three years of studying Mechanical engineering, 180 credits, at Chalmers University of Technology this report is a result of our bachelor thesis containing 15 credits. A presentation about Dassault Systemes by Agneta Lundquist during our second year made us interested in working together with the company, a mail was sent to Agneta and the contact was reached. The cooperation during the project together with the DELMIA office in Gothenburg has worked very well, from the beginning we have been very welcomed and appreciated.

Especially thanks to:

Agneta Lundquist as the first contact with Dassault Systemes office in Gothenburg.

Jonas Fredriksson, our supervisor at DELMIA, for making this project possible and for giving us the tools to achieve success with the project.

Sven Ekered, our supervisor at Chalmers University, for a great support during the whole project.

Olivier Huron, which was the teacher in Aix-en Provence who taught us how the machining tool works which has made the whole project possible to fulfil.

Niklas Lewander and **Jim Bergman**, as our “technical administrators”, who helped us with computer, licenses and getting started.

SUMMARY

Machining is a conventional method using sharp cutting tools to physically remove material to achieve a desired geometry. There are a lot of different CNC machines and many ways on how to use them, the focus in this project is about 3- and 5-axis milling machines.

Dassault Systemes is a world leading company delivering the whole view of a product's lifecycle in 3D, from creation to maintenance, including manufacturing and recycling. To control the machines computer aided manufacturing software is used, something Dassault Systemes have provided in the CATIA V5 software. Dassault Systemes has launched the Version 6 and the products for machining are moved from the CATIA portfolio to the DELMIA portfolio which leads us to the purpose of this project.

When it comes to machining the competence in Sweden is focused around V5. This project aims to develop a more local competence around the machining capabilities in V6 and expand the already successful cooperation between Dassault Systemes and Chalmers University of technology.

The project aim is to obtain knowledge about machining capability in DELMIA V6. The experience after one week at Dassault Systemes office in Aix-en Provence, France gave the correct tools to build up a milling machine placed at Chalmers Lindholmen workshop and furthermore create and verify operations with this machine. The working process was documented and a step-by-step guide created.

Table of Contents

EXPLANATIONS	1
1 INTRODUCTION	2
1.1 Background	2
1.2 Purpose	3
1.3 Questions at issue	3
1.4 Boundaries.....	3
2. THEORETICAL BACKGROUND	4
2.1 CNC machines	4
2.1.1 3-axis milling machines	4
2.1.2 Multi-axis milling machines	5
2.2 Machining methods	6
2.2.1 Manual programming	6
2.2.2 Computer-aided manufacturing	7
2.2.3 Methods to verify	8
2.3 Pedagogical learning process	9
2.3.1 Companion	9
2.3.2 Writing a manual	10
2.4 DELMIA V6 interface.....	11
2.4.1 The machine programming interface	11
3 METHOD.....	12
3.1 Acquire knowledge	12
3.1.1 Preparatory studies	12
3.1.2 Further studies.....	13
3.2 Work in DELMIA V6	13
3.3 Documentation	14
4 WORKING PROCESS DELMIA V6.....	15
4.1 Building the machine.....	16
4.1.1 Defining kinematics.....	16
4.1.2 Machine tool builder	20
4.2 Creating the assembly.....	22
4.3 Setup	23
4.4 Operation	29

4.4.1 Prismatic Machining Operations	35
4.4.2 Axial Machining operation	37
4.4.3 Surface Machining operation	37
4.4.4 Choosing operations	43
4.5 Verifying	44
4.5.1 Simulation	44
4.5.2 Generate NC code.....	46
5 CREATING EXERCISE	47
5.1 Identify vital areas	47
5.2 Building exercise	48
6 CONCLUSION AND RECOMMENDATIONS	49

Appendix step-by-step guide

EXPLANATIONS

CNC – Computer numerical control

CAM – Computer aided manufacturing

NC code – Numerical control code

Post-processor – Translates CAM code to specific NC code

V5, V6 - Version 5 and Version 6

Prismatic machining – 2.5- 3-axis operations

Milling machining – 3- and 5-axis operations

Extended milling machining – Advanced 5-axis operations

Part – Product to be machined

Check – Fixture holding the part to be machined

Rough stock – origin material before any operation

PPR tree – Product Process Resources tree

1 INTRODUCTION

This chapter aims to give a short explanation of what Dassault Systemes are working with and a short history how they developed through time. It also explains the purpose of the work and specifies the questions at issue.

1.1 Background

Dassault Systemes is a world leading company delivering the whole view of a product's lifecycle in 3D, from creation to maintenance, including manufacturing and recycling. It all started in 1981 as an independent division of Dassault Aviation, later that year a marketing contract was also made with IBM. The development of Dassault Systemes from start to today can be defined by three periods. The first period is called the 3D, the tool for complex shapes and forms that were developed and launched in 1981, the product was CATIA V1. In the year of 2000 Dassault Systemes together with IBM introduce product lifecycle management (PLM). Later the same year the acquisition of DELMIA took part as a strategic step towards a broader portfolio and towards the second period, who took place in year 2002, the PLM 1.0. Dassault Systemes goal toward bringing their customers a full product lifecycle view of their product lead us to the third period where we are today, the introduction of the PLM 2.0. With products as CATIA, SolidWorks, SIMULIA, DELMIA, ENOVIA, 3DVIA, 3DSwYm and Exalead the new platform is the first real step toward allowing a full product lifecycle management. The PLM 2.0 is a 3D online environment, connecting users through a database and all user interactions generate an Intellectual Property. It allows users to imagine, share and experience products in the universal language of 3D.

In the PLM 2.0 Dassault Systemes has launched the Version 6 and the products for computer aided manufacturing (CAM) are moved from the CATIA portfolio to the DELMIA portfolio. Transferred products including modules to create and simulate 3-axis and 5-axis milling operations with computer numerical control (CNC) machines.

Dassault Systems and Chalmers University of technology are having a successful cooperation in many areas including both CATIA and DELMIA. Current activities are focused around V5. Also when it comes to knowledge about Dassault Systemes machining solutions the competence in Sweden is around V5.

There is a need to develop more local competence around the machining capabilities in V6. Dassault Systemes also see an advantage in expanding the cooperation with Chalmers around machining and to have the ability to demonstrate an End-to-End process including a real CNC Machining. As a result of this Dassault Systems has made the decision to initiate this project.

1.2 Purpose

The purpose of this thesis is to collect expertise about how the software DELMIA V6 works, how to perform machining preparations and verify the operations. Furthermore how to do modeling, relating to the kinematics, the CNC-machine where the detail is to be manufactured in. The last-mentioned is to simulate and analyze possible collisions in the machine when an operation is run.

The acquired knowledge should be documented and presented in an educational and easily understandable way so that preparations and simulations can be executed single-handedly by the customers of DELMIA.

1.3 Questions at issue

To achieve the purpose of this project four questions were listed to be answered.

- How to describe and document the machining preparation methodology for DELMIA V6 in a pedagogical way so the reader can easily understand the software structure?
- How is the 3-axis and 5-axis operations created in DELMIA V6?
- How is the operations verified and how to generate an NC code?
- What methods to use to model a real CNC machine regarding to the kinematics?

1.4 Boundaries

The report will only include preparations with respect to the 3-5 axis milling machines. Operations and parameters included in this report are only those to create the product in the appendix. A post-processor will not be included in this report and therefore there will be no manufacturing of the product in the reality. The modeling process of the different machine parts will not be included in this report.

2. THEORETICAL BACKGROUND

There are many different CNC machines to fill a lot of different purposes, also machining methods and verifying the numerical control (NC) code can be done in many different ways. In order to provide clarity this chapter explains the most common machines and methods used today. Dassault Systemes learning system, Companion learning space will be explained and also different theories on how to write a pedagogical manual.

2.1 CNC machines

Machining is a conventional method using sharp cutting tools to physically remove material to achieve a desired geometry. It is a relatively fast method and can be used on many kinds of material. There are different kinds of machining as milling, turning, drilling, probing, sharpening etc. The first milling machining operations was as early as 1814-1818, the machines at that time was manually driven and could only provide easier operations. The milling machines evolved and got more complex and the big breakthrough and the beginning for a new era was in 1940 when CAM was implemented in the machines. But it was not until 1952 when the first NC machine could be presented. The development continued and in 1960-1970 CNC machines were created. This step made it a lot easier to create complex geometries and thanks to CAM programs you could now create 5-axis surface operations. Milling machines today can come in many different sizes and shapes, from small sized bench-mounted types to huge room-sized machines [1].

2.1.1 3-axis milling machines

The most common milling machines are 3-axis, where the table is the X and Y axis and the Z axis goes through the head, most of these machines look the same, like the one in figure 2.1, except from size and production year. The greatest advantage of using 3-axis milling machines instead of a multi-axis machine is that the cost of a 3-axis machine is usually much cheaper and the machining time can be increased. The disadvantage of using 3-axis machine is reduced ability to create complex parts without remounting the part. 3-axis milling machines can also be converted to become a multi axis machine with an adapter with two rotation axis A and B [2].



Figure 2.1 3-axis milling machine [1]

2.1.2 Multi-axis milling machines

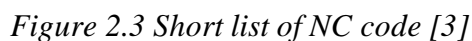
Then there are also multi axis machines, these can be from 4 up to 9 axes but the 5-axis is the most common one. The 5-axis is mostly built up the same as the 3-axis except from two extra rotation axes. But the 5-axis can be much more complex than that, the head could be placed like a traverse with the head moving through X and Y axes and the rotation axes on the head as well as illustrated in figure 2.2. These are made for big complex geometries like a full scale model of a car. Another common type is where the table is as it were on the 3-axis but the difference is that the head is placed on the rotation axis. This type is very common for smaller and very complex parts as it allows the head to move freely minimizing the collision risk when the space is very tight. The biggest advantage of using multi-axis machines is that the complexity of the part can be much greater. To cut down the machining time more axis than necessary should not be used.



Figure 2.2 Multi-axis machine [2]

There are a couple of different ways to program a CNC machine. Either the programming can be done manually or a CAM program can be used to build up the machining process. There is a lot of different software to use today, not only to build up the machining process but also to verify that the program works.

Easy geometries in 3-axis machines can manually be written down by NC codes directly in the machine. This is only possible for 2D geometries and takes quite a long time to write down each coordinate, this also requires the programmer to know the parameters for the specific CNC machines and to have knowledge in the different NC codes. Depending on the control system of the machine different types of codes and parameters are needed. An example of how a NC code could look like is shown on figure 2.3. The code is on the right side and the geometries that are to be computed on the left. A code contains tool changes, start positions and coordinates whom the tool follows and so on.



2.2.2 Computer-aided manufacturing

As the geometries and the CNC machines become more and more complex there is need for programs to be able to manufacture these products. It is no longer possible to compute the codes manually if you want to create a 3D geometry. To be able to do this you need computer software, a CAM program. A CAM program allows you to plan and program the tool paths both for extremely complex geometries, like in figure 2.4, but also for easier geometries and creates a faster and more efficient production. Tool paths, energy consumption and parameters can be analyzed directly in the program. To program geometry in CAM you need to import a computer-aided design (CAD) model and then choose appropriate tool paths and strategies for the part. This means that even though it is computer aided software it does not eliminate the use of highly skilled manufacturing engineers or NC programmers. In fact it has the need of highly skilled professionals thanks to the variety of complexity on the products, the visualization, simulation and optimization tools [3].

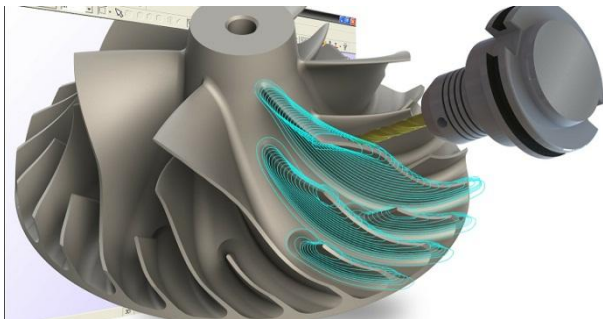


Figure 2.4 Tool path replayed in CAM

Some popular CAM software today is: Pro/E, SolidCAM, MasterCAM, NX, CATIA V5 and DELMIA V6. Some of these programs only have the ability to visualize tool paths and others have the ability to visualize the whole machine process with collision checking.

When the NC code is generated by the CAM program it is usually an automatically programmed tools (APT) code, a cutter location data (CL) and also international standard organization (ISO) code. These data is general and does not include parameters for the CNC machines control system. To convert this data to specific instructions for the machine there is need for a post-processor. Different machines may have different specific instruction for their control system and because of that the post-processor must be tailored for the specific machine [4]. The post-processor is the direct link between the CAM system and the CNC machine and therefore a vital part in machining which is illustrated in figure 2.5.

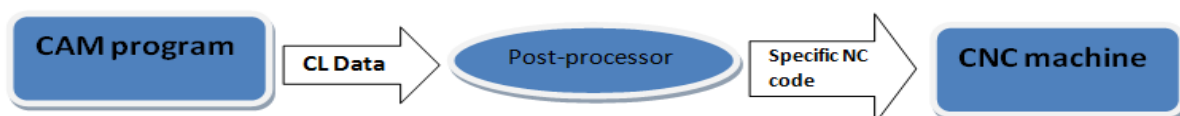


Figure 2.5 Illustration on the process between CAM and CNC machine

2.2.3 Methods to verify

Before producing a part with the generated NC code it has to be verified. The reason to verify the code is to detect inefficient motions or error that could ruin the part or the machine which can be very costly. Another reason to verify the code is to optimize and reduce the cycle time. The verification can be done either in the CNC machine or with software on a computer.

Verify in the machine

If the code is generated by a manually written program it is probably not a very complex program, then verification can be done directly in the machine. This can be done in two ways; the first option is if the CNC machine has the capability to display the tool path from the NC code, in this way unnatural tool paths can be detected and corrected. Another way to verify the NC code is to test drive it directly in the machine, this is done by slowing down the feedrate in order to avoid collisions. It can also be done with a material that is easier to machine or in the empty space. This method is a good way to verify the tool path but it is hard to optimize and reduce the cycle time without using any kind of verification software. It also requires an available machine which could be costly when no production is done.

Verify in computer

In order to analyze and verify the NC code there are different programs for this, VERICUT is an example of this. These programs are a great help to verify and optimize the tool path. It is also able to make shop floor simulations and detect collisions during the operation. Building up the machining process in one program and verify and simulate it in separate program is time consuming. That is why some programs, including DELMIA V6, have the capability to verify the program built-in. This makes it possible to build it right from the beginning which reduces the time and maximizes machine tool utilization [5].

2.3 Pedagogical learning process

For companies that sell their products worldwide it is important to have an efficient learning system. The easiest way to do this is to use e-learning literature to allow the user to access the information worldwide. There are some things to consider when creating a learning manual explained in this chapter, Dassault Systemes e-learning system Companion is also explained.

2.3.1 Companion

Dassault Systemes have a lot of brands and cooperates with many different industries. Companies require employees to continuously learn and perform at lowest possible cost. That is why Dassault Systemes have worked with instructional design experts and costumers to create a flexible and open learning system. Today Dassault Systemes offers over 1000 companion courses, covering all brands and industries for every profile, end-users, managers, designers and so on. Companion learning space allows companies to design and assign personal learning programs and analyze user performance. Among the companion courses for DELMIA V6 there are *DELMIA NC Machine Builder*, *Prismatic Machining*, *Milling machining* and *Extended Milling Machining*. Below in figure 2.6 is an example on a screenshot from *Prismatic Machining Companion* [6].

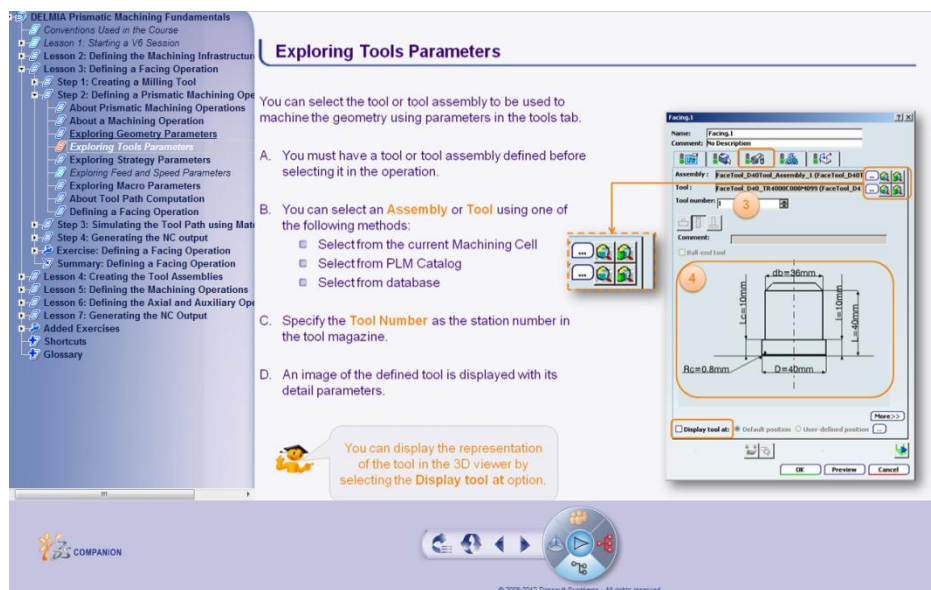


Figure 2.6 Example on how the Companion look

The Companion courses are built up with different lessons, divided in different steps. The lessons are a rough separation of the content in the course. The lessons are separated in different steps where the first steps explain all the theory and every lessons ends with an exercise on all the theory. After the exercise there is a summary of the lesson.

2.3.2 Writing a manual

In order to write a pedagogical and efficient manual it is very important to know the product inside and out. To be able to explain all steps the writer of the manual has to have both practical and theoretical understanding of the product. An important thing is to assume that the reader does not know anything about the process, keeping this in mind makes it easier to not forget any steps or leaving any gaps. To make the manual easier to understand it is important to use diagrams and pictures to explain things that cannot be explained in words. Making the user understand why every step is done helps the user to understand the process. If the user understands the purpose of each action it makes the user not skip any steps. By breaking down the process into small steps presented in natural order, being sure that every step leads to the next, the user easier remember the working flow. The language in the manual is important to prevent misunderstandings, it is important to write in a way that only can be understood in one way. To make sure there are no misunderstandings or anything missing in the manual it is always a good idea to test the manual. Letting sample groups work through the manual is one way to get feedback to improve the manual [7].

2.4 DELMIA V6 interface

To understand the structure and how the program works an introduction to the Version 6 is defined. The special part with the V6 is that everything created is tied to a database, this means that no files are created. Without files you always have the latest update on the product because everyone works toward the same database. The menu in figure 2.7 is always available on the bottom of the window. Number 1 is the menu where the workbenches are. Number 2 is the search function, by typing a search string you find your results directly in the database. Number 3 is propagate, when saving a change to the database you simply click on propagate and the product will be updated.



Figure 2.7 V6 menu: 1) Workbenches, 2) Search engine, 3) Propagate

2.4.1 The machine programming interface

There are two main lists in the machine programming workbench, in the Product Process Resources (PPR) tree everything imported is listed, illustrated to the left in figure 2.8. Product, parts, fixture, CNC machine, tools and NC code are all listed in the PPR tree. In the Activities Process Tree all created operations are listed, to the right in figure 2.8.

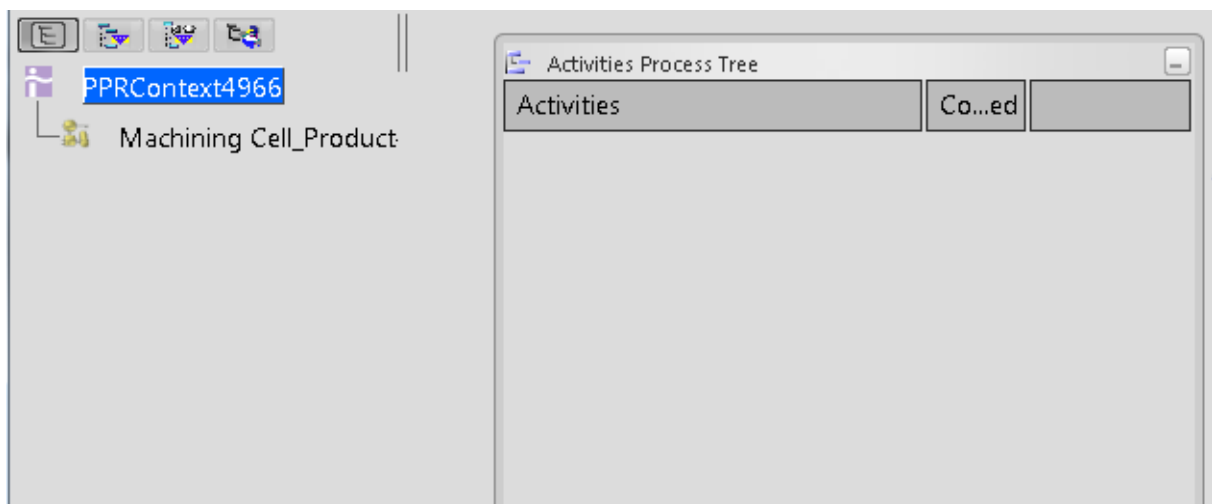


Figure 2.8 PPR tree and Activites Process Tree

3 METHOD

In this chapter the methods used during the project are explained. The working process can roughly be divided into three steps; acquire knowledge of how DELMIA V6 works, use this knowledge to create a milling machine and perform a full 3- and 5-axis CAM preparation in DELMIA V6 and finally to document every step in a pedagogical way. The process can be visualized by figure 3.1:

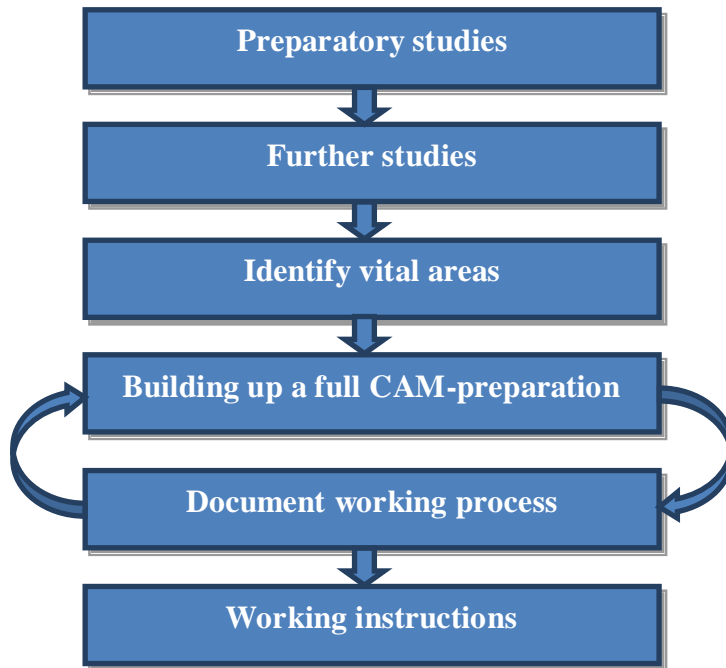


Figure 3.1 Working process

3.1 Acquire knowledge

To gather the required knowledge preparatory studies in DELMIA V6 and design tutorials in CATIA V5/Machining have been done. The next step in the process is further studies in France on how to perform CAM preparations in DELMIA V6.

3.1.1 Preparatory studies

To gain a basic knowledge of how CAM preparations in DELMIA V6 are made, self-studies in CATIA V5 and DELMIA V5 were made. This preparatory work was done using examples in CATIA- and DELMIA-help files. The modeling of the machine during this project was made in CATIA V5 and assembled in DELMIA V6. Also some basic examples in how CAM preparation in DELMIA V5 works were studied. The structure in DELMIA V6 is different from DELMIA V5, to understand the structure and how the new software works there was an introduction in DELMIA V6 at Dassault Systemes in Gothenburg.

3.1.2 Further studies

Due to the lack of competence in Sweden about how CAM preparations work in DELMIA V6 a one week course was obtained in France. The course content was divided into *Prismatic Machining*, *Milling Machining*, *Extended Milling Machining*, *Machine tool builder* and *Program Simulations* in DELMIA V6 was obtained during a one week course in France.

Prismatic Machining allows you to create 2.5- and 3- axis operations. Programmers are immersed in a V6 3D environment that allows them to create, optimize and validate their prismatic machining programs in the context of the physical workplace. The CAM features included are work piece set up, cutter definition and tool path simulation with material removal.

Milling Machining allows you to create 3- and 5-axis milling operations in a virtual environment. The element includes basic machining but also the ability to program operations for 3D parts that require advanced 3-axis milling, including the ability to switch to 5-axis motion.

Extended Milling Machining is an extension to *Milling Machining* and fills the needs of all industrial processes. It makes it possible to machine even the most complex parts as a set of multi-axis strategies as spiral milling, tube machining, curve following, flank contouring, helix and interpolated axis along isoparametric mesh.

Machine tool builder is a tool to build NC-machines. In the machine tool builder all parts in the NC-machine are put together and kinematics are defined. In order to make the virtual machine work the same way as the physical machine lots of parameters can be defined e.g. maximum feed rate, home position and travel limits. To generate NC code it is not necessary to build the machine that will be used, there is an opportunity to use a generic machine. There is a huge disadvantage to use a generic machine because it is not possible to do a full validation if the machine is able to produce the part in the planned way.

Program Simulations is a tool to perform virtual NC program validation. Programmers using this tool can control and simulate machine tool motions, along with material removal, using either the NC tool path or post-processed NC code. The *NC Machine Simulation* is a tool that delivers optimized, high-quality NC programs by finding potentially damaging collisions, excessive non-value-added machine motion, out-of-travel machine motion, or wrong material removal.

3.2 Work in DELMIA V6

The first step of the work in DELMIA V6 was to build the CNC machine located at Chalmers Lindholmen. This was done using the parts modulated in CATIA V5 during preparatory studies. In this step machine kinematics, tool and work-piece mount points, travel limits, home position and axis names were defined. When the machine was finished the knowledge in how to structure the work and build up the operations was obtained and the most vital areas

were defined. This was done by building up a full CAM preparation using a 3D model of the Iron Man helmet. After the most vital areas were identified, a part was modulated to build up a CAM preparation with all the important steps and document this in a pedagogical way creating a step-by-step guide.

3.3 Documentation

To be able to transfer the knowledge of how the work in DELMIA V6 is done the documentation is a vital point. To easily understand and learn the program an appendix with a step-by-step exercise was created. There are a lot of different ways and strategies to document the process. After a brief research a list of seven tips for writing an efficient instruction manual was set as base for the documentation. First tip is starting from square one, the manual have to be adapted to the audience. Assuming the audience will have zero knowledge of the subject matter is a good starting point to not write a too complicated manual. Second tip is to know exactly what your manual needs to cover; otherwise it is easily done to give too much information which can be confusing. Before start the writing, a high-level outline of what the manual will cover is a good help to structure the manual. Using lists for doing something makes it easier to work through the manual step by step. Using screenshots in the manual also helps the user to understand the workflow. The last tip is to work through or let someone else work through the manual to get feedback in order to fine tune the manual [8].

4 WORKING PROCESS DELMIA V6

During the one week course in France a methodology of four steps to build a full CAM preparation was obtained.

1. *Assembly*, assembling the part with the fixture.
2. *Setup*, mount the assembly in a machine.
3. *Operation*, building up the operations.
4. *Verify and generate NC-code*, verify the tool path, collision checking and generate NC-code.

During this work one more step had to be done, building the machine. That is why the content in this chapter is divided in five sub chapters. To fulfill these steps all parts to the machine were modulated and all parameters like travel limits, feed rates and so on to the machine collected before the start of this project. To make a CAM preparation a part to machine is necessary to have. The part chosen to the project was the Iron Man helmet, shown in figure 4.1, and was downloaded from the free 3D CAD library grabcad.com.



Figure 4.1 Iron Man helmet [4].

The reason why the Iron Man helmet was chosen is that it is something many people recognize and it allows a variety of different operations. To make the part easier to machine some changes in the surfaces were done and a neck to mount the part in the machine was added.

4.1 Building the machine

Building an NC-machine is done in two steps. The first step is defining the kinematics of the machine, which is where the definition of how the machine moves is done. When the kinematics of the machine is defined it is transformed to an NC resource by defining all features for the milling machine.

4.1.1 Defining kinematics

Defining the kinematics of the machine can be done in two different ways. In order to investigate which way is easiest to use, the machine kinematics were built in both ways.

Alternative one: Building kinematics with coordinate systems.

Building the kinematics with coordinate systems was done in the assembly design workbench. First of all the NC machine parts was imported and placed like they should be mounted. The fundament is not supposed to move and was fixed with the engineering connection *Fix* as shown in figure 4.2.

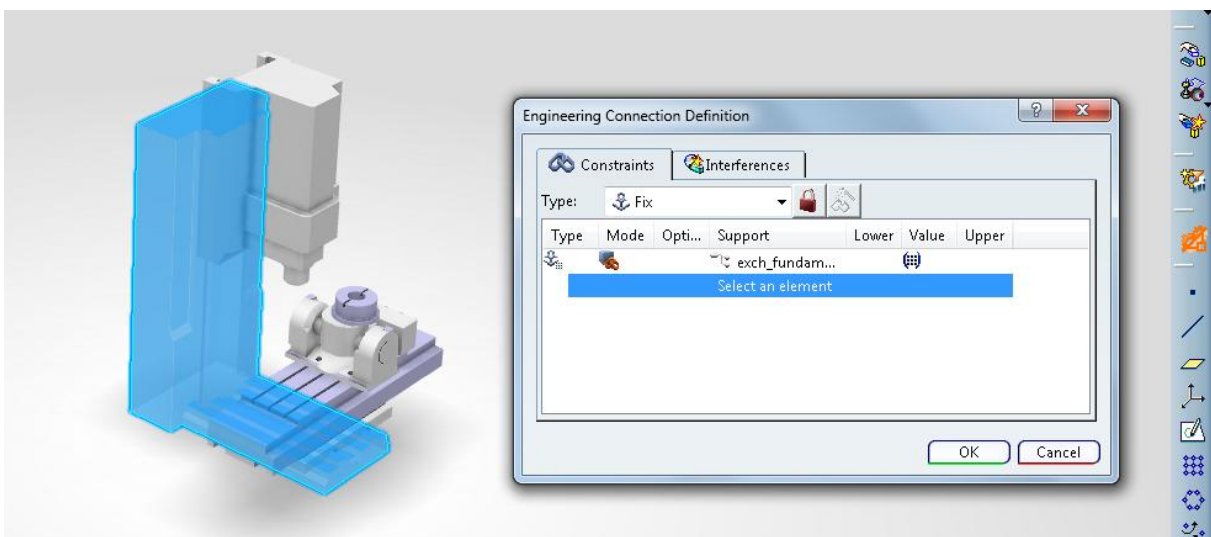


Figure 4.2 Creating a *Fix* engineering connection to the fundament.

The engineering connections between two parts were a little bit more difficult to define. To simplify the work with the engineering connections axis systems with the Z-direction in the positive moving direction with the origin in home position of the machine were placed when the parts were modulated. A prismatic engineering connection between the axis system on the fundament and the Y-table were defined in figure 4.3. The prismatic connection was defined to follow the Z-axes between the two axis systems, have a contact between the two ZX-planes and a 0 mm offset between the two XY-planes.

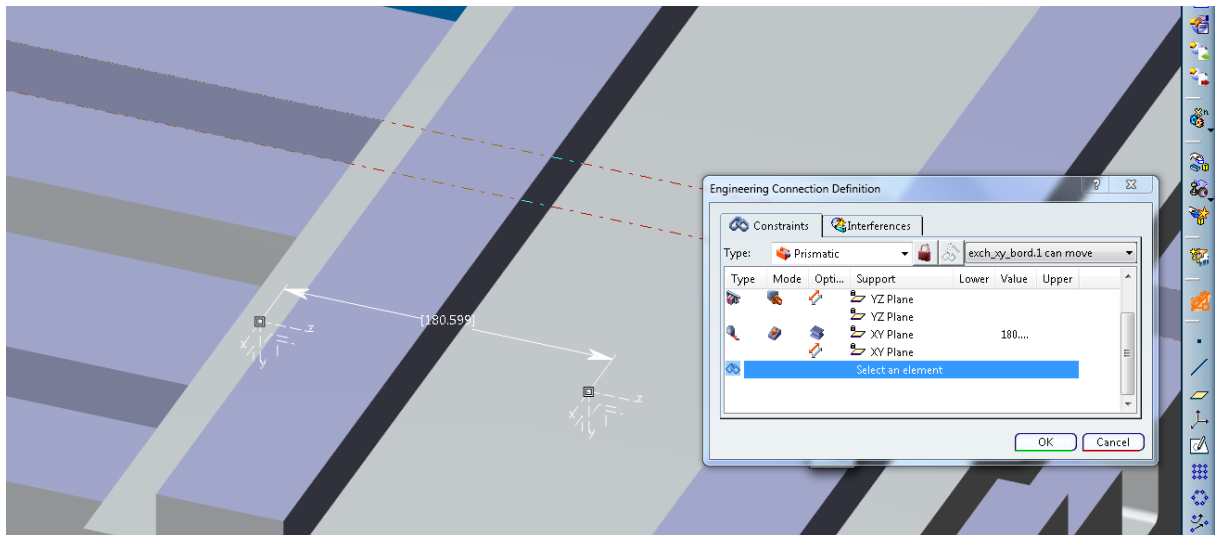


Figure 4.3 Creating a Prismatic engineering connection

In the same way the engineering connections between the X- and Y-table and between the spindle and the fundament were done. To attach the AB-fundament to the X-table a rigid engineering connection between them was set.

The rotating parts, B-axis and A-axis, were problematic to define generic connections for. The revolute connection was used with the Z-axes as rotation axis, the XY-planes as connection and a 0mm offset between the two YZ-planes between the AB-fundament and B-axis, illustrated in the picture 4.4. What was difficult with this step was to get axis systems right because the positive direction for the CNC-machine will be from the X- to Y-axis when the Z-axis is used as rotation axis illustrated with the arrow in Figure 4.4. The same process was done to connect the A-axis to the B-axis.

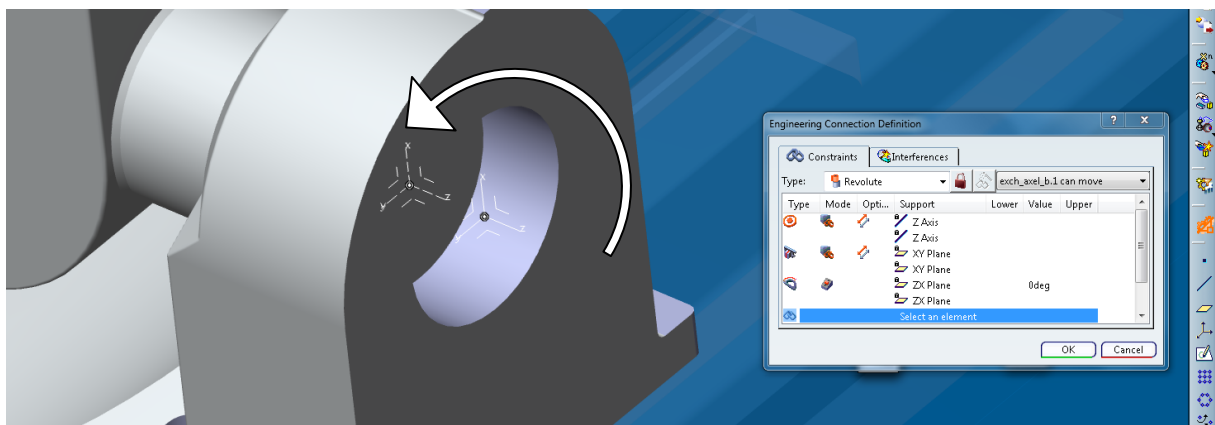


Figure 4.4 Revolute connection on rotating part

When all the engineering connections were done a mechanical representation was done, to be able to create a machine. The representation was named and a container for the joints and commands was created by clicking on *Mechanism representation* shown in figure 4.5.

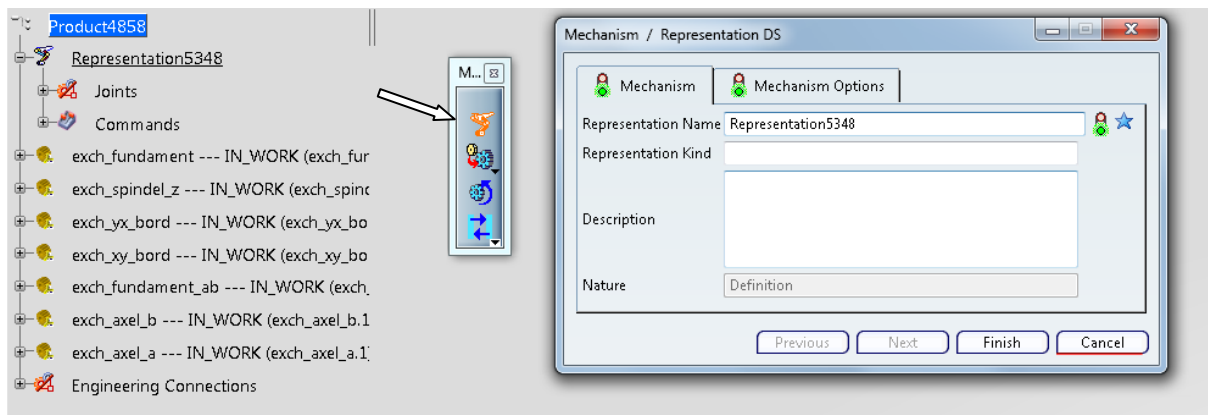


Figure 4.5 Mechanic representation created

Alternative two: Building kinematics with Robotics Device Builder

This method was done in two steps. First all the parts were imported and mounted together with engineering connection in the assembly design workbench. In order to build the machine in the home position the axis-systems were mounted together with coincidence engineering connection, illustrated in figure 4.6.

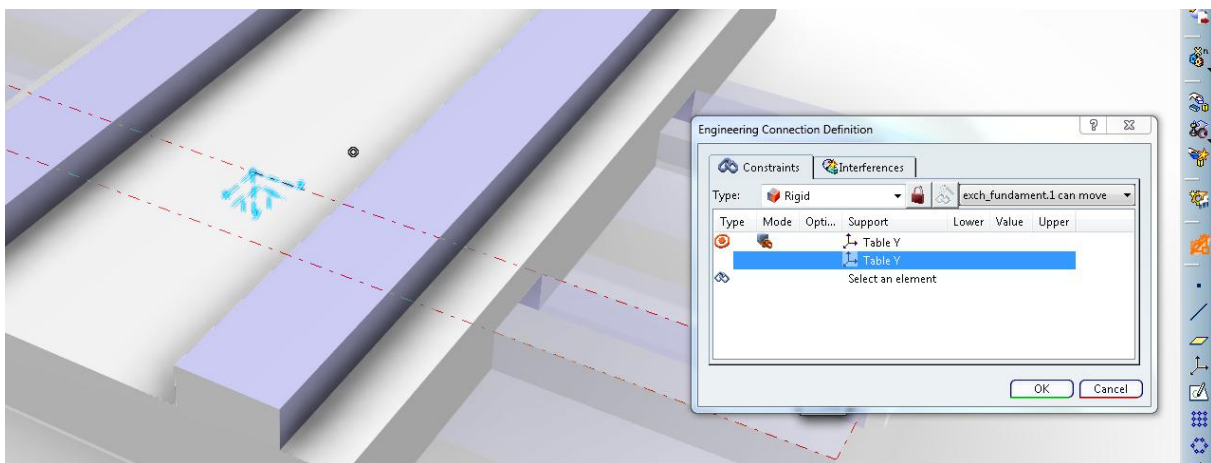


Figure 4.6 Building the machine in home position

When all the parts were put together in the home position the engineering connection were changed to kinematic connections, in order for the machine to be able to move. This was done in the Robotic device builder workbench marked with number 1 in figure 4.7. The engineering connections defined in the assembly design workbench were deleted, as illustrated with number 2 in figure 4.7, because they were only defined to build the machine in its home position. Instead kinematics was defined using the Define kinematics button in the west quadrant of the compass marked with number 3 in figure 4.7. First a fixed joint was created on the fundament, marked with number 4 in figure 4.7.

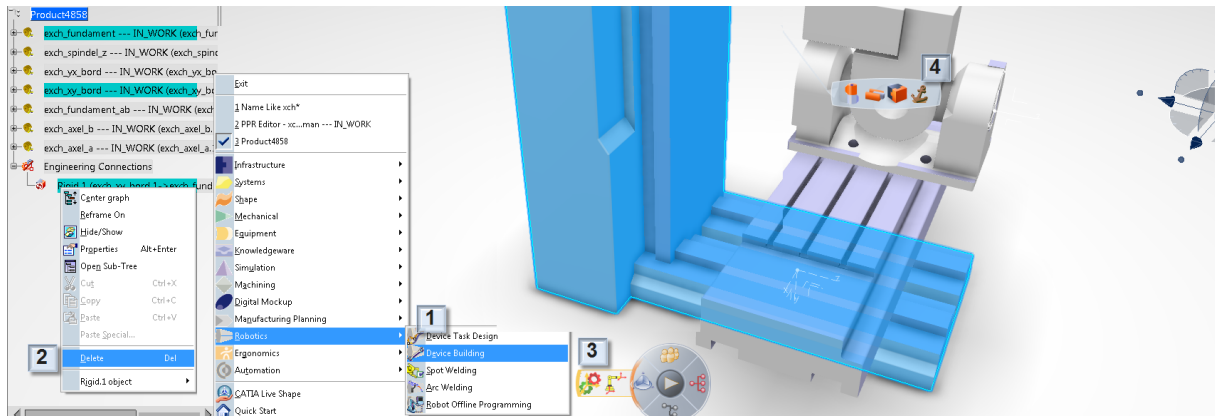


Figure 4.7 Robotics Device Building illustration

The kinematic between the Y-table and the fundament was defined with the same button, but instead of the fixed joint a prismatic joint was defined. In what direction the Y-table will be able to move is defined by the green line and the positive direction for the machine by the arrow as seen in figure 4.8.

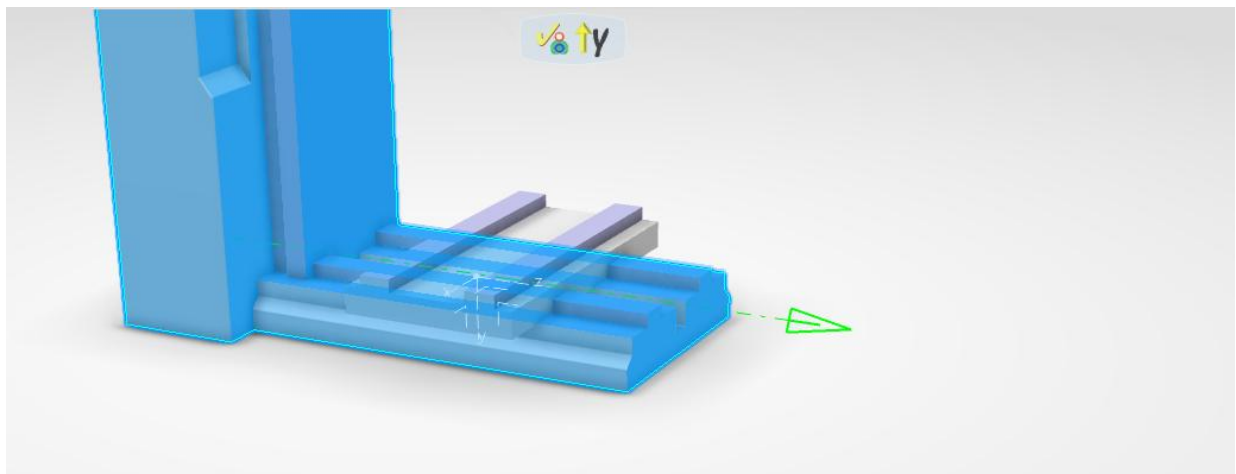


Figure 4.8 Selecting positive motion direction

In the same way the engineering connections between the X- and Y-table and between the spindle and the fundament were done. To attach the AB-fundament to the X-table a rigid engineering connection between them was set.

The kinematics between the B-axis and the AB-fundament was set with a revolute joint. The green axis indicates the rotation axis and looking in the direction of the arrow, the positive rotation of the axis will be clockwise, from x to y, illustrated in figure 4.9. The same way the kinematics of the A-axis was defined.

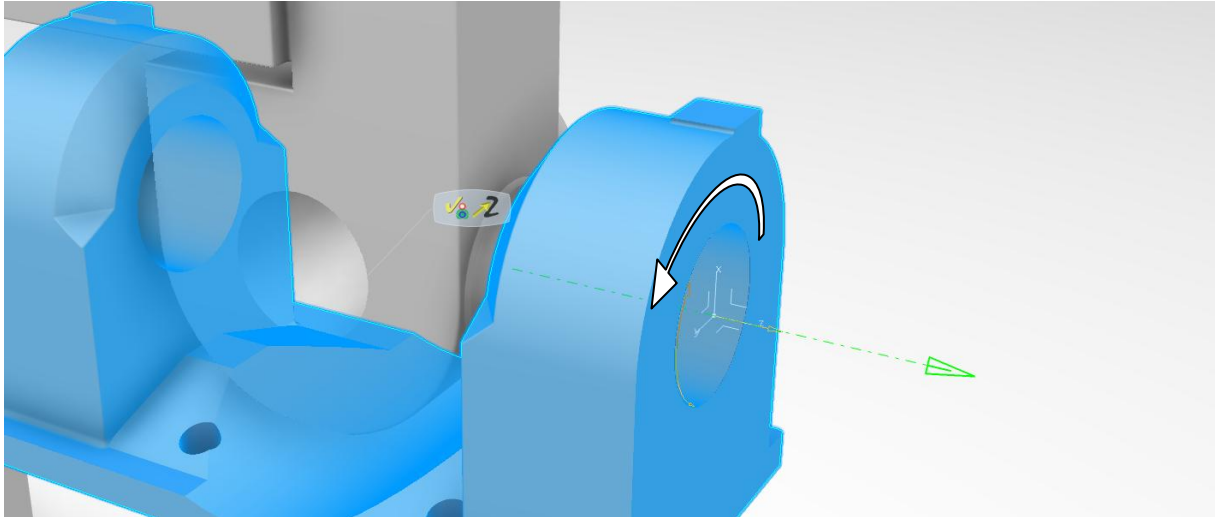


Figure 4.9 Setting rotation direction

4.1.2 Machine tool builder

When the kinematic of the machine was fully defined there was only one step left to complete the building of the machine, define it as a machine and set all parameters. This is done in the machine tool builder workbench. First of all the machine was defined as a milling machine as illustrated in figure 4.10.

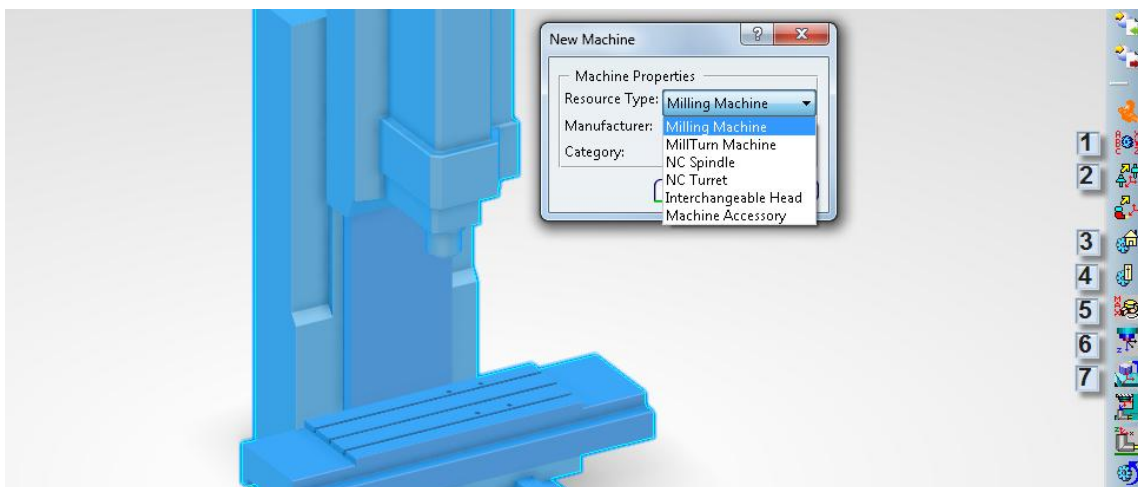


Figure 4.10 New machine created

All the parameters to the machine were set using the buttons in the machine building toolbar. The name of the axes were defined using icon one, this was done to easier separate which axis is which. The tool change position was defined with icon two so the machine knows in which position the machine swap tools. The home position, where the machine stands in default was set using icon three. To make sure that the machine does not operate outside its boundaries the travel limits of the machine was defined using the fourth icon. The acceleration and maximum speed of the machine was set in the fifth icon in order to make the machine move in the same way as the real machine. Where to mount the tools and the work piece was defined with the sixth and the seventh icon.

By right-clicking on the machine and select Edit NC Resource a new window popping up as illustrated figure 4.11. In this window settings for postprocessor and tool catalog for the machine is set. For the machine built there is no postprocessor defined yet, which is why either a postprocessor or a tool catalog was assign in this step.

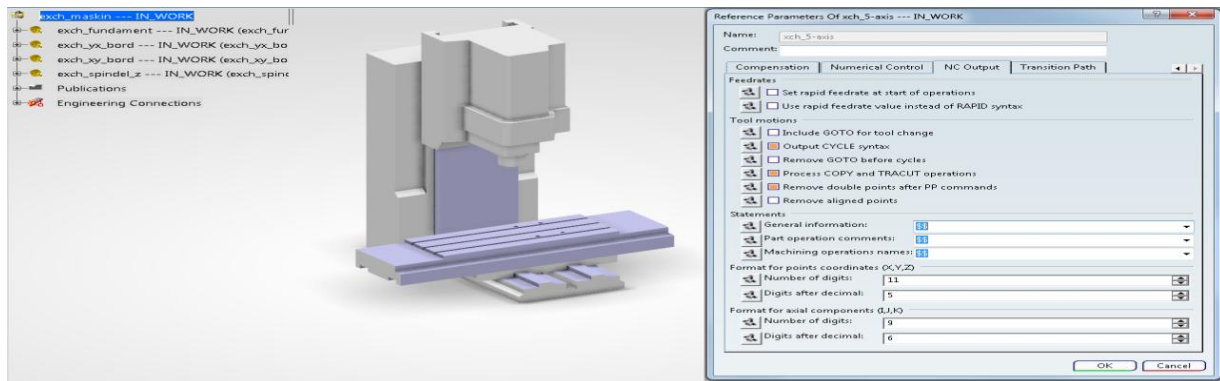


Figure 4.11 Postprocessor and tool catalog is assigned

When this step was done all the parameters and settings for the NC-machine were defined and the machine was ready to use.

4.2 Creating the assembly

To be able to machine the Iron Man helmet part it had to be mounted into the machine. In order to do this some kind of fixture to hold the part in the machine had to be mounted on the part. Also an axis system for the mount point had to be created.

Mount fixtures on part

This was done in the assembly design workbench, the Iron Man helmet part was imported together with fixtures as shown in the figure 4.12. The fixtures were mounted on the part with a coincidence connection and an offset of 1.6 mm in order to mount the fixtures in the middle of the part.

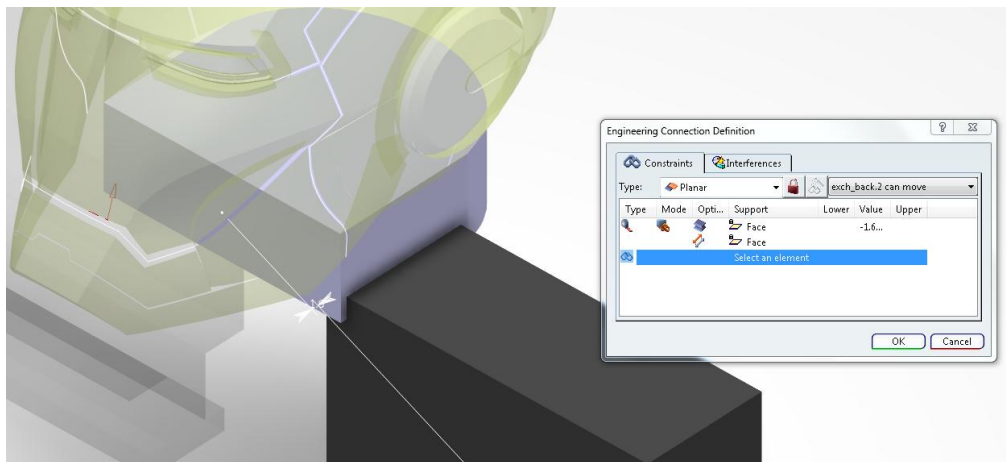


Figure 4.12 Fixture mounted on the part

Insert axis system for mount point

When the fixtures are mounted on the part there is only one moment left in creating the assembly, insert an axis system for the mount point. This is done to ease the mount of the assembly in the machine. The axis-system is placed in the midway between the fixtures in the same level as the bottom of the fixture as illustrated in figure 4.13.

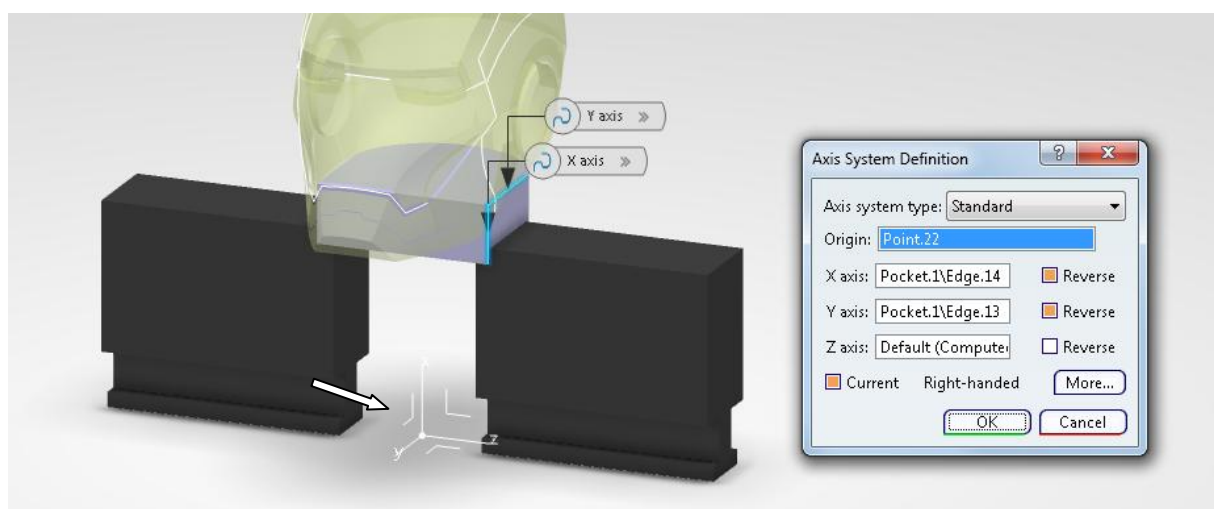


Figure 4.13 Axis system for mount point assigned

4.3 Setup

To start working with the actual machining the assembly has to be inserted to a PPR context. The PPR context contains the products or parts required to be machined, in this case the assembly with the Iron Man Helmet part and the fixtures. Also the PPR context contains a machine cell that includes the NC machine and tools that is used in the machining process. To create a PPR context the assembly was opened in the machine programming workbench. A new PPR context was automatically created with an empty machine cell as shown in figure 4.14.

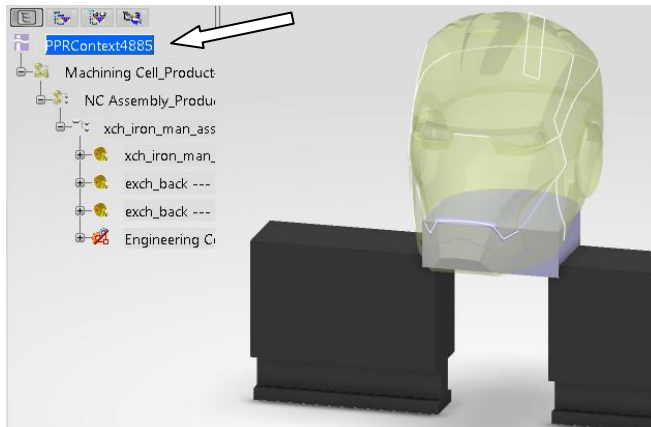


Figure 4.14 The PPR context created

Create a generic machine

Before creating a machining activity a NC machine has to be assigned to the machining cell. This can be done in two ways, either a generic machine can be created or a machine can be assigned from the database, like the one created in chapter 4.1. A generic machine is assigned using the Create generic machine functionality shown in figure 4.15. 3-axis, 3-axis with rotary table and 5-axis machines can be created using this functionality.

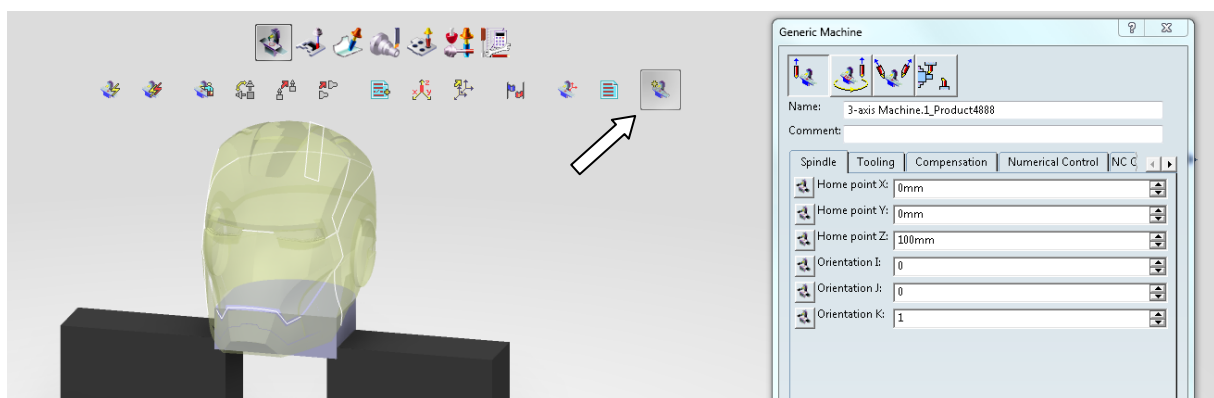


Figure 4.15 A generic machine is created

All the settings that were set in the machine tool builder can be set here to get a realistic simulation. There is one disadvantage of using a generic machine; it is not possible to see if there are any collisions in the machine during the simulations.

Assign machine from database

To assign a NC machine from the database the Assign a machine, from database icon was used, circled in figure 4.16. When clicking on the button a window popped up where the machine created in chapter 4.1 searched up and assigned.

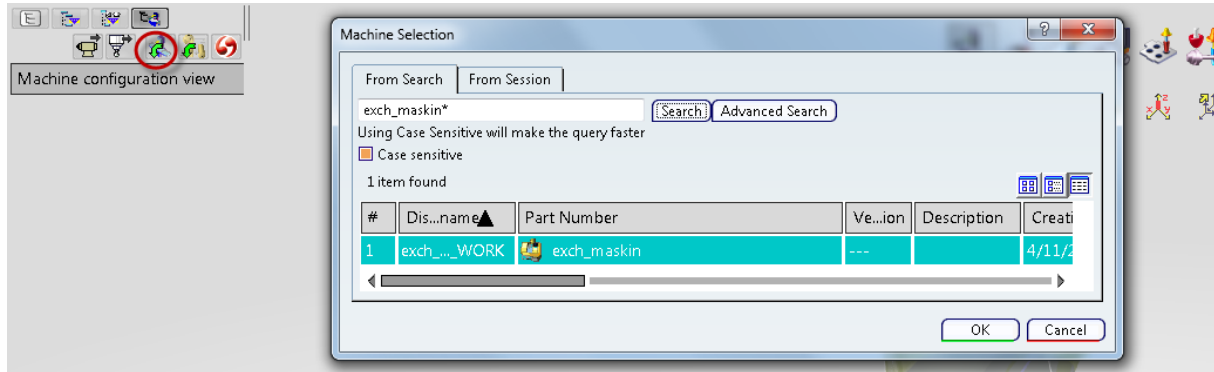


Figure 4.16 A machine is assigned from the database

When a machine is assigned a window with the activity process tree, where the operations built up in the chapter 4.4 will be gathered, automatically is created.

Mount assembly in machine

To mount the assembly in the machine Workpiece advanced mount functionality marked with number one in the figure 4.17 was used. When clicking on the icon a new window with the assembly appeared. By clicking on the Mount the assembly icon, marked with number two, in the window and selecting the axis-system placed on the assembly in chapter 4.2 the part was mounted in the machine.



Figure 4.17 The assembly is mounted in the machine

Create a rough stock

To machine the part the starting geometry, the rough stock, has to be defined. This was done using the rough stock creation in the utilities menu shown in figure 4.18. In the window for rough stock creation the Iron Man part is selected as the destination and part body, this automatically creates a box by the outer geometry of the part.

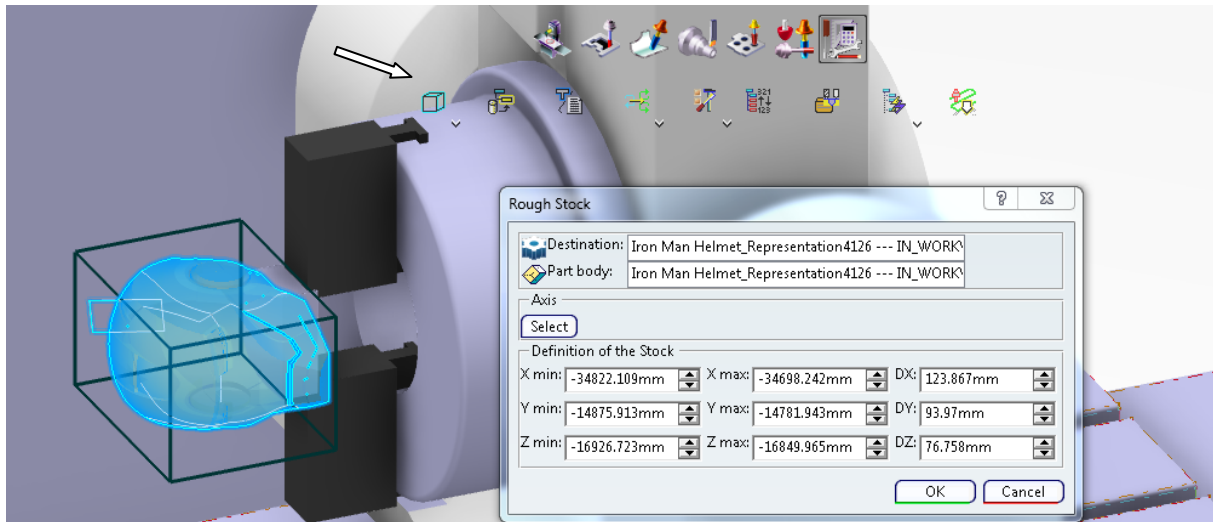


Figure 4.18 A rough stock is created

Set the rough stock, fixtures and design part

By double-clicking on the Part operation in the Activities process tree the part operation dialog box is displayed, shown figure 4.19. In this window the reference machining axis system is defined. To make the machine understand what parts that shall be machined the design part is defined as the Iron Man part. The rough stock is set as the earlier generated rough stock. The fixtures added on in the assembly shall not be machined and are set as the fixtures.

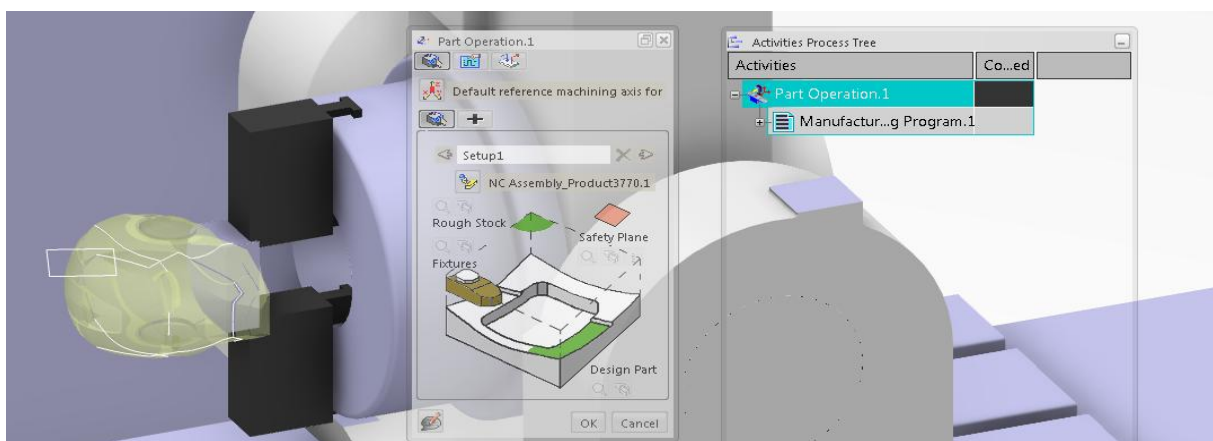


Figure 4.19 The rough stock, fixtures and design part is set

Activate intermediate stock and collision checking

In the second tab of the Part operation dialog box intermediate stock is activated shown in figure 4.20. This option makes it possible to after every executed operation generate a new stock with the shape of the part after the operation. Also collision checking on design part and fixtures was activated to be able to identify collision. To check that the machine does not work out of the defined kinematics the kinematic checking is also activated.

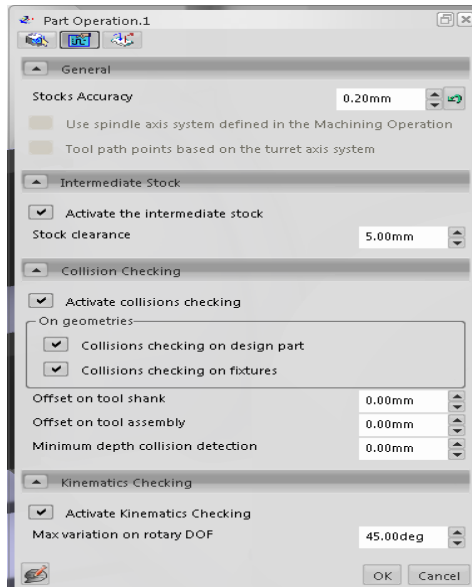


Figure 4.20 Activating intermediate stock and collision checking

Create and assemble tools and tool holders

To machine a part it is necessary to have a tool. Tools can be imported, the machine can have a predefined tool catalog or the tools can be created. The tools used to machine the Iron Man helmet had to be created; this was done using the Tool builder toolbar. Building the tools is done in three steps, first create the tool, than create a tool holder and finally assemble them together. Creating the tool and the tool holder is done in similar way, first select which kind of tool/tool holder that shall be created in the menu and set the geometry in the dialog box as shown in figure 4.21.

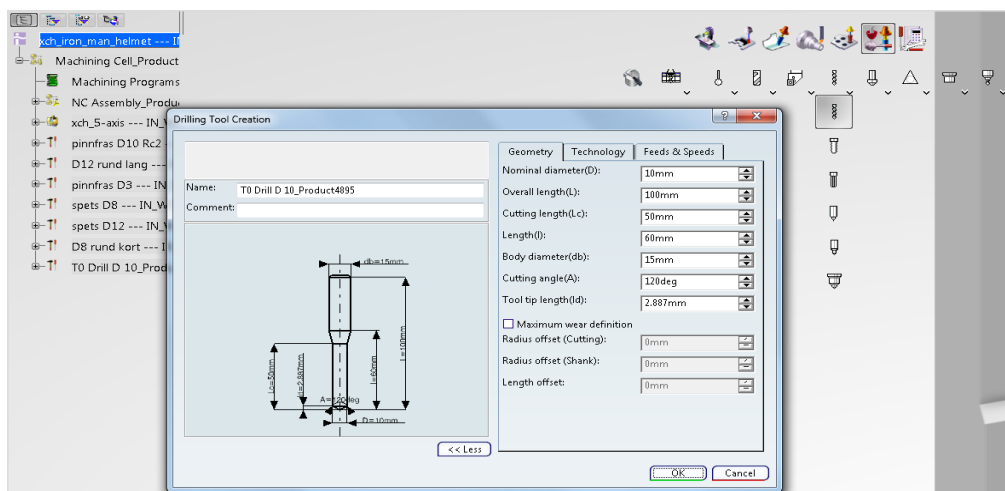


Figure 4.21 Tool created

When a tool and a tool holder are created they have to be assembled. To assemble the tool and a tool holder the tool assembly functionality was used, illustrated in figure 4.22. When clicking on the icon a new window pops up, the tool and the tool holder was imported and assembled.

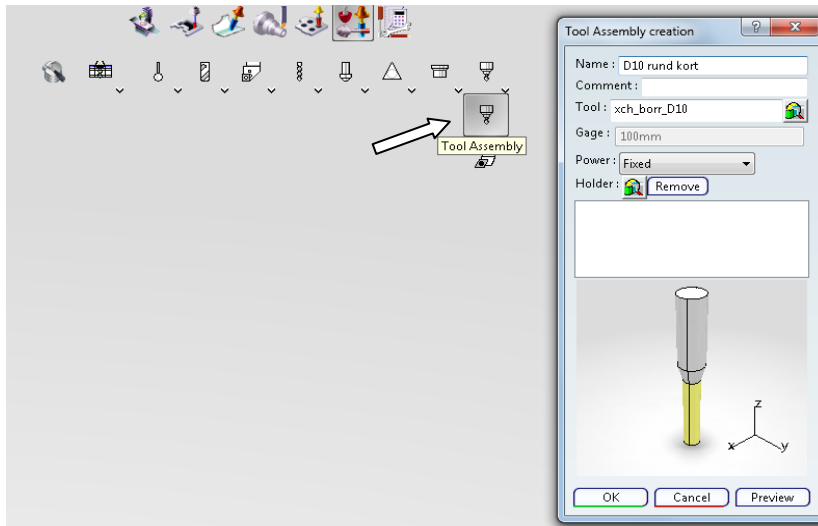


Figure 4.22 Tool and tool holder assembled in tool assembly creation

Create a tool catalog

In the same way all the tools to the machine the part was created. When the tools are created they automatically linked to the machine in the PPR context shown to the left in figure 4.23. The tools can be grouped in a tool catalog. By right-clicking on the tool assembly and selecting Send to tool catalog under NC-recourse a new window appears. In this window a new catalog was created and all the tools imported.

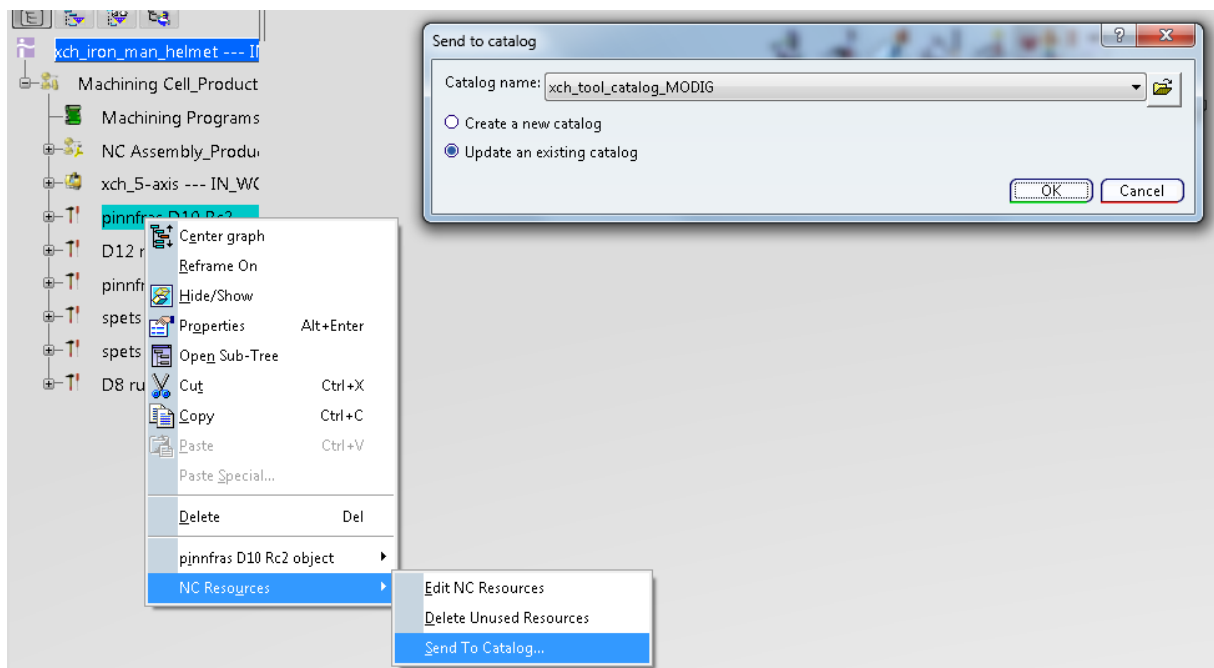


Figure 4.23 Tool assembly sent to tool catalog

Check reachability for tools

To check the reachability of the tools the Workpiece advanced mount functionality, used to mount the assembly to the machine can be used. By clicking on the check reachability icon as shown in figure 4.24 and selecting a tool it is shown in the window. By dragging the tool around it automatically moves the machine to the same position. When a collision is detected or the kinematics are exceeded the tool turns blue. This is a good way to get an overview of which tools can be used to machine which surfaces of the part.

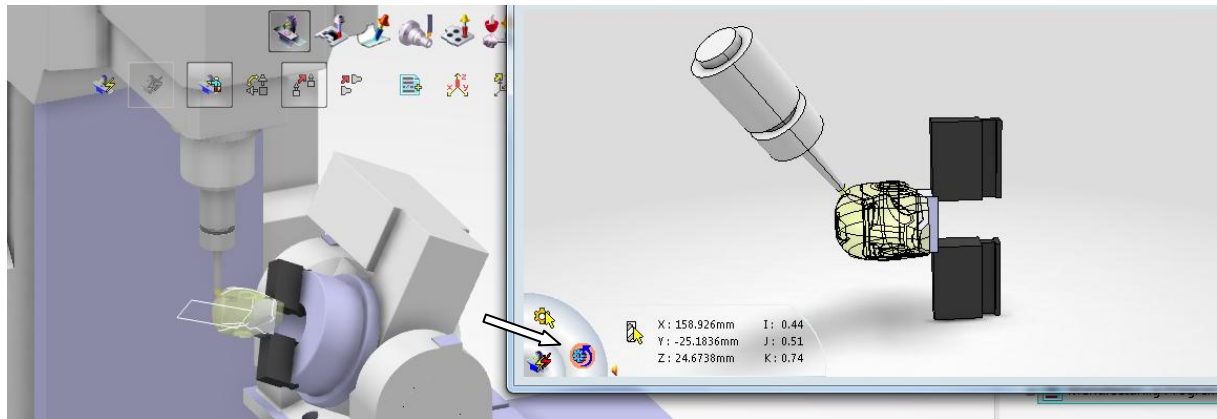


Figure 4.24 Checking the reachability using workpiece advanced mount functionality

4.4 Operation

Milling operations in DELMIA V6 Machining are divided in three categories; Prismatic Machining Operations, Surface Machining Operations and Axial Machining Operations. In these main categories there are sub categories and then the machining operations. Thanks to the categories there are many different operations available. The placement of these with a transparent menu, which can be seen in figure 4.25, gives the user a smooth and user friendly interaction which makes the required operation easy to find and select. Each created operation is placed in the Activities Process Tree and allows the user to easy manage the different operations.

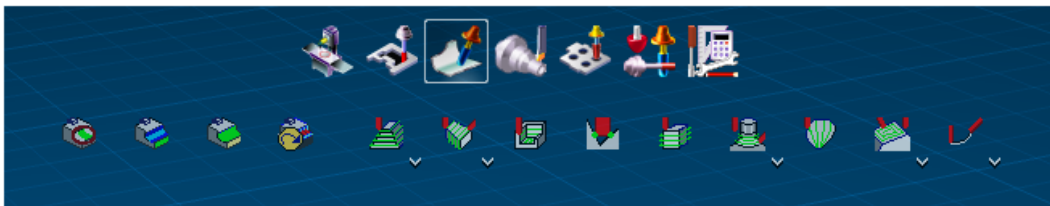


Figure 4.25 Machining menu

When creating a new operation a dialog window appears, in common for all kinds of operations are that the Parameters Tab looks the same. As you can see in figure 4.26, number 1 is the *Strategy tab*, here there are a couple of sub tabs where you define suitable parameters for machining strategy, radius strategy, axial strategy, tool axis etc. The number 2 is the *Geometry tab*; here the desired geometries are defined, limiting contours, which part to machine, bottom, top, check element etc. Number 3 in the picture is *Tools parameters*, here you select which tool or tool assembly you wish to use, you can choose one from a tool catalog or one from the database. Number 4 is the *Feedrate and spindle speed parameters tab*, required feeds and speeds are defined in this tab. Number 5 is the *Macro parameters tab*; in this tab approach, retract, linking and clearance motions are defined, all motions when the tool is not cutting. Each tab has a green, yellow and red light. Greens means that the mandatory parameters are defined and can be computed, yellow means that the parameters are defined but might need modification, red means that the mandatory definitions are not completed and more information is needed before the operation can be computed.



Figure 4.26 Parameters tabs for operations

Geometry tab

In the Geometry tab the different geometry selection are user friendly and very illustrative, they look like in figure 4.27. The geometries which must be selected are red and optional geometries are orange. Also there are options like offsets on check, part and contour. You can also select start and end points for the tool. All these works similar, you start by selecting the red mandatory option and then make a selection on the part. You can also choose the whole part or if you want to make a optional choice with just selecting a certain part or surface you right click on the option and select for example Select Face or Select Bodies.

For example if you would like to choose a surface on a part. Simply click on the red part on the picture which stands for your desired geometry, in this case a part. When clicking the dialog window hides and you can now select the desired surface on the part. The red part on the picture will turn green as in the picture below.

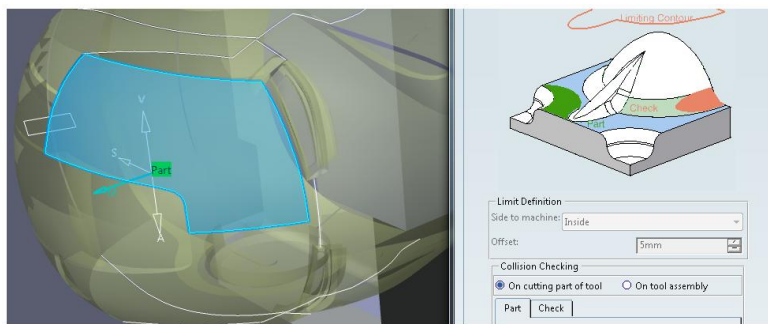


Figure 4.27 Selecting part in geometry tab

The geometry tabs in figure 4.28 are all used in the appendix. Number 1 is *Multi-Axis Sweeping*, here there are only three geometry options: Part, Limiting Contour, Check and two offset parameters. Number 2 is *Multi-Axis Flank Contouring*, here the geometry options are Bottom, Top, Drive Element, Start and Stop. Here you can also select offset parameters along tool axis and drive element, you can also choose if the tool will start or stop inside, outside or on the selected start and stop selection. Number 3 is *Multi-Axis Curve* which is similar as Number 2 except instead of a Drive Element you select a Guiding Curve. Number 4 is *Roughing* here there are many more selections but only Part is mandatory optional selections are Rough stock, Safety plane, Imposed plane and Zone order. Number 5, *Profile Contouring* has the same selection as 2 and 3 except it is a 3-axis operation. Number 6 is *Drilling*, selections are Top, Bottom and Circle pattern, selecting the bottom surface and the top the circle pattern is recognized automatically. Number 7 is *Pocketing* and the selection which is new here is the Island in the middle which can be removed and a choice if the pocket should be open or closed.

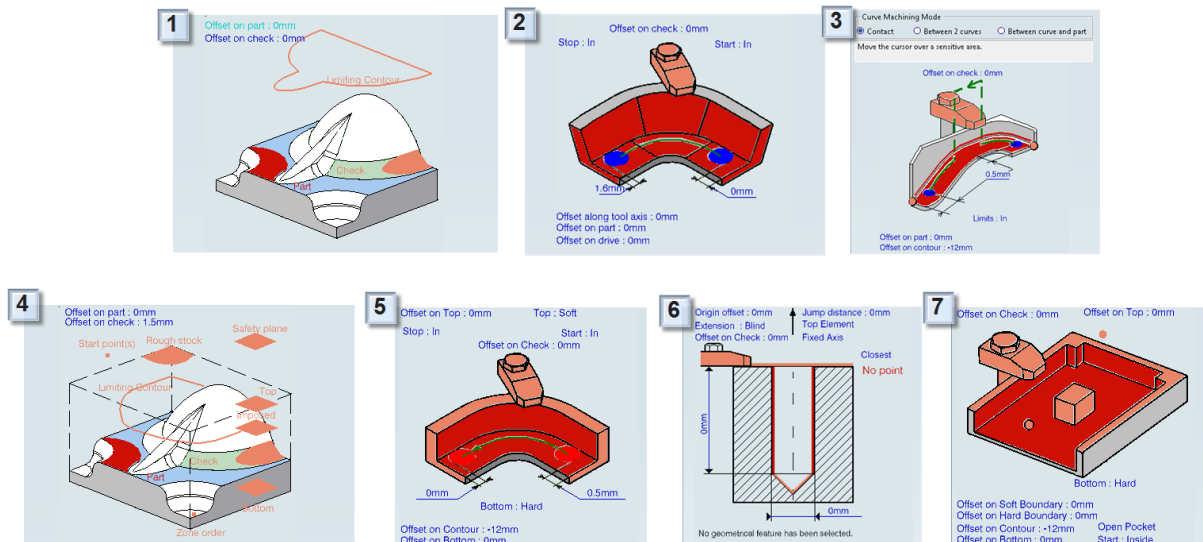


Figure 4.28 Geometry tabs: 1) Sweeping, 2) Multi-Axis Flank Contouring, 3) Multi-axis Curve, 4) Roughing, 5) Profile Contouring, 6) Drilling, 7) Pocketing.

Strategy parameters tab

The strategy tab looks little different depending on which operation you wish to create, this can be seen more precisely in the appendix, but the fundamentals are pretty much the same. One great help in this tab is that there are question marks behind each selection, when clicking on the question mark a small window pops up with a short explanation on the selected strategy, see figure 4.29.

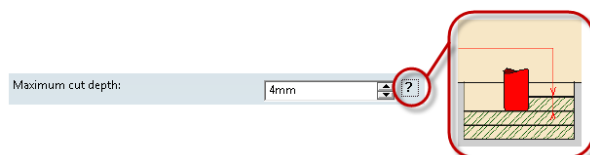


Figure 4.29 Maximum cut depth

In the strategy parameters tab there are a couple of sub tabs, depending on which operation is selected the tabs might be different and depending on which parameters you select in one tab different selections are available in the other. In figure 4.30 the operation is Roughing, in the Machining tab you select strategies as tool path style, machining mode, cutting mode etc. In the picture above the tabs you can see the selected tool path in an illustrative way. In the Radial tab you can select strategies as which kind of radial stepover you desire and what parameters. The axial tab has the option to select what cutting depth you wish to have, the High speed milling (HSM) tab you select corner radius etc.

The big difference between 3-axis and 5-axis operations in the strategy tab is that the 5-axis operations have a tool axis tab. In the tool axis tab you select what kind leaning on the axis you wish to have, as for example a lead and tilt which means that the tool will lean in the cutting direction and lean normal to it as well. This option is very useful to be able to define and optimize cutting angle which gives both better surface and longer lifetime on the tool.

This tab can be very complex and you can select different angles in the same operation if you have a side or a pocket with a curved bottom and different angles on the walls.

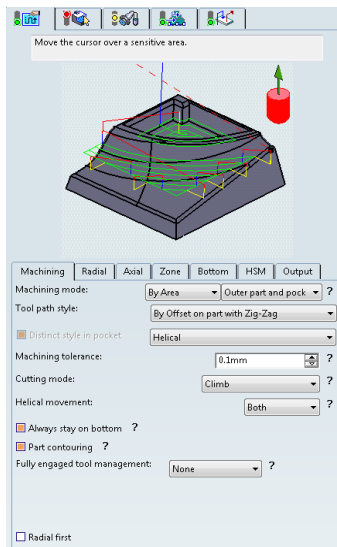


Figure 4.30 Strategy sub tabs

Tool parameters tab

In the tool parameters tab you select either tool or the tool assembly, there are three ways to import tools, see example in figure 4.31. If you already have imported tools to the machining cell you can simply select between those tools. You can also choose to select tools from an existing tool catalog, or import them from the database. When you select a tool the drawing of the tool appears as you see in the picture below, you can also select a wider view to see the tool parameters as predefined tool technology, feeds and speed and compensation.

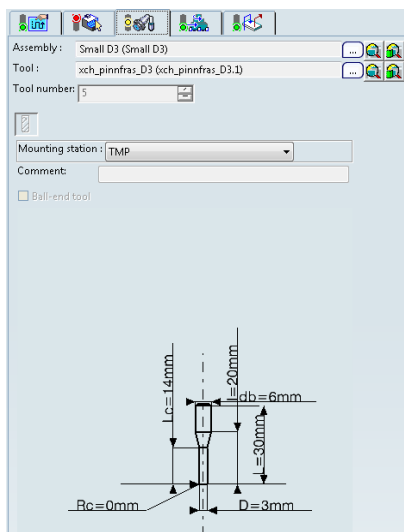


Figure 4.31 Tool parameters

Feedrate and spindlespeed parameters tab

Selecting feedrate and spindlespeed depends on which kind of tool cutter is selected, which type of material and which type of operation. You can select to choose the predefined feeds and speeds from the tool or define specific parameters. It is possible to have a percentage reduction in corners and different parameters can be selected for roughing and finishing.

Macro parameter tab

Macro parameters are the machine motions when the cutting tool is not in the workpiece. These are vital to optimize the machining time, try to avoid unnecessary motions when selecting the linking between one tool path and another. The approach motions are important to be able to have a fine surface on the part, to avoid a undesirable result try to use approach motions that follows continuing tool path. Selecting the desired motions can also be used to avoid collisions in a transition between one operation and another. Because of this it is important to have knowledge about how the CNC machine works and what motions are used when changing a tool or when starting a new operation. You can also define different speeds in the motion to optimize the operation. You can see an illustrative explanation in figure 4.32 on how motions can look like.

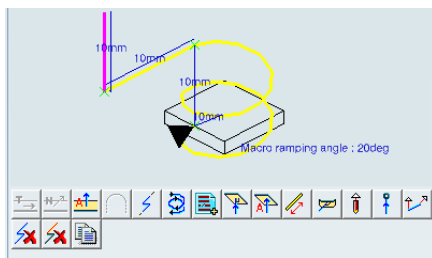
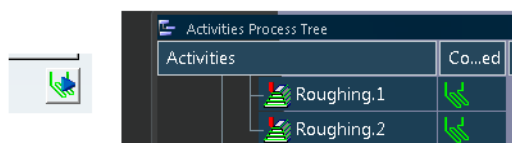


Figure 4.32 Example of macro parameters

Computing an operation

Each operation must be computed to verify and control that the tool path is as desired and no mandatory options missing. When an operation is successfully computed and a green pattern is displayed behind the operation in the Activities Process Tree illustrated to the right in figure 4.33. The easiest way to compute the operation is simply to press the compute button in the operation window, to the left in the picture. Replaying the tool path is not possible if any mandatory option is missing or not correct. If any errors occur like collisions a dialog box appears and informs on this, clashes are then displayed in red and out of boundary kinematics movements are displayed in blue, these colors you can edit and select yourself.



Picture 4.33 Computation in the Activities Process Tree

Computing the tool path is a way to see every tool motion and movement. In figure 4.34 a tool path computation is illustrated, you can choose between different viewing options. You can select to view point by point replay with or without axis movement etc. In the replay positions you can see how long the tool path is and which the current position is, below feedrate and axis motion moments are displayed and total time.

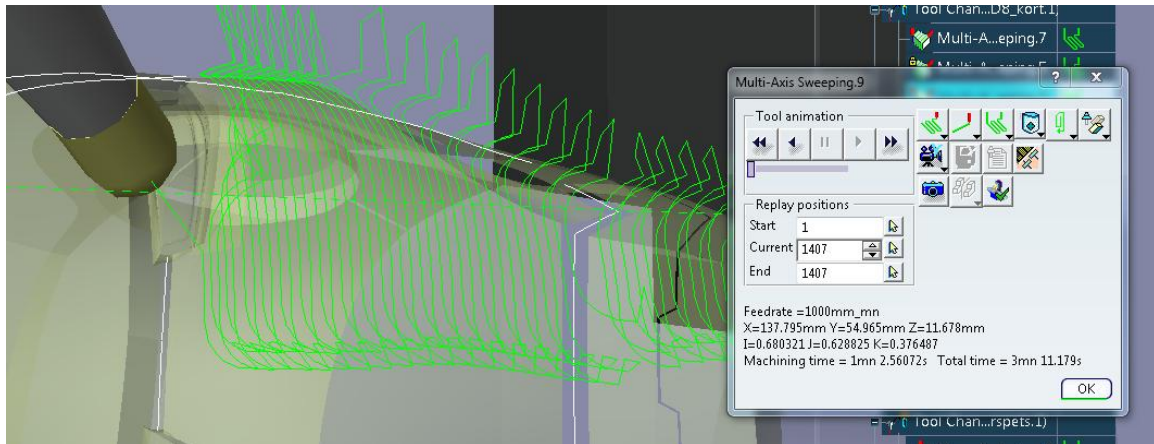


Figure 4.34 Computing the tool path

Computing the tool path with material removal is a way to see how the finished result will look like depending on surface etc. There are options to select if you would like to analyze the result and how much material is left until the original design is achieved. Different colors are displayed depending on which tool is used. In figure 4.35 is an example of how the result could look like.

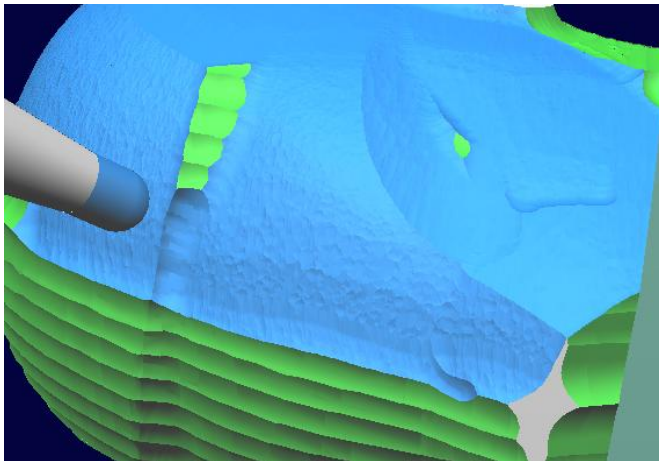


Figure 4.35 Material removal

4.4.1 Prismatic Machining Operations

Prismatic machining operation is a category with 3-axis operations where all selections are mostly manual comparing to 3-axis roughing for example. When selection an operation and after that the part the machining feature of the product design is recognized automatically which gives a reduced tool path programming time. For the Iron Man part only one operation where used from this category, for the appendix a second, a third and a fourth operation was selected as well.

Pocketing operation

This is a operation to create open or closed pockets, with or without island and with a hard or a soft bottom. To optimize the geometry definition there is an option the select automatic island and contour detection. By having the option marked when you select a bottom those parts will be detected automatically. In the strategy tab there are different tool path styles to select between and the high speed milling (HSM) selection is also available which you can see to the right in figure 4.36. With the HSM option available there will be a corner radius, as illustrated in the small box in the picture to the right, this will allow the tool to move much faster without having tight turns which gives a longer lifetime to the cutter. More specific about pocketing operations is explained and illustrated with an example in the appendix.

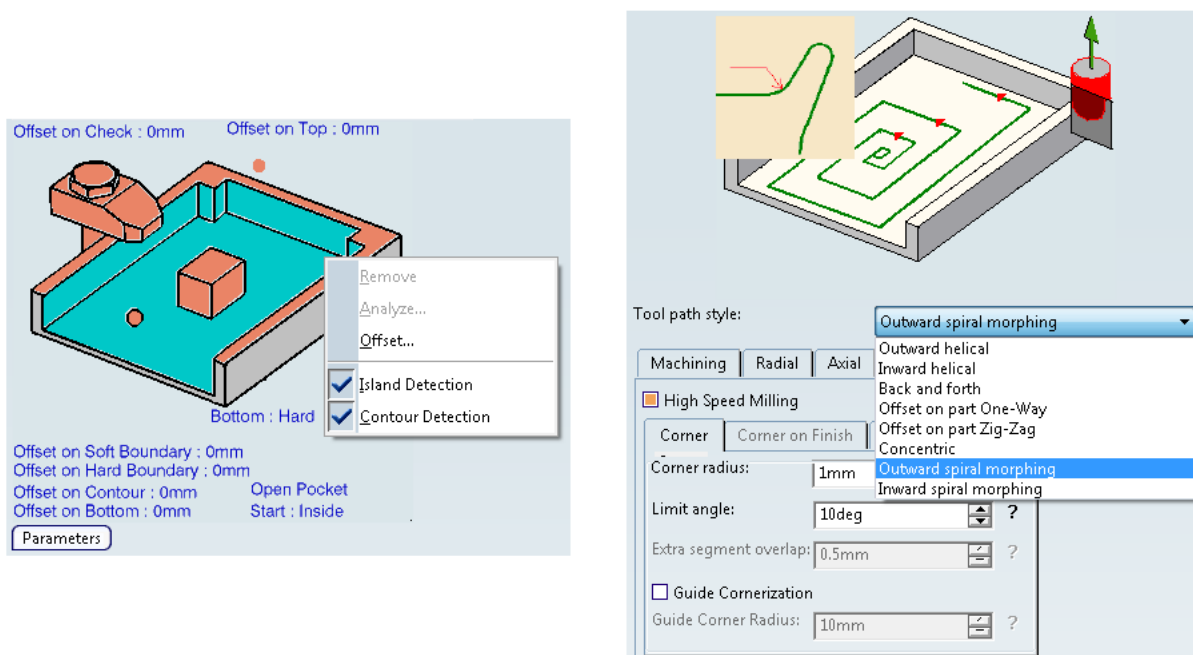


Figure 4.36 Geometry tab and Strategy tab in Pocketing operation

Profile Contouring

The operation works sort of that the tool will follow a hard profile and remove the material in axial direction. There are four different modes to select between in the in geometry tab, as can be viewed in figure 4.37; *Between Two Planes*, *Between Two Curves*, *Between Curve and Surfaces* and *By Flank Contouring*. In the strategy tab the tool path styles are *Helix*, *Zig zag* and *One way*, illustrated to the right in the picture is *Helix* tool path style. Other selections are the sequencing, you can choose between axial first or radial first.

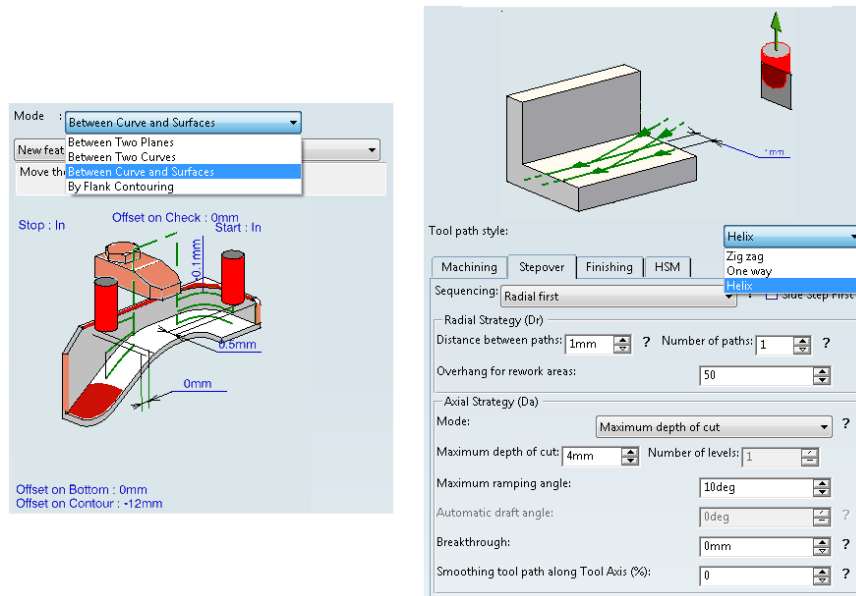


Figure 4.37 Profile contouring, geometry tab and strategy tab

Point to Point Operation

The operation allows you to move the tool from one point to another. In this project the *Go to* strategy was selected, that means that the tool goes from the earlier operation to the selected point. The operation was used in the exercise in the appendix because of limiting space when going from one prismatic operation on one side to another on the other side. By moving the tool to the selected point the A axis had space enough to rotate and aloud the next operation to proceed without any clashes.

Curve Following operation

This operation is used by selecting a curve or a line and the tool will follow it, there is also an ability to choose an offset to the line. This operation can be used to create a small pattern or a text on a surface. In the Iron Man helmet it was used to follow the contouring lines in the side of the face as you can see by the yellow lines in figure 4.38.

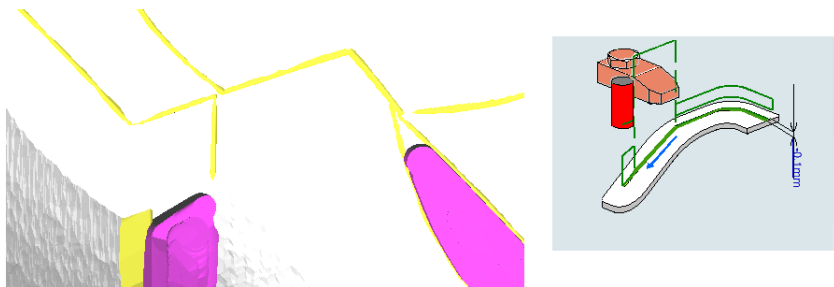


Figure 4.38 Curve Following operation on the Iron Man helmet

4.4.2 Axial Machining operation

There are different operations available in Axial Machining operation; Drilling, Drilling deep hole, Thread Milling, Reaming, T-slotting etc. When select a drilling operation, which is more specific in the appendix, in the geometry tab you can select top, bottom surface or positioning. When selecting a hole, in e predefined hole pattern, it is available to choose the whole pattern and then to select which hole to start with. In figure 4.39 is the Axial machining operations selection menu.



Figure 4.39 Axial Machining operations

4.4.3 Surface Machining operation

In the surface machining menu are the most complex operations, selecting between 3-axis and 5-axis operations, roughing and finishing. There are many different options available for each operations and even more parameters. The 3-axis operations are mostly defined automatic, you select boundaries and the product to machine and the tool path is selected automatically whit macros etc. This does the selection a lot faster and optimizes the process. The big different between 3- and 5-axis operations in surface machining are that the 5-axis operations are defined more specifically where it is important to find the right tool axis angle and cutting directions.

Roughing operation

The definition of the operation is very smooth and easy defined. As you can see in figure 4.40, you select rough stock, check elements and the whole part. Then you define the top, where to start, and the bottom, where to stop, and the angle of the axis. The tool path is selected automatically, the same was done in the appendix as well.

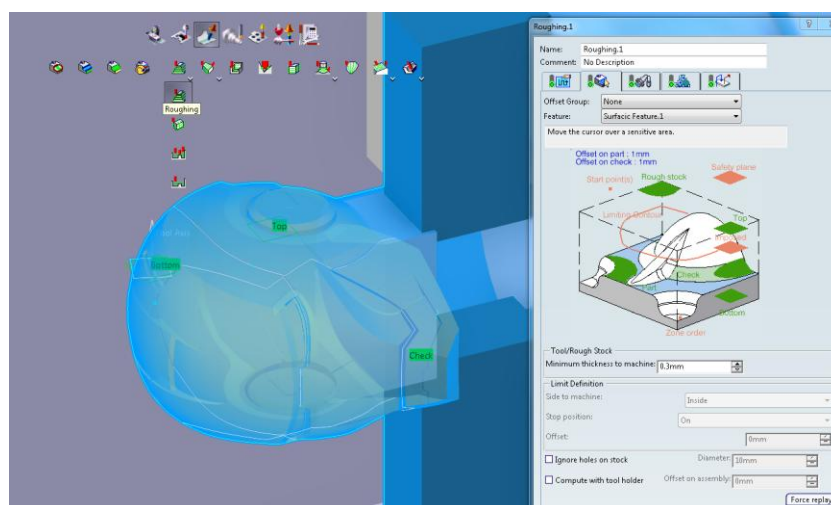


Figure 4.40 Geometry tab in Roughing operation

The parameters in the strategy tab and the macros are as seen in figure 4.41. The tool path style is selected as By Offset on part with Zig-Zag, it means that the tool will start the cutting from the outside in a motion that follows the geometry. The radial, axial and HSM parameters are defined as well and there is no big difference there between earlier operations. The difference is in the macros, as you can see on the top right corner of the picture the macro tab look different. The macros are selected automatic and editable parameters are listed below, you can choose between different modes as Ramping, helix, plunge and radial only. In macros there are also an option to select *Optimized retract*, with this option the tool retract motion is optimized taking in mind that material have been removed so the retraction does not have to go up to a safety plane and by this saves a lot of time. The option also takes in consider the rough stock, but it is important that the material have been removed so no collisions occur. One problem with the Iron Man helmet was that the bottom was defined as a plane in the middle of the face. To avoid a big scallop when roughing on one side first and then the other side, and negative offset was put on the bottom, the problem now was that the tool could not reach that for down and collisions were found, this because of the parts radius and the tool cutting length.

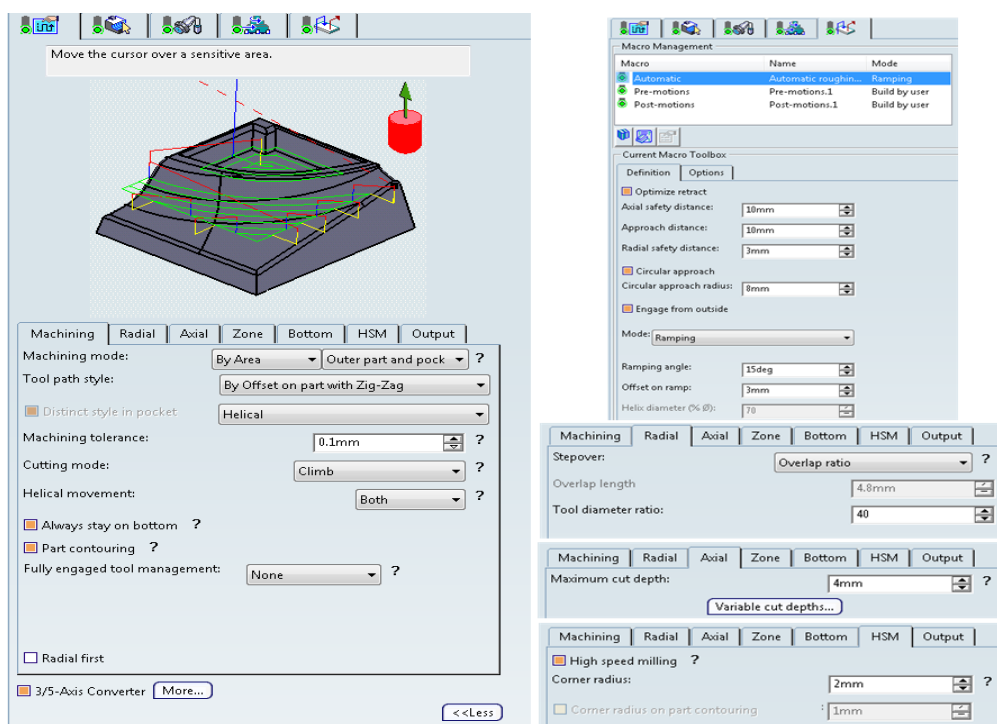


Figure 4.41 Strategy tab and Macro parameters

To avoid these collisions and be able to do the roughing in only two steps one more option is available and very special for this operation. It is the 3/5-Axis Converter as you can see marked in the bottom left corner in the strategy tab above. This option allows you to give a 3-axis operation 5-axis motions to avoid the collision. In the 3/5-Axis converter you can select collision checking which is explained in figure 4.42. This selection forces the tool to move in an angle when a collision is detected. Selecting the Machine kinematics tab allows only movement in the machine kinematics and tells you if a motion is possible or not. It also

removes points created outside the axis limits. These two options are very effective and give you a great opportunity to optimize the operation to have 5-axis motions, which are slower, only where needed. This was not possible to create in the Iron Man operations because of the complexity and the Global modification was needed. The global modification puts an angle on the tool axis on the whole operation. In this case the angle was put at 8 deg. The next option is specific macros to the 5-axis motions, the available macros are approach, retract and clearance. When these were selected the operation could be computed and successfully simulated.

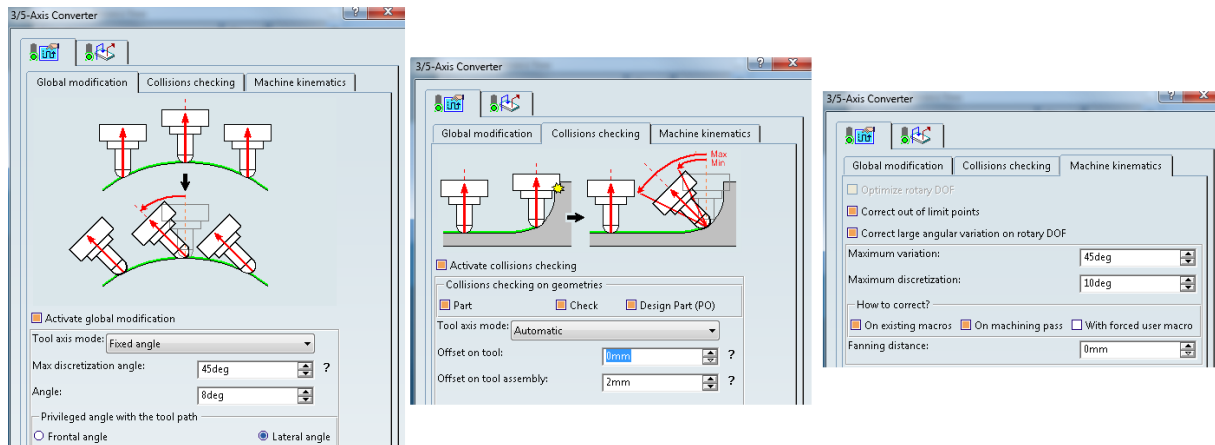


Figure 4.42 3/5-Axis Converter option

The result of the roughing is illustrated in figure 4.43, the next step was to copy this roughing operation and reverse the tool axis and the complete roughing was done.

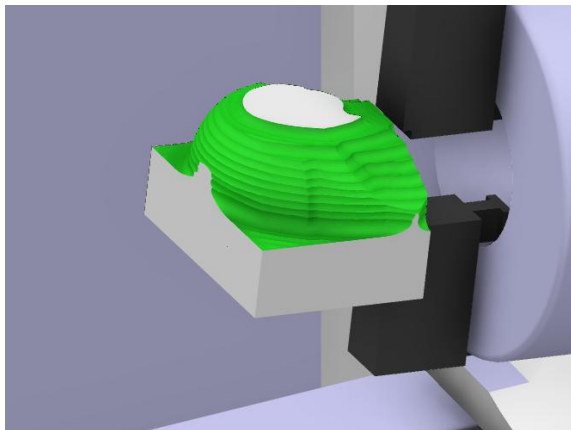


Figure 4.43 Simulation on the roughing

Multi-Axis Curve Machining operation

There are three different types of selections in the geometry tab; *contact*, *between 2 curves* and *between curve and part*. In all three modes you select two different geometries, either two curves or one curve and one part. One you choose as guiding element and the other as auxiliary guiding element. In the strategy parameters and macros are the same options as

profile contouring except from tool axis choices where you can select different angles on the axis depending on your part and tool. This operation is more specified in the appendix.

Multi-Axis Sweeping operation

This is a finishing operation and because of this there are not as many options in the geometry tab. There should be only a thin surface left to create the finishing operation and get the desired surface. There are two ways of selecting part; either by selecting the body and defining limiting contour or by right clicking and selection *Select faces...* the selection is very effective and if there are no sensitive parts no other selection is needed as you can see in figure 4.44.

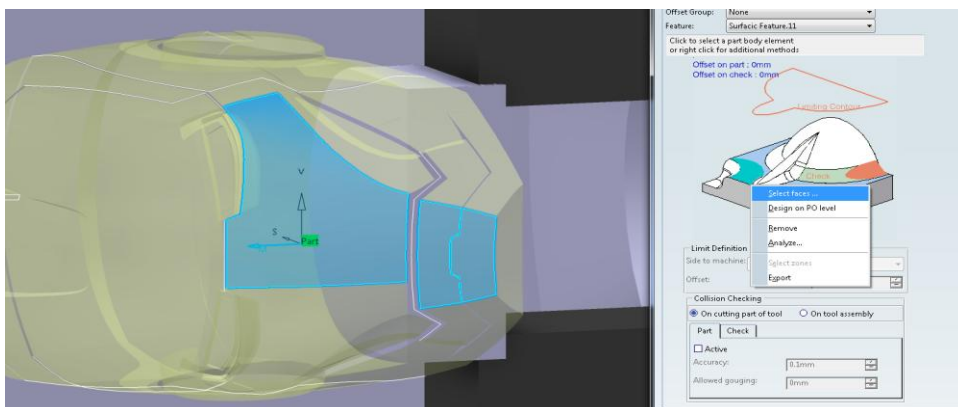


Figure 4.44 Selecting face in Multi-Axis sweeping operation

In the strategy tab there are two important parameters, selecting *start direction* which is the S axis on the part in the picture above. The start direction is the tool path direction, depending on the part and check sensitive elements near this one is very important, it is also important to get the finished surface as desired. The tool axis definition is important for the surface finish as well, the angles of the tool should be optimized for the part to be machined. Because of the importance on the surface finish in a finishing operation as multi-axis sweeping there are a lot different Tool axis modes available. The different modes are illustrated in figure 4.45. The tool axis mode used in the Iron Man helmet and Appendix is Lead and tilt, the angles are explained more specific in the appendix. The tool path will be in a sweeping motion and have a lot of linking and different levels and because of the importance in the surface finish this also leads to the importance in the macros, the linking between tool paths are important to be smooth and at the same time effective because the 5-axis operation takes longer time.

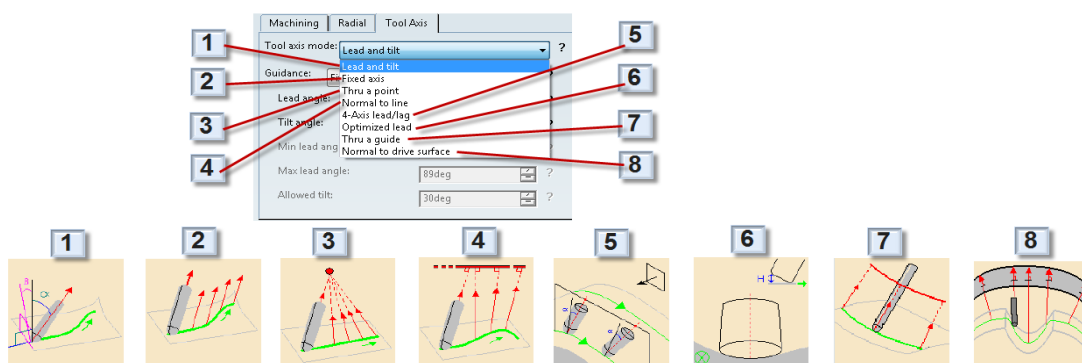
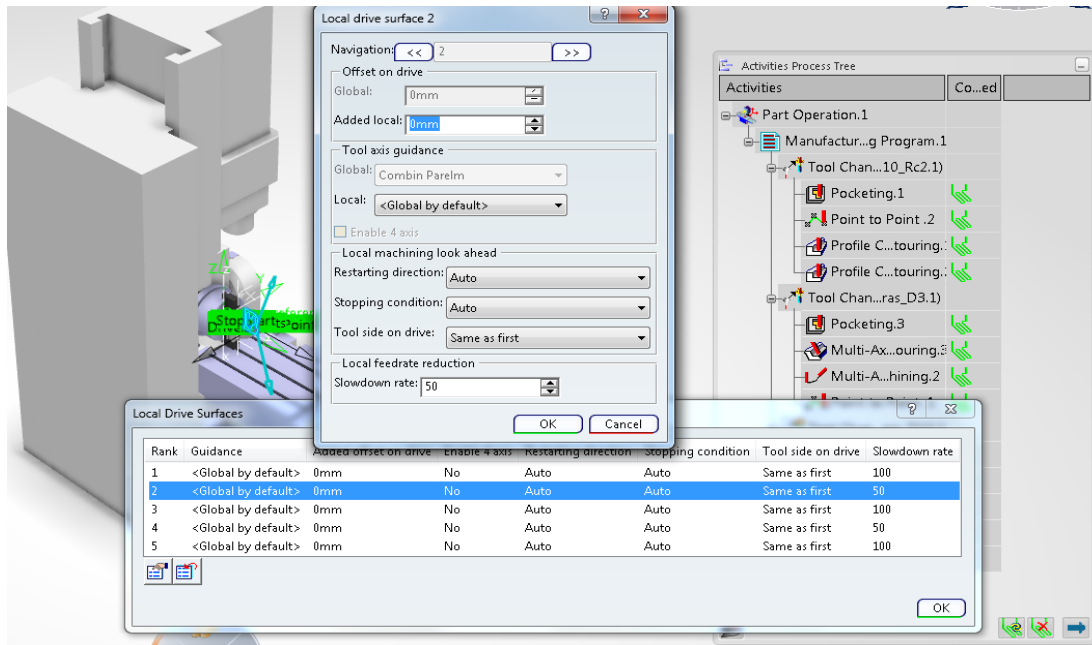


Figure 4.45 Illustration of different tool axis modes

Multi-axis Flank Contouring operation

This is one of the most advanced operation and is used on flank sides, either pockets or profiles. The operation is mostly used in the aerospace industry where profiles need different strategies in the same operation to avoid collision with the drive surface. Selecting drive elements, parts and start and stop is the same as in the earlier operations. When the selections are done there is an option to edit local modifications. Right click on the drive element selecting and select *Local Modifications*. A list with the different drive elements appears and by double clicking on one an option box appears. Here you can edit navigation on one drive, specific tool axis and slowdown rate.



Picture 4.46 Edit local parameters

The big difference is in the strategy tab, tool axis, where there are many parameters and all are depending on which part you want to machine. The fanning distance to avoid collision with a drive surface is illustrated in the small box in the top left corner in figure 4.47.

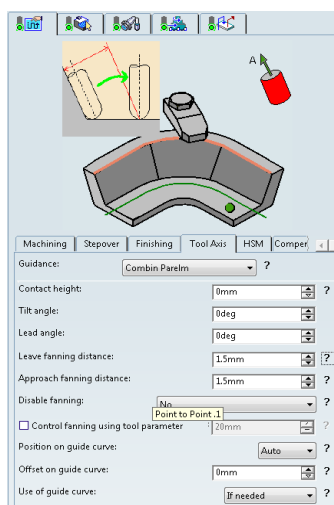


Figure 4.47 Strategy tab, tool axis with fanning explanation

Different Tool guidance are:

Tanto fan: This is the basic strategy and gives a good continuity through the tool motion. For structural parts it is less used because you cannot control the collision with the driving surface illustrated in figure 4.48.

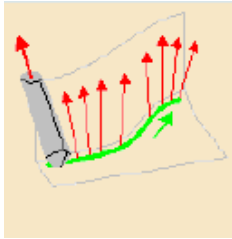


Figure 4.48 Tanto fan guidance

Combin Tanto: The strategy ensures that the tool stays normal to the part in the forward direction with a fanning at the beginning and at the end of the tool motion. It is suitable for circular pattern and planar drive surfaces, illustrated in figure 4.49.

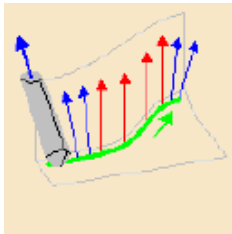


Figure 4.49 Combin tanto

Combin Parelm: This strategy makes the tool follow the isoparametrics of the surface with fanning at the beginning and end. The strategy is suitable when the drive surface have a good orientation, illustrated in figure 4.50.

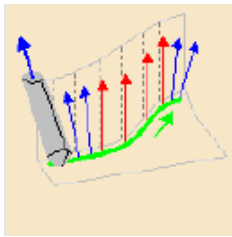


Figure 4.50 Combin Parelm

Mixed Combin: This strategy provides a *Combin Tanto* on planar and cylindrical surface and a *Combin Parelm* in other cases. It is the best suitable strategy for structural part flank contouring illustrated in figure 4.51.

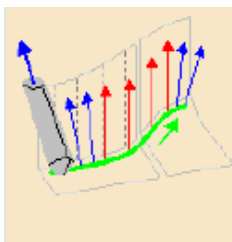


Figure 4.51 Mixed Combin

4.4.4 Choosing operations

Identifying which operations are suited for a certain part depends on three things; what kind of part is to be machined, what kind of machine is available and what kinds of tools are available? The Iron Man helmet is a part with many shapes on the surfaces and a lot of tight spaces. The machine has relatively small motion movements in Z-axis and in B-axis and the smallest tool available was a D3 End Mill with an axial limit.

With the limits on the machine motions and the shapes on the surfaces of the Iron Man part the possibility to create a 3-axis operation was not an option. But thanks to the 3/5-axis converter in Roughing operation, the roughing could be done relatively fast with mostly 3-axis motions and 5-axis were needed. On the rest of the big surfaces Multi-Axis Sweeping could be used which allowed change on the tool axis angles to fit the machine motions.

On the back side of the part and on a place on the top the easiest way was by Multi-Axis Contour Driven where you could control the limits on the contour better. This was needed because of the height difference between the surfaces and the edge limits where macro motions were strictly limited.

In the eyes and the ears the walls are leaning and the bottom is a shaped curve at the same time as the space is very tight. To be able to create a fine surface finish on both bottom and wall there were two different types of operations needed. One was Multi-Axis Curve Machining and the other Multi-Axis Flank Contouring, with these operations the machining on these complex parts of the part was possible. In the end the last operations needed were Curve Following which could bring out all the contours.

4.5 Verifying

It is important to be able to verify the machining operation to be able to eliminate errors as collisions and motions outside the kinematics. When the machining program have been verified you need to generate an NC code to be able to produce the product. In this chapter is described a couple of ways to verify the operations and then to generate an NC code.

4.5.1 Simulation

There are a couple of different ways to verify an operation, the easiest way is by computing the tool path as described in the operation chapter. This simulation only regards the tool motions and you are not able to see the machine movements. There are two different ways to simulate the operations regarding the machine kinematics, one way is directly in the machine programming workbench and the other is by selecting the machine simulation workbench.

Simulation in the Machine programming workbench.

There are a couple of ways to do the simulation, either by tool path or by an existing NC code. You access the simulation either by right clicking on the Part Operation in the Activities Process Tree or by selecting it on the west side of the compass in figure 4.52.

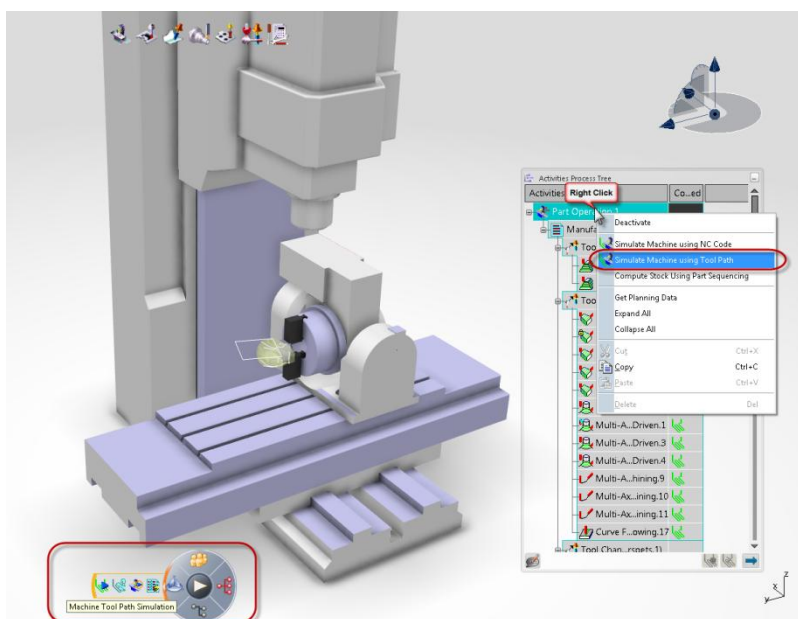


Figure 4.52 Accessing the Machine Tool Path simulation

When the simulation is started there are a couple of selections available, to view the machine kinematics select Data Readout in figure 4.53. On the simulation player you can view cycle time and stop and play the simulation. In this mode clashes on the part is displayed in the color red and the same with failure in the kinematics motions. Though they will not be displayed automatically or logged.

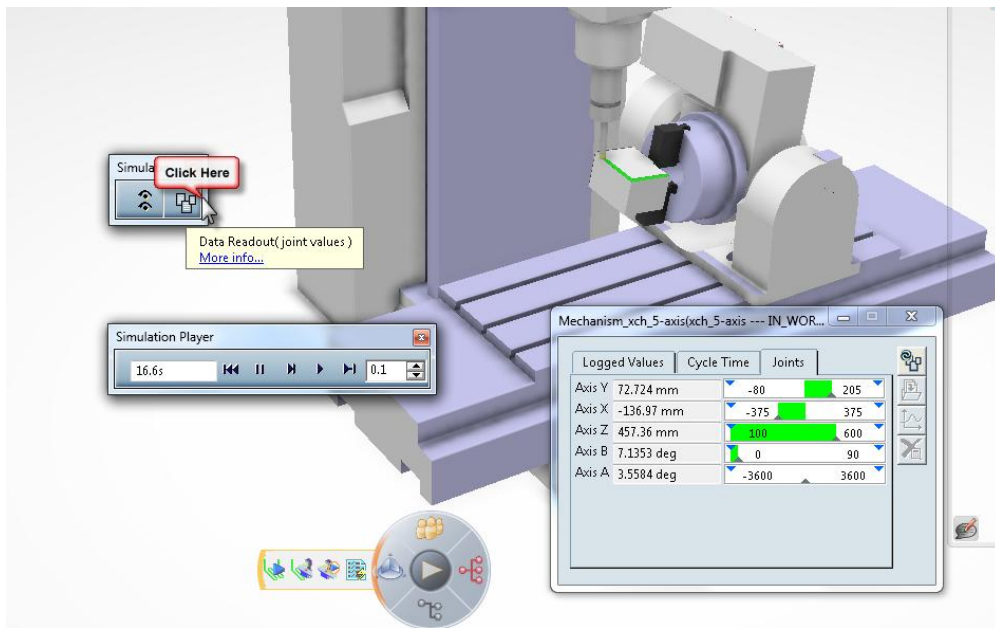


Figure 4.53 Machine tool path simulation

Simulation in Machine Simulation workbench

Accessing the workbench is done by Selecting Machining Simulation on the west side of the compass in figure 4.54. When selecting it a dialog box pops up where you name the machining simulation, when that is done the machining simulation workbench is open.

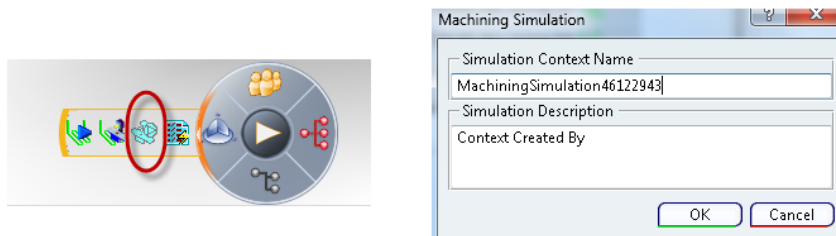


Figure 4.54 Accessing the Machining Simulation workbench

The simulation look very similar but the difference is that you can access more tools to analyze the operations. In this simulation collisions and motions outside the machine kinematics will be notified and warnings appear as you see in figure 4.55, you can also view which operation is working right now. You can select to save the simulation to analyze it later or to create a demo.

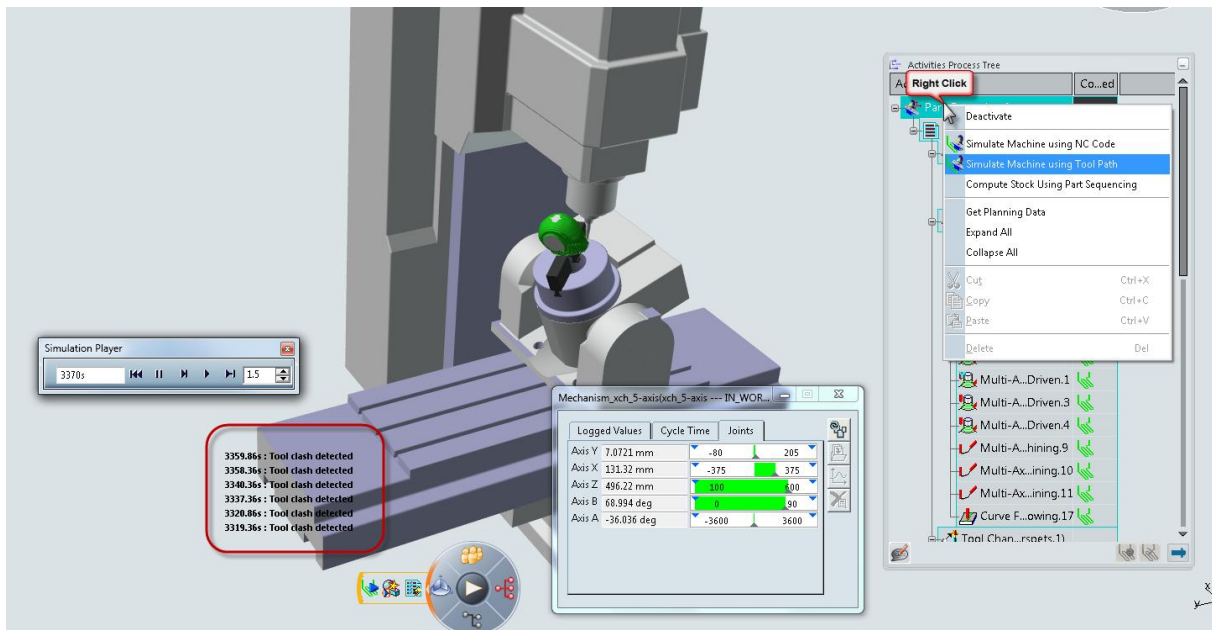


Figure 4.55 Simulation in Machining Simulation

4.5.2 Generate NC code

To create a specific NC code for the machine you need a post-processor. If you do not have a post-processor you can still create a general NC code in APT, CL or ISO format. The post-processor is defined in NC-resource which is described in the Setup chapter, and the specific selections to create and view the NC code are described in the appendix. In the figure 4.56 is an example of an NC code and the generate NC output dialog window where you select which format you want the code.

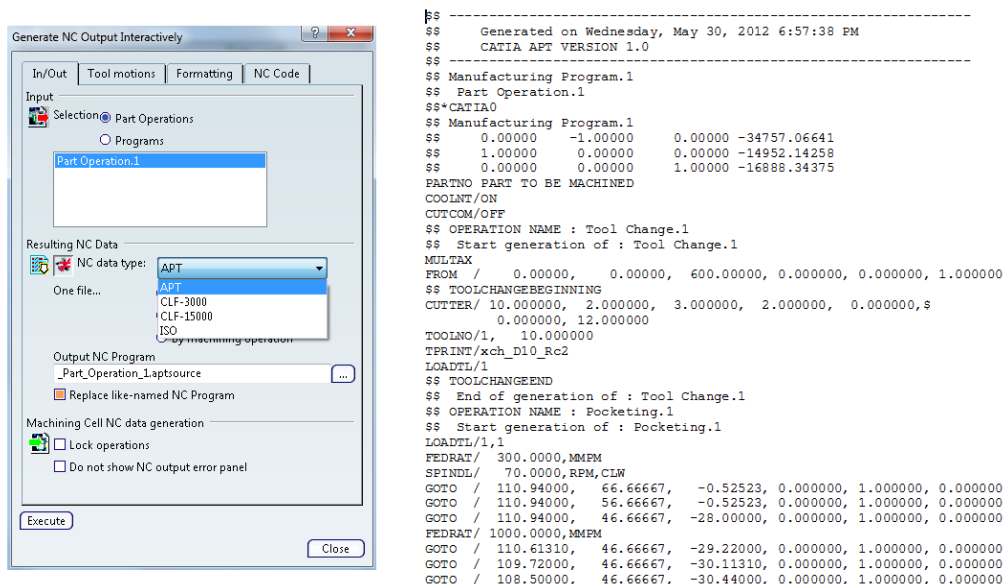


Figure 4.56 Create NC code in APT format

5 CREATING EXERCISE

This chapter describes the work to identify which operations and steps in DELMIA V6 that was identified as the most vital areas and how the step by step exercise in the appendix were built after this steps.

5.1 Identify vital areas

To write a step by step manual how to perform a CAM preparation a division of which functions to include and which not had to be done. This was done by starting at square one to make sure that the manual is adapted to the audience. The machine was built and a full CAM preparation of the Iron Man helmet was done. Five steps for a full CAM preparation were identified; *Building NC machine*, *Assembly*, *Setup*, *Operation* and *Generate NC-code*.

Building NC machine is not typical what a NC programmer works with and it is not necessary to have a NC machine to build up a CAM preparation, which is why the appendix does not contain this part.

When it comes to the *Assembly* three things were defined as important to transact a full CAM preparation. The part that will be manufactured always has to be mounted in the machine somehow, this is why assemble the part with the fixture is an important step for the manual. To be able to mount the part in the machine an axis system for mount point and a reference axis system have to be defined.

In the *Setup* phase four main steps to make a full setup were identified. The first step is how to connect the assembly with the machine. Second step how to define the machine axis system. Third and fourth step are how to create tool assemblies and how to check the reachability for the tools.

Operation

a. 3-axis

i. Prismatic machining

1. Pocketing
2. Profile contouring
3. Drilling
4. Point-to-point

ii. Surface machining

1. Roughing

b. 5-axis

i. Surface machining

1. Multi-axis curve machining
2. Multi-axis flank contouring
3. Multi-axis sweeping

Figure 5.1 Selected operations

There is a lot of different milling *Operations* in DELMIA V6, to make a more pedagogical manual a selection of which operations were most useful was done. The selected operations are shown in the figure 5.1.

Verify and generate NC-code is the last phase to build up a full CAM preparation, two important steps were identified. The first step is verifying tool path and make a full collision checking. The final step to complete the preparation is to generate the NC-code.

5.2 Building exercise

To assemble the part with the fixture, insert axis systems, assigning an NC machine and create tools are done quite similar no matter what part you have. Also verifying the tool path and generating the NC-code are done in the same way. What is different for different parts are how the operations are built up. To make the exercise easy to follow through one part with all the different operations were created. The idea with this is that the user can follow the whole working process from having a part to generating the NC code. The part that was modulated is shown in figure 5.2.

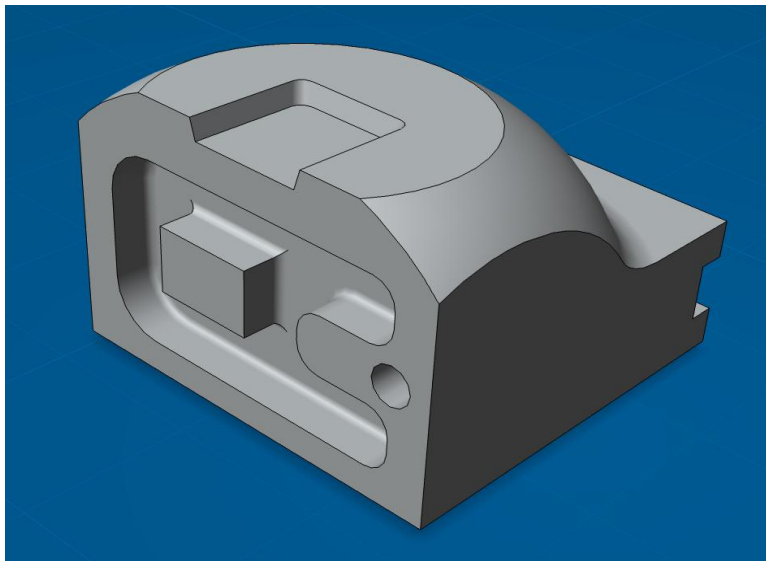


Figure 5.2 The part to create with the step-by-step guide

The pocket on the front was made to use the *pocketing operation* and the hole to the right of the pocket for a *drilling operation*. The profile on the back of the part was made to use a *profile counteracting operation*. In the pocket on the top with the sloping walls *multi-axis flank contouring* and *multi-axis curve machining* is used. The material between the top and back is removed with a *3-axis roughing operation* before it is finished with *multi-axis sweeping*. *Point-to-point operation* is used to avoid collision when switching from one side to another. When the part was done a full CAM preparation on the part was done. Each step was documented in a pedagogical and explanatory way with pictures and relevant explanations. The document was complemented with a general introduction of how DELMIA V6 works. The full working process in DELMIA V6 can be seen in the appendix.

6 CONCLUSION AND RECOMMENDATIONS

As a conclusion we feel that the machining program in DELMIA V6 is very mature and well developed. The machining program have a very wide ability for creating any kind of machining operation, all from easy 2.5-axis operations to the most complex and advanced multi-axis operations. With the illustrative and easy understanding parameter tabs it makes it fast and easy to define operations and optimizing the whole process. The overall feeling is that the program is very user friendly and thanks to the transparent menu it is easy to select all the different operations and utilities. Together with the database the whole working process is made smoother and faster, it is easy to access different products, parts, machines, PPR programs and tools. The total design of the program and working environment with the database, PPR tree and Activities process tree felt very natural to work in and is something all companies should aim for. The implementation of machining in DELMIA V6 gives it a totally new dimension from the CATIA V5, because you can use it to optimize the whole process together with robots and workers in the workshop.

Creating the machine in the Robotics device building was much faster and easier than with the old system with axis systems. Defining machine parameters was also very smooth and connecting the machine to a defined tool catalog spares you a lot of time.

If there would be more time for this project we would have done some more tests with the step-by-step guide to see that we have not missed out on any point. We also wish that we would have more time with the simulation tools to do some analysis to be able to optimize and create a process with high efficiency.

Some recommendations and ideas for future work are to create a post-processor so that the ability to create a part in reality is possible. Another recommendation is also to broaden the view and implement the machining operations in manufacturing planning and by that optimizing the complete production process and create an efficient workshop strategy.

BIBLIOGRAPHY FOR SOURCES

1. <http://en.wikipedia.org/wiki/Cnc> (acc 2012-05-25)
2. http://en.wikipedia.org/wiki/Milling_machine (acc 2012-05-25)
3. http://en.wikipedia.org/wiki/Computer-aided_manufacturing (acc 2012-05-25)
4. http://www.icam.com/html/products/whatis/what_is_post.php (acc 2012-05-25)
5. <http://www.3ds.com/se/products/delmia/portfolio/delmia-v6/v6-portfolio/d/digital-manufacturing-and-production/s/program-control-engineering/p/milling-machining/?cHash=b618a4993ea91a4c17a69fd633ff5f7f> (acc 2012-05-31)
6. <http://www.3ds.com/se/education/ds-university/elearning/why-an-elearning-management-system/> (acc 2012-05-29)
7. http://www.ehow.com/how_2273446_write-manual.html (acc 2012-05-29)
8. <http://www.sitepoint.com/7-tips-for-writing-an-effective-instruction-manual/> (acc 2012-05-28)

BIBLIOGRAPHY FOR PICTURES

1. http://smaskin.se/web/Produkter/CNC-maskiner/CNC-frasmaskin_KX3-Sieg/CNC_FRASMASKIN_KX3_SIEG-6600.aspx (acc 2012-05-25)
2. http://www.5axisncrouter.com/images/Dmi%205mb1015-48%20clean_lrg.jpg (acc 2012-05-25)
3. <http://www.cadnc.se/nc.html> (acc 2012-05-25)

Step-by-step guide to create a machining operation in DELMIA V6

Valjon Hasani

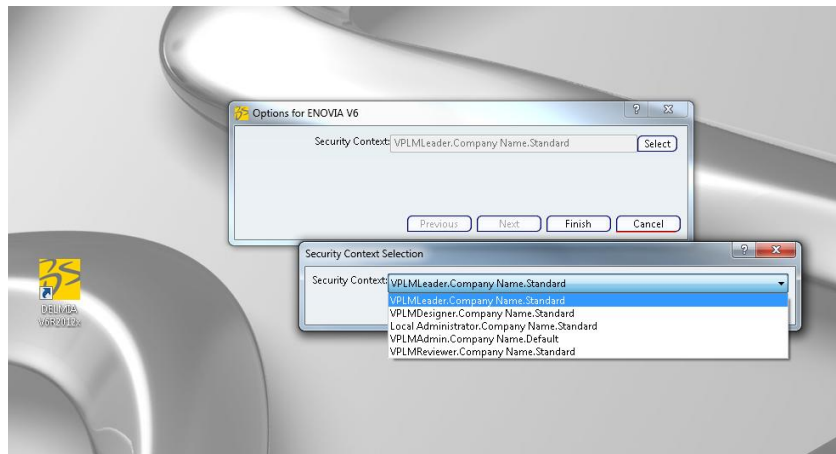
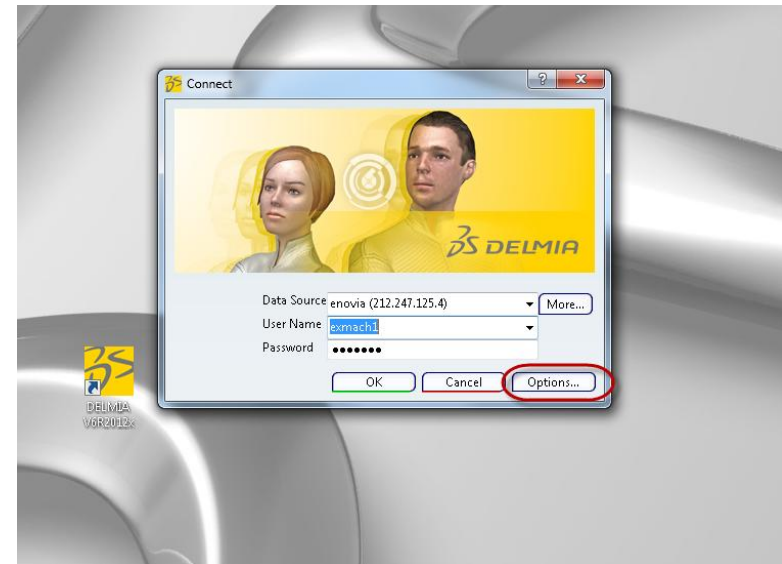
Adam Nilsson



Introduction

To start a new session first select a **data source**, the server you want to connect to. To connect to the ENOVIA server you need to type down your **user name** and **password**..

..then press **options** and choose your **security context**

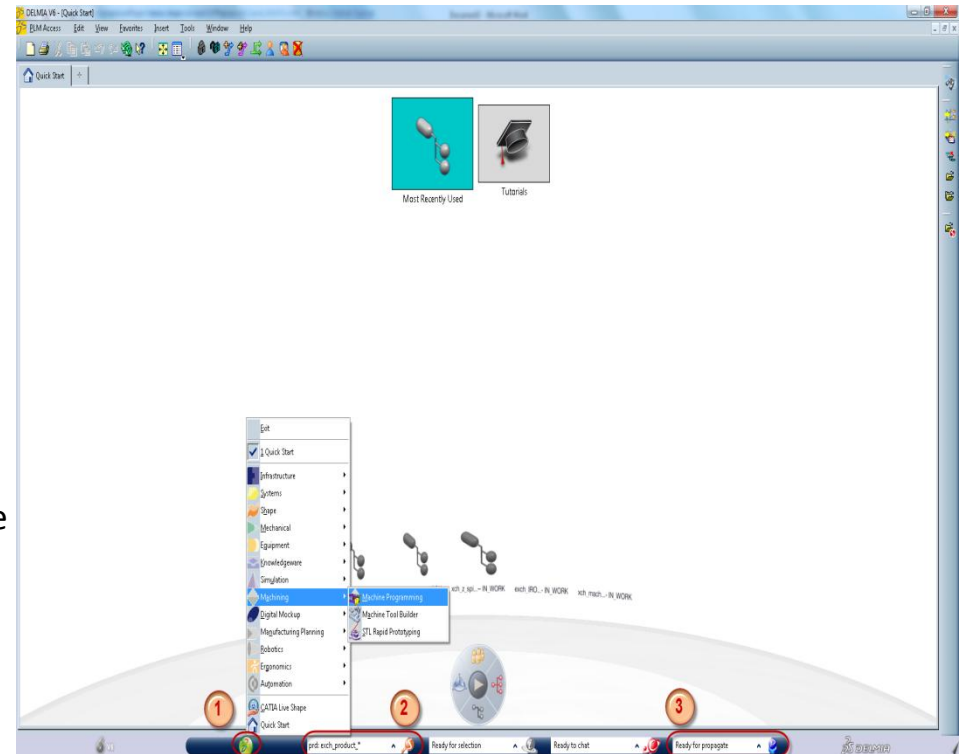


..press **OK** and the DELMIA browser opens

Overview

The first page that opens is the **Quick start**, here you can see your recently used files and and open them directly for a quicker start

1. The **start** button, here you will find all the workbenches
2. The **search** button. To search in the database you need to write down a search string, example product called “exch_product_machining”, the search string should be “prd:exch_product_”
3. The **propagate** button is to save a change to the database

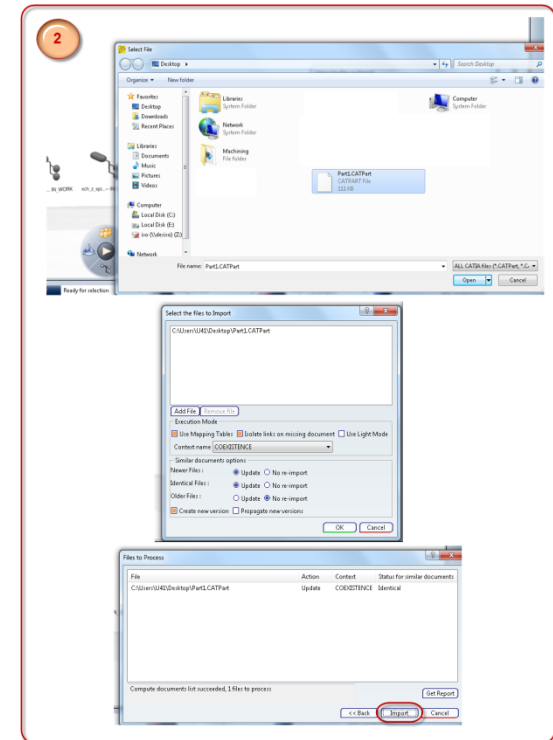
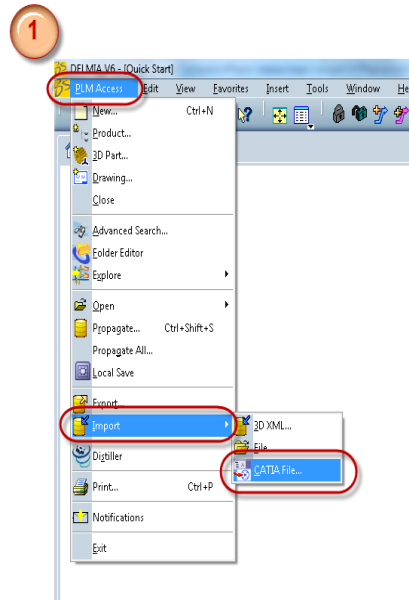


Import a file

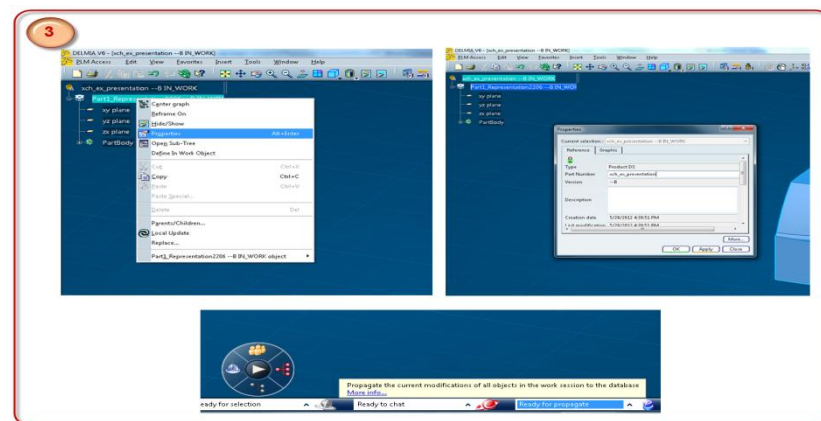
To import a file to the database there are three different ways depending on the format.

Import CATIA file

1. **PLM Access > Import > CATIA file**
2. Select your file and press **Open**, click **OK** and **Import**



3. Select **properties** and change the products name and then **propagate** the product to the database

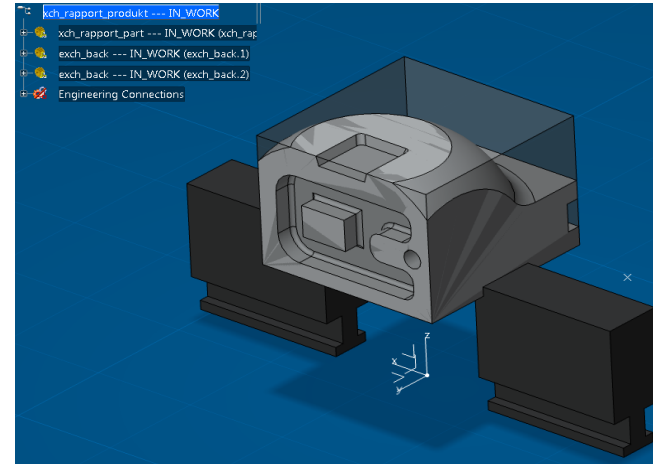


To create a complete machining operation in DELMIA V6 there are four steps you need to follow

1. Assembly

- a. Part
- b. Fixture
- c. Rough stock

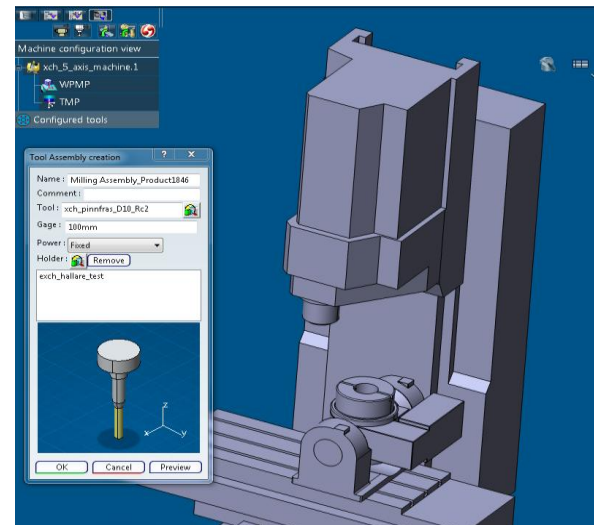
In this step you choose your part and assembly it with your fixture. Insert axis systems for mount point and reference axis system

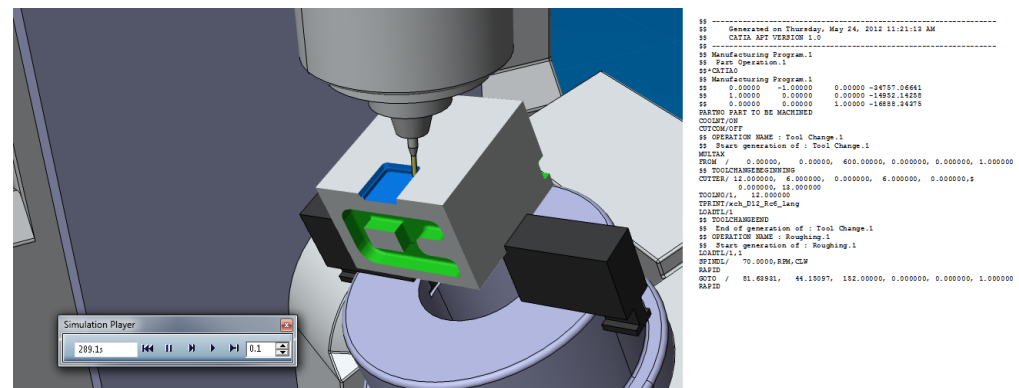


2. Setup

- a. Machine
- b. Tool assembly
 - i. Tool
 - ii. Tool holder
- c. Define axis system

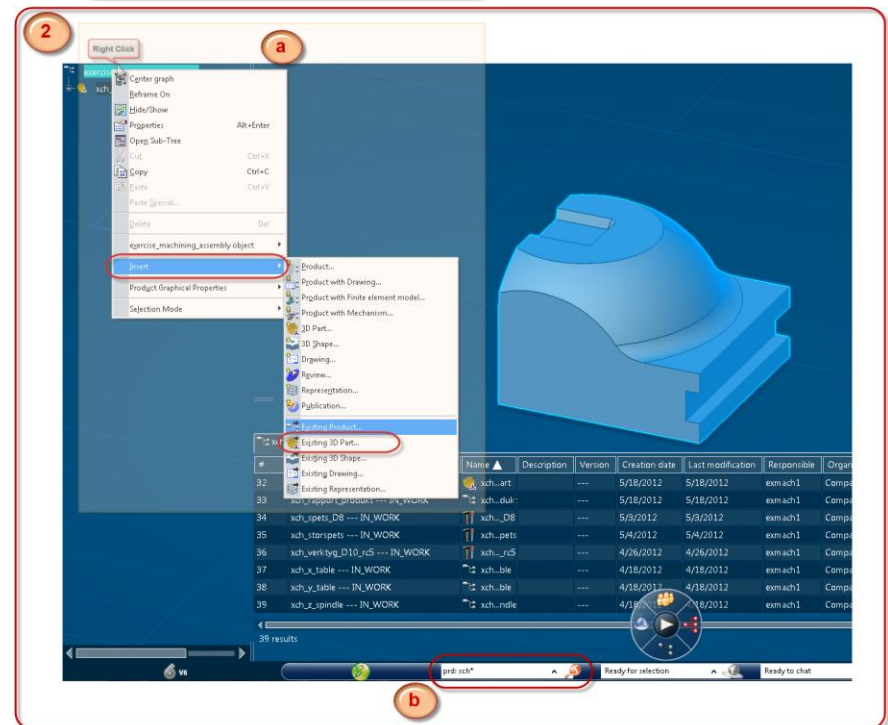
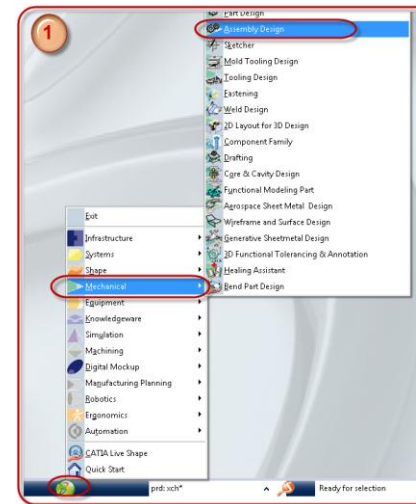
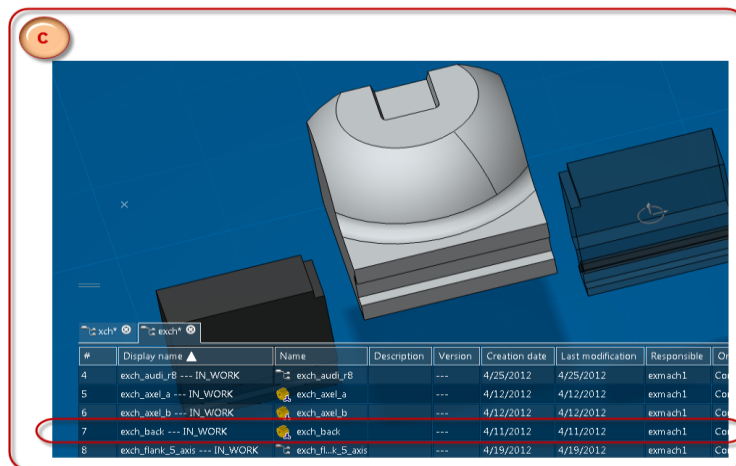
In the setup you connect a machine to the part and the tools to be used. Define reference axis system to use and check the reachability



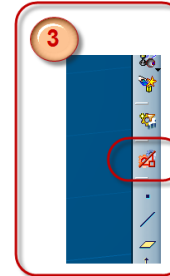


ASSEMBLY

1. Open the **Assembly design** workbench
 - a. Name the product
2. Insert part and fixture
 - a. **Right click on top of the tree > Insert > Existing 3Dpart**
 - b. Type down the **search string**
 - c. Select your **part** and place it on the window
repeat step a for the fixture parts
To make the engineering connection easier place the fixture parts close to the main part



3. Click on the **Engineering connection** button



a. **Select the edge** on the fixture

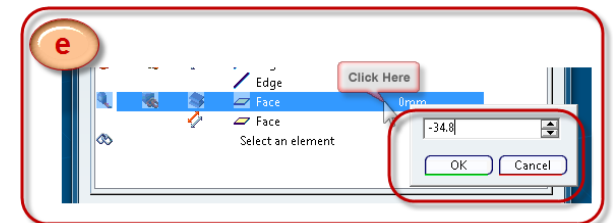
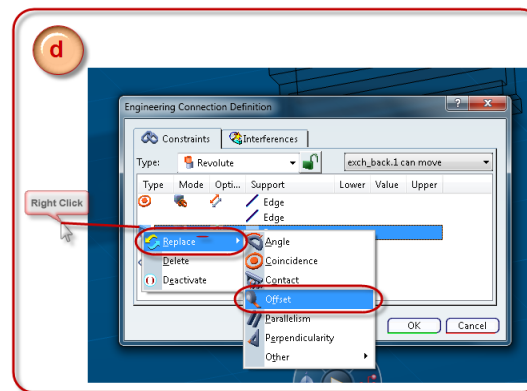
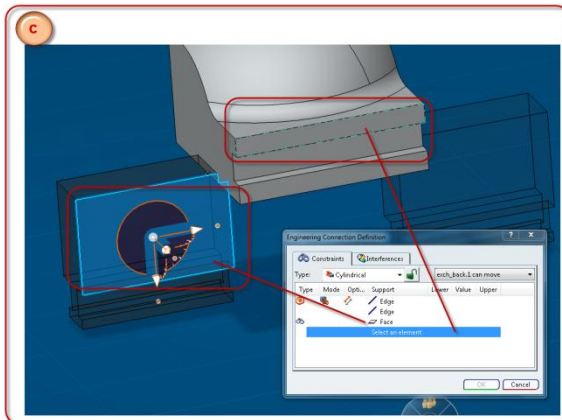
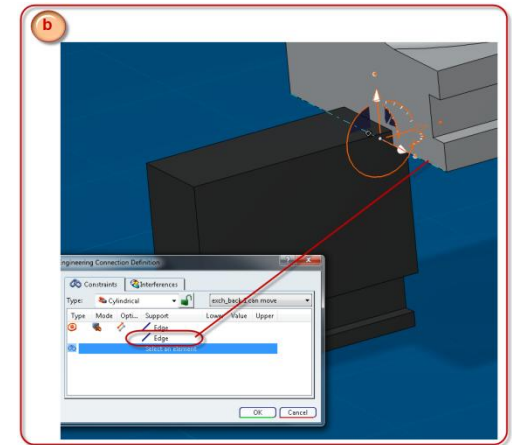
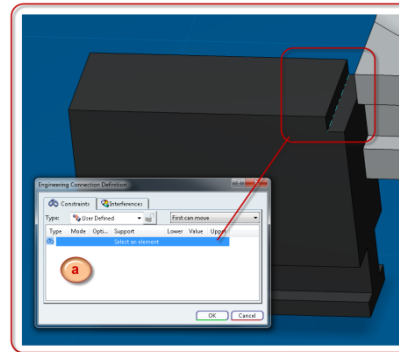
b. Then **select the edge** on the **part**

*The two parts will automatically move and
Be placed as you see in picture b*

c. Select the **face** on the **fixture** first
and then the **face** on the **part**

d. **Right click** on the **type of connection** >
Replace > **Offset**

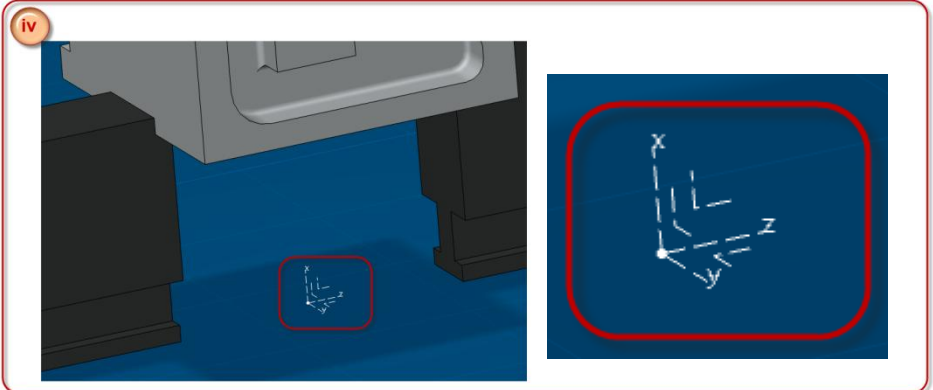
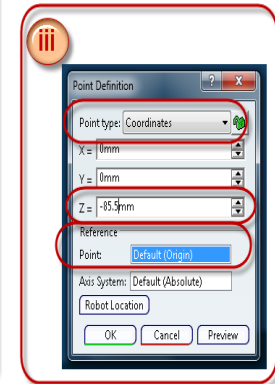
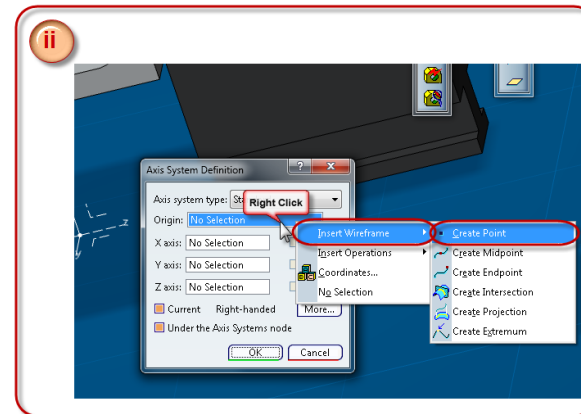
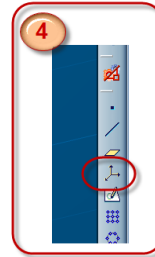
e. Change the **offset parameters** to **-34.8mm**
press **OK** > **OK** > and **YES** to the dialog box
*Repeat the whole **step 3** for the other side*



4. Insert axis system

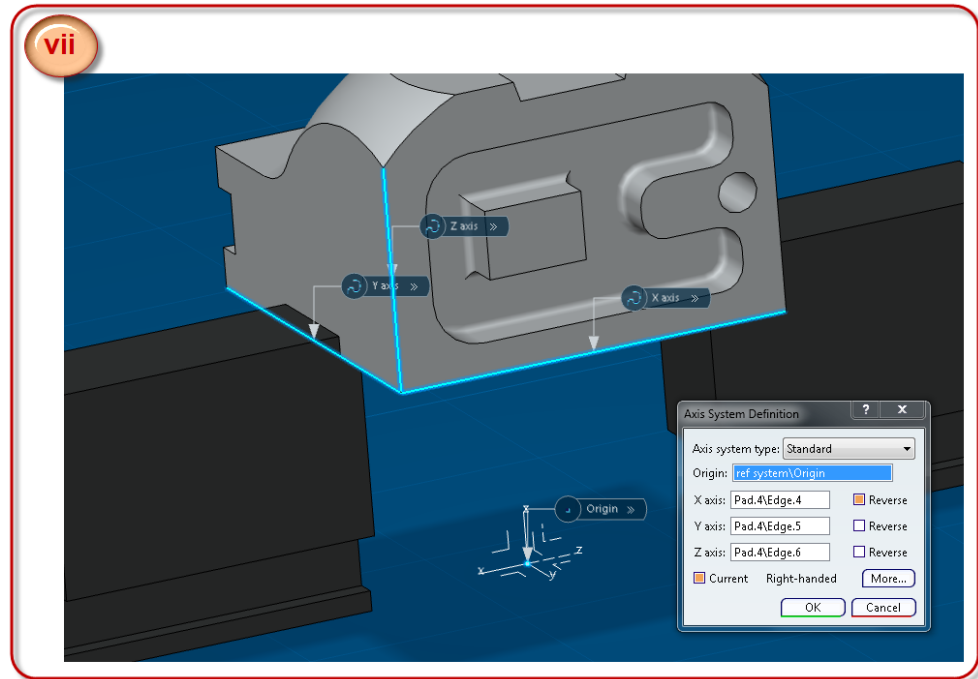
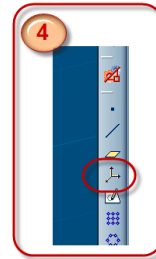
a. Axis system for **machining reference**

- i. Click on the **axis system** > select the **part as 3D shape** > click **OK**
- ii. Right click on **origin selection** > **Insert wireframe** > **Create point**
- iii. Select the **Reference point** on the bottom surface and then select the **point type** as **Coordinates** and edit the **Z = - 85.5mm** > **OK**
- iv. Define the X, Y and Z as you see in the picture
- v. Change name on the axis to **ref system**

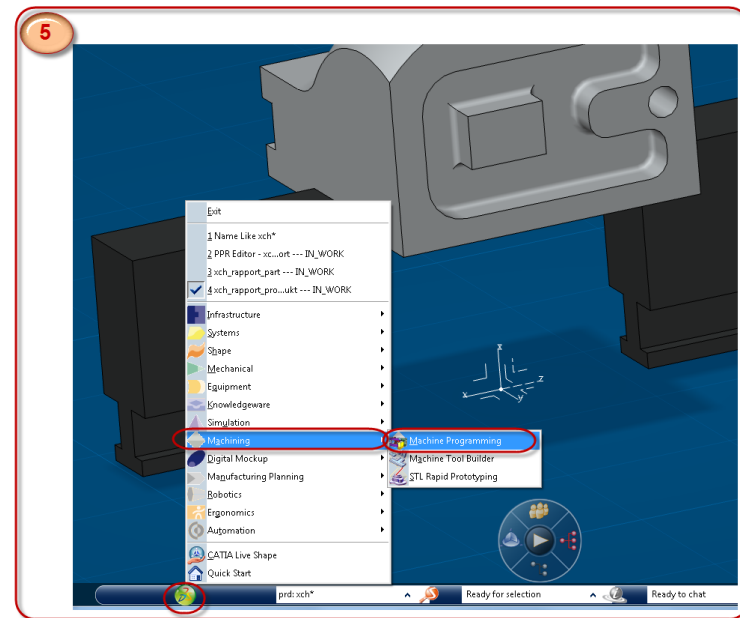


b. Axis system for mount point

- vi.** Repeat step i.
- vii.** Select the **origin** by clicking on the **ref systems origin** then **define the axis** as you see in the picture
- viii.** Name the axis system **mount axis**



5. Start a **Machine Programming** session
- a. Click on the **start** menu > click **Machining** > select **Machine Programming**

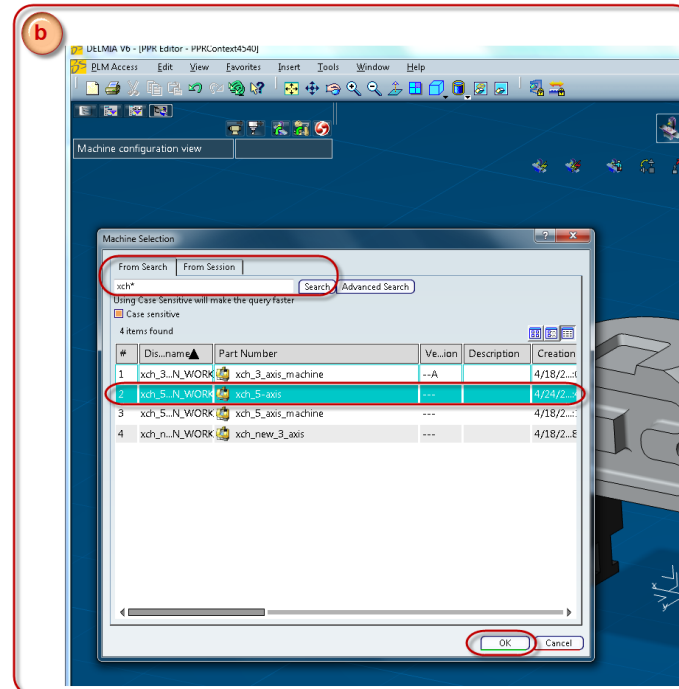
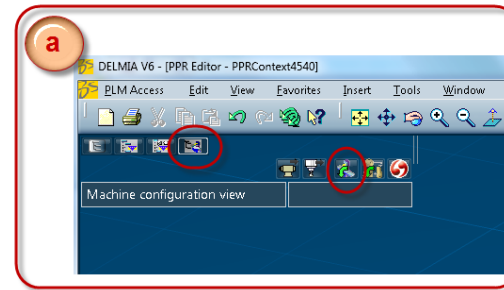


SETUP

1. Importing a machine

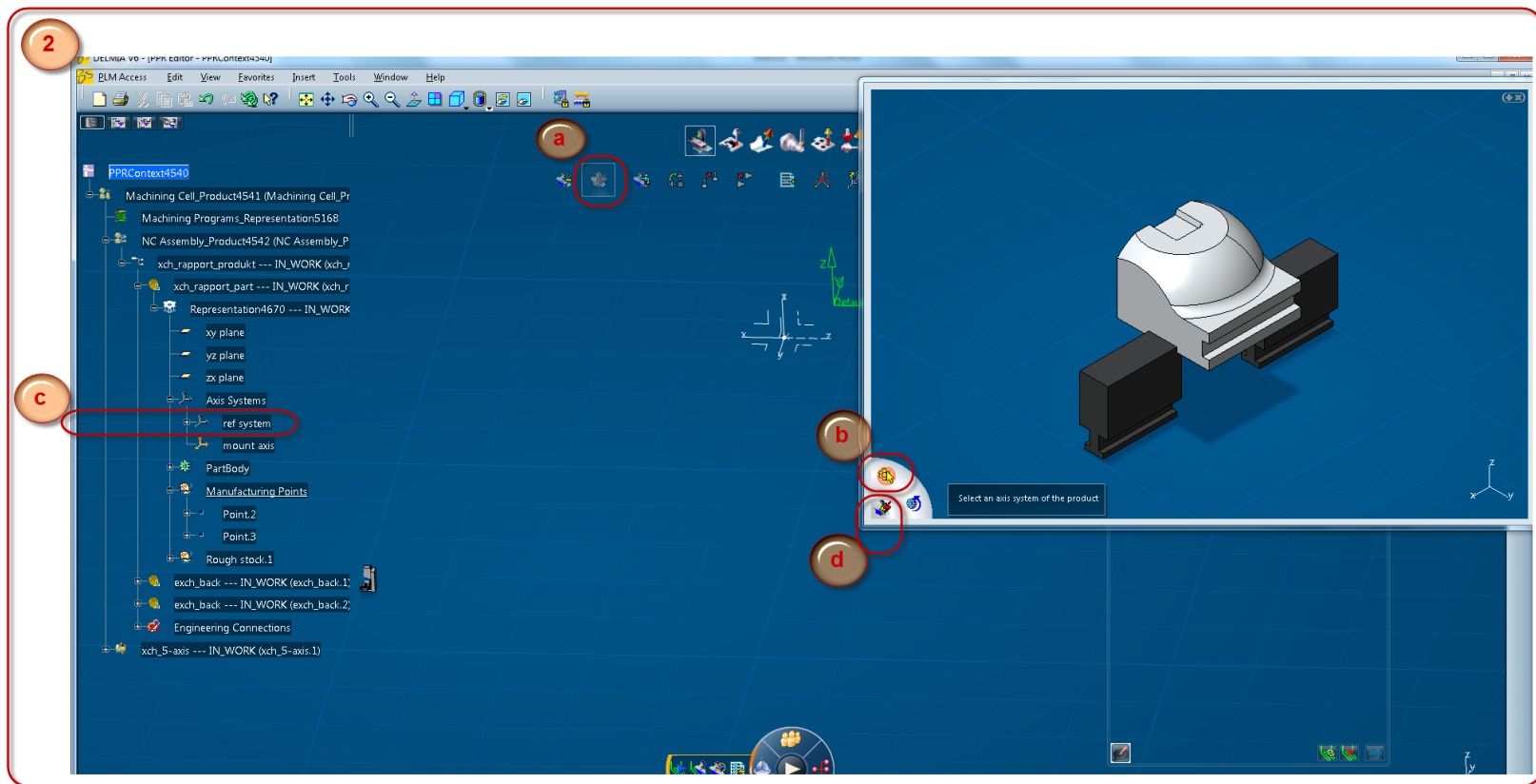
- a. Select the **Machine configuration view** > click on **Assign a machine from database**
- b. Select **from search** > enter the **search string** > **Search** > **choose machine** and click **OK**

*Machine, part and reference system are placed very far from each other. If you want to see them all press **fit all** on the top tool bar.*



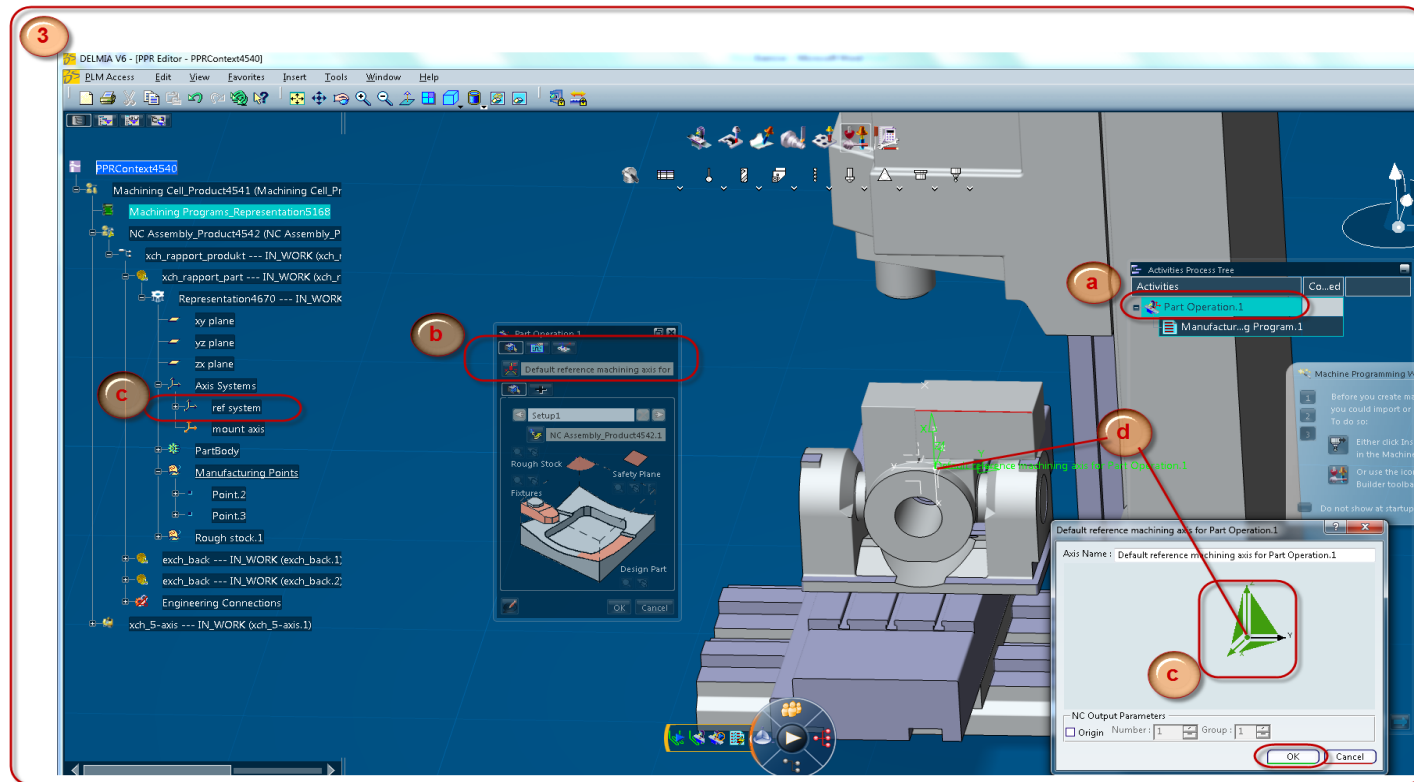
2. Mount product on the machine

- a. Select **Workpiece advanced mount**
- b. Click on **Mount the assembly**
- c. Select the **mount axis** on the PPR tree
- d. Click to **validate** and press **OK**



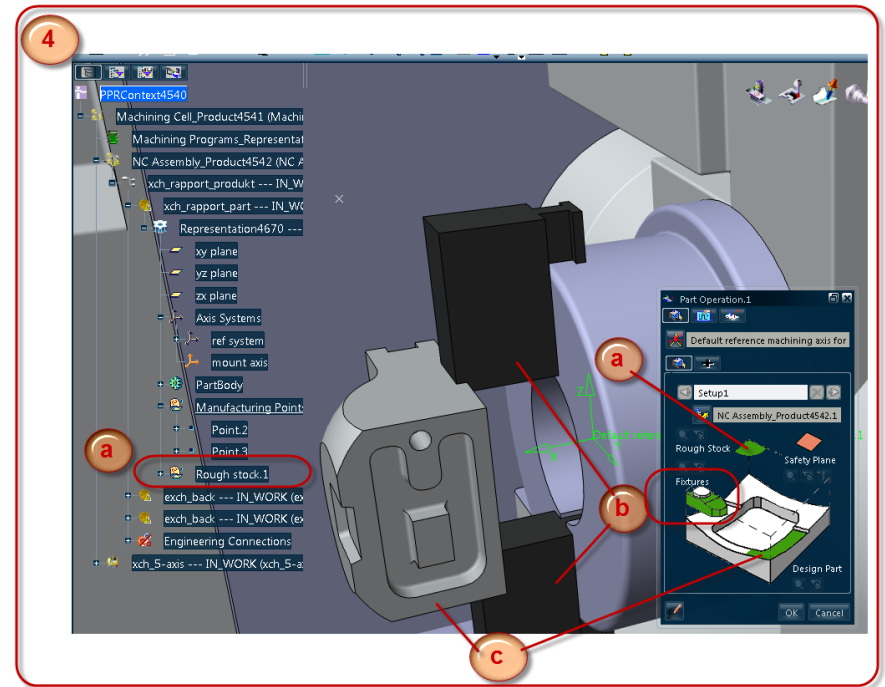
3. Define ref axis in part operation

- a. Double click on the **part operation. 1** in the **Activities Process Tree**
- b. Select **Setup** and **Select reference machining axis system**
- c. In the dialog box that appears click on the **origin** and then select the origin **on the ref system on the part**
- d. Click on the **axis system** in the dialog box and then select the **ref system** on the PPR tree, press **OK**



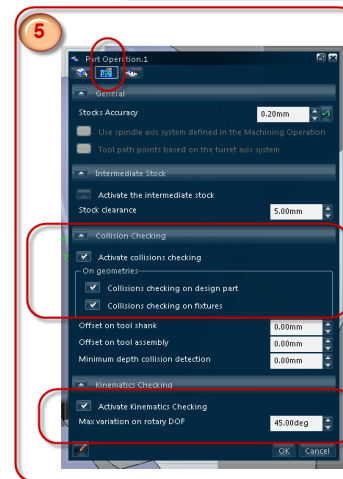
4. Define Part operation

- a. In the **part operation.1** dialog box click on the **rough stock** and then select the **rough stock** in the **PPR tree**
- b. Click on the **fixture** in the dialog box and select the **two fixtures** from the product
- c. Click on the **part** in the dialog box and select the **part**



5. Define options

- a. Click on **options** and select
Activate collision checking
Activate Kinematics Checking



6. Tools

Tools will be imported from an already created tool catalog in the database.

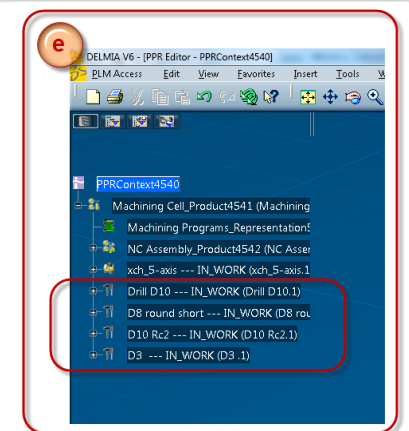
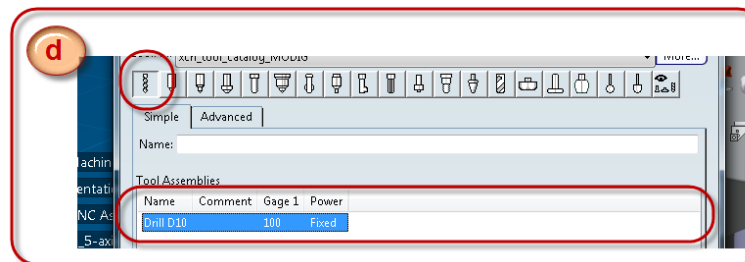
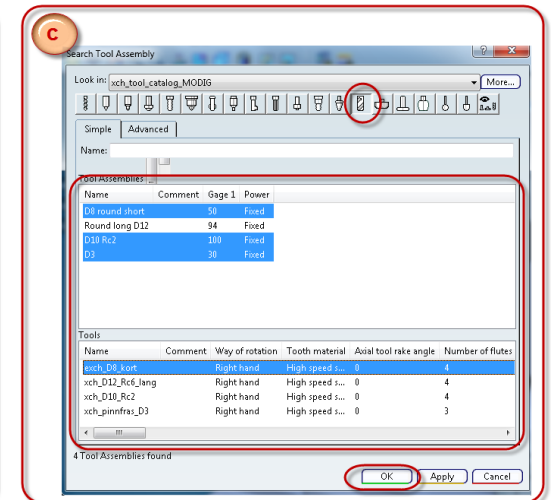
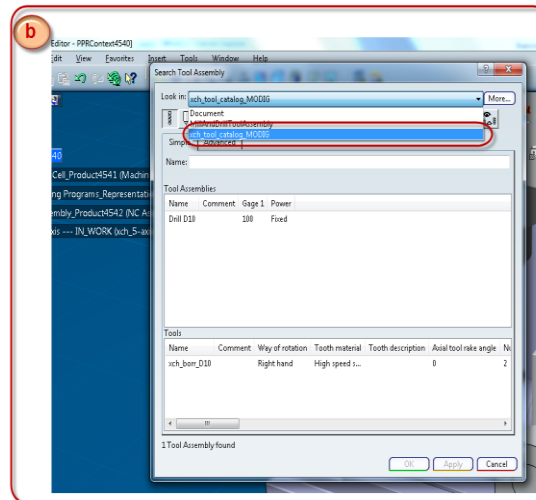
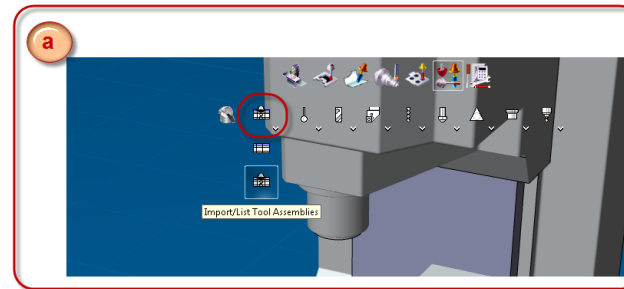
a. Click on **Import/List Tool Assemblies**

b. Select the **required tool list**

c. Select **end mill tools**, mark the tools that are required: **D8, D10 and D3**

d. Repeat step a and b again, this time select **drill tools** and **Drill D10**

e. Imported tools are placed in the **PPR tree**

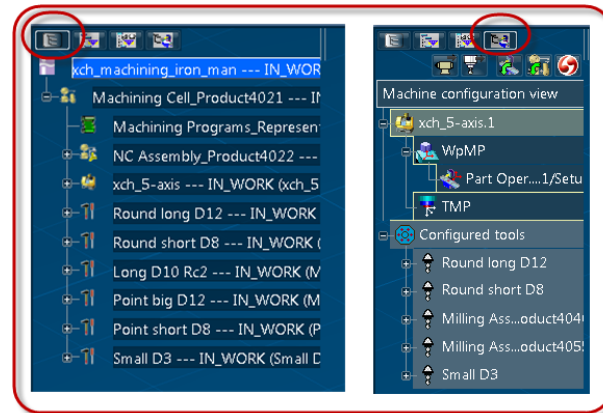


OPERATION

Overview

On the top left corner is the **PPR tree** where the whole **Machining Cell Product** is with saved data as the **Assembly, the Machine and the tools**.

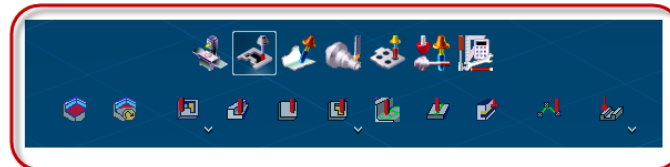
On the right is the **Machine configuration view** where you can see **Work piece mount point** **Tool mount point** and the **Configured tools** for the operations



Prismatic Machining Operations

Pocketing, Profile contouring

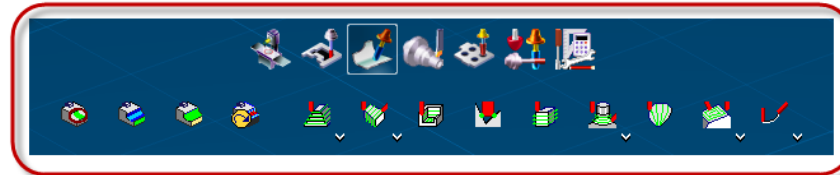
Point to Point



Surface Machining Operations

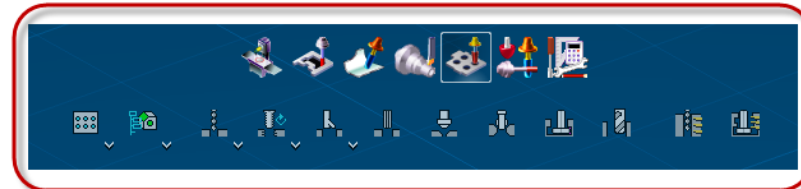
Roughing, Multi-axis sweeping

Multi-axis curve, Multi-axis flank contouring

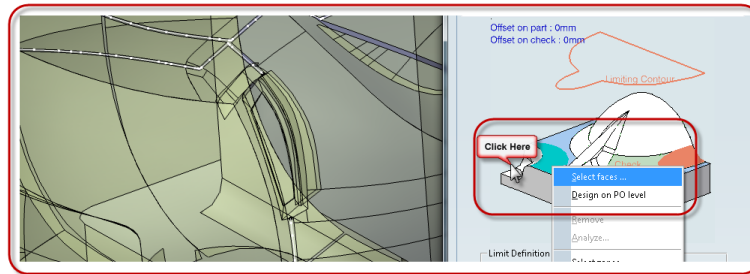


Axial Machining Operations

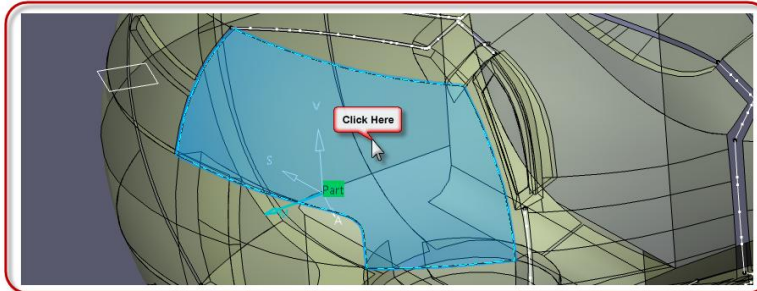
Drilling



To select faces for a **machining operation** you must **first click in the operation window** on what you want to select
In this example **multi-axis sweeping** on a **section of the face**.



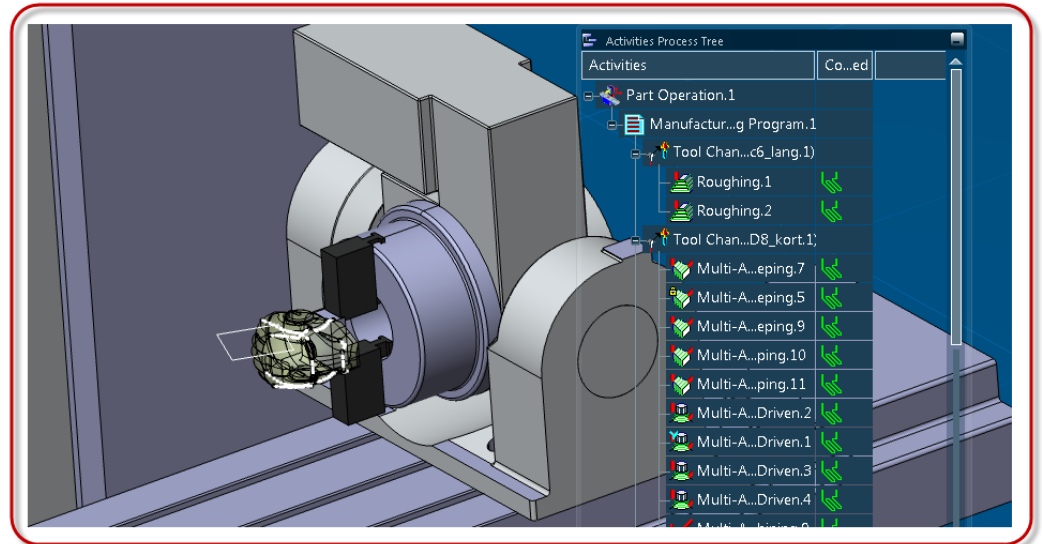
Secondly you choose the section on the face you wish to machine.
This method is used for all selections on the part, Surfaces, bodies, guiding lines, limiting lines etc.



The **Activities Process Tree** is where each created **operation** is listed together with the whole **Part Operation.1** the **Manufacturing Program** and **Tool changes**.

All the rest of the operation will follow each other in the tree.

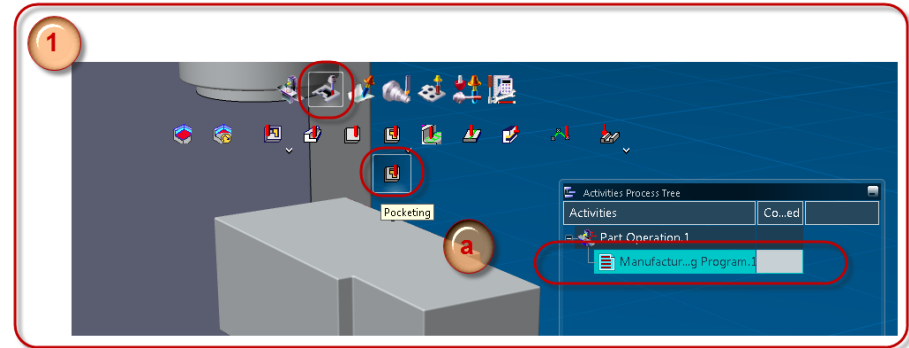
The **Green pattern** behind a operation means that the operation have been **computed**.



Starting with the 3-axis operations.

1. Pocketing.1

- Mark the **Manufacturing program** in the **Activities Tree** click on **Prismatic Machining Operations** and select **Pocketing**



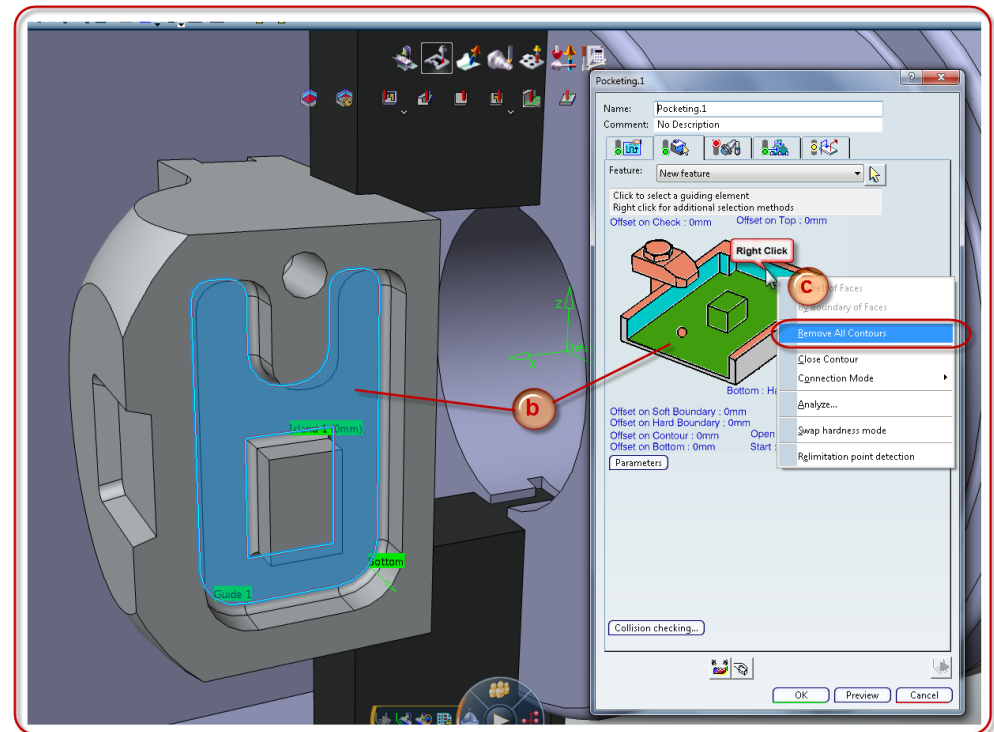
The Pocketing dialog window will open, the window

On the picture is the Geometry Parameters. Here you

choose the geometries to operate in the selected operation

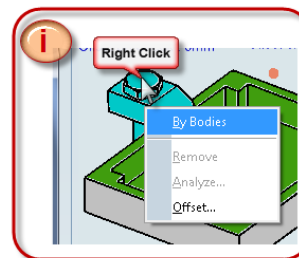
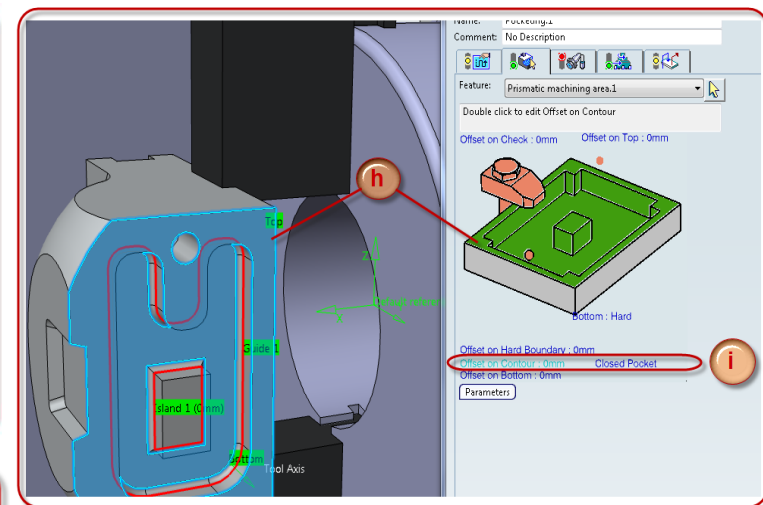
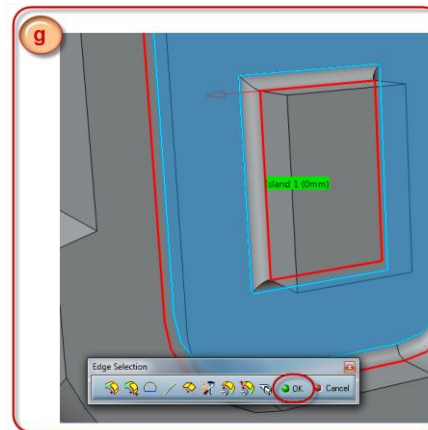
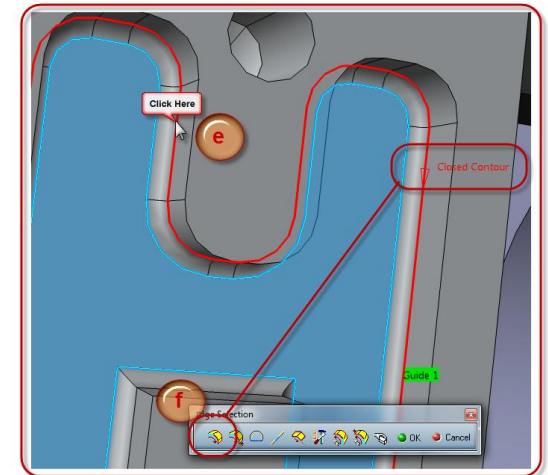
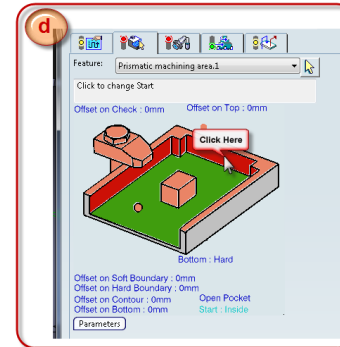
- Start by selecting the **bottom** in the **Operation window**, then select the **Bottom** on the part

The contour on the **Island** and on the **Walls** will automatically be selected, **Green** in the operation window. Because of the radius on the bottom of the pocket these needs to be defined separately.



- Right click on the wall and select **Remove All Contours**, repeat this step for the **Island**

- d. Click on the **wall** to select a **guiding element** for the pocketing operation
- e. **Click on the edge** as you see in the picture
- f. Select **Navigate on belt of edges** the dialog box until you have a **Closed Contour**, press **OK**
- g. Repeat for the **Island** and select the **contour** as you see in the picture. This time you have to **select the edges manually** because of the angle
- h. Select the **top**
- i. Simply click on **Open Pocket** and it will change to **Closed Pocket**
- j. Right click on **Check** in the **Operation window** and select **By Bodies**. Select the **two fixtures** who are attached to the main part in the assembly product



k. Continue to the **Strategy** parameters, in the **Machining** tab select parameters as in the picture

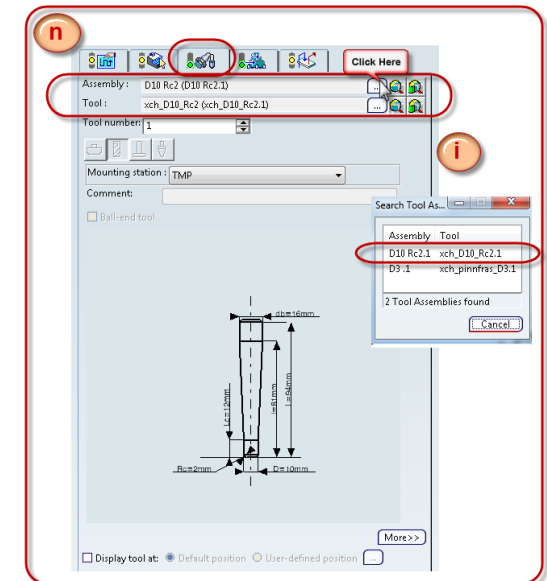
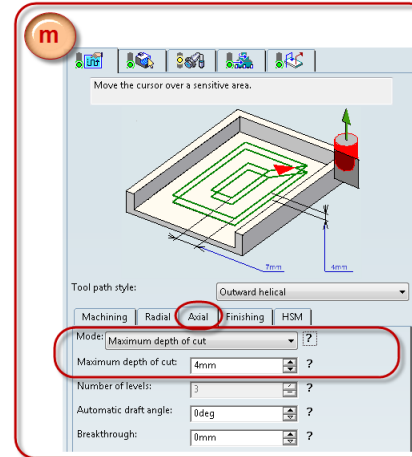
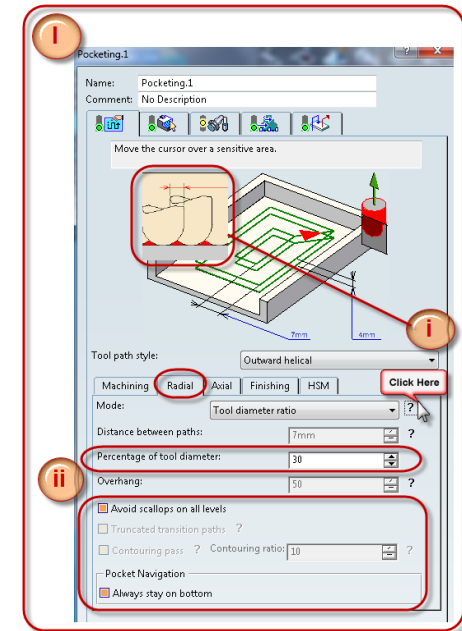
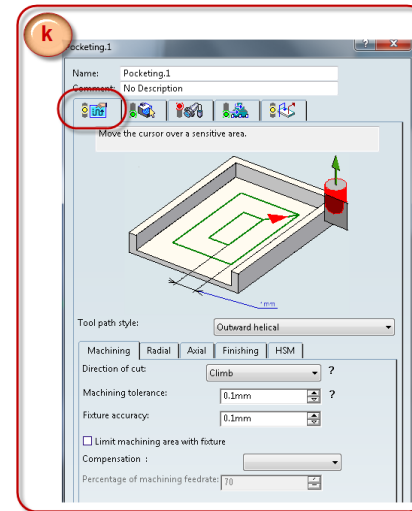
l. In the Radial tab select **Mode** as **Tool diameter ratio**

- By clicking on the **question mark** a **dialog box** appears with an explanation
- Change the parameters to **Percentage of tool diameter: 30** mark **Avoid scallops on all levels** and **Always stay on bottom**

m. In **Axial** tab select **Mode** as **Maximum depth of cut** and the **Parameter to 4mm**

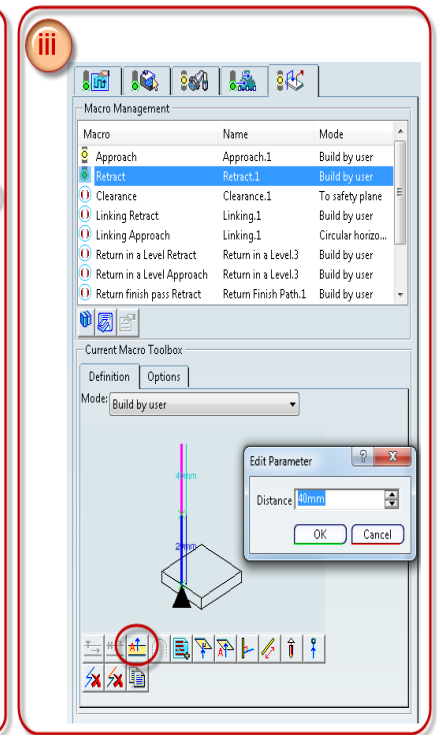
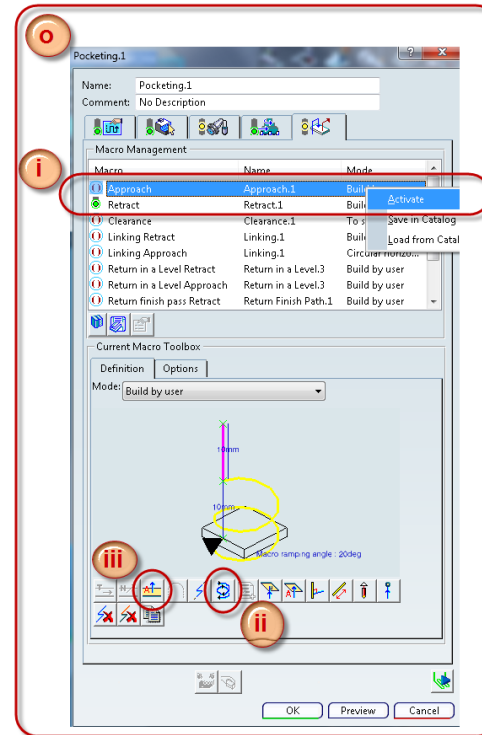
n. Continue to the **Tool selection**

- Click on **Select tool assembly from current machining cell**
Select the tool assembly **D10 Rc2**

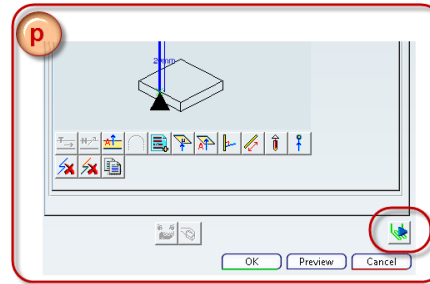


o. Continue to **Macro parameters**

- i. Right click on **Approach** and **Retract** and select **Activate**, **Green light** will appear when activated and computed
- ii. For **Approach**: select **Add Helix Motion** and enter the parameters by double clicking on them
Macro ramping angle: 20 deg
Axial ramping: 10 mm
- iii. Select **Add Axial Motion**
For **approach**:
Axial: 10 mm
For **Retract**:
Axial 20 mm
Axial 40 mm

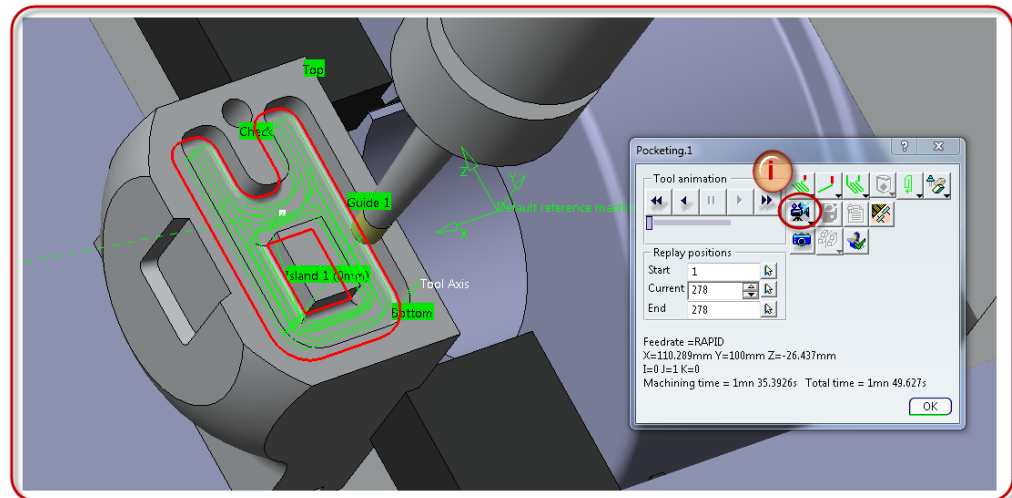


- p. To **compute** click on **Tool path replay**, if any dialog box appears click **OK** and ignore

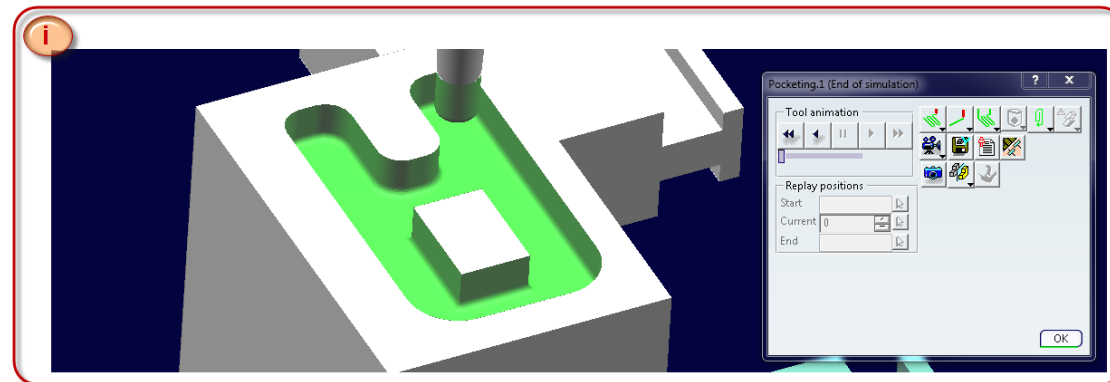


Tool path simulation appears, here you can verify that the tool path is correct.

- i. Select **Video** to see the simulation on the tool path with **material removal**



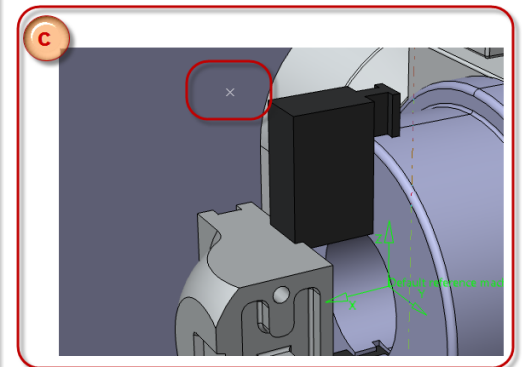
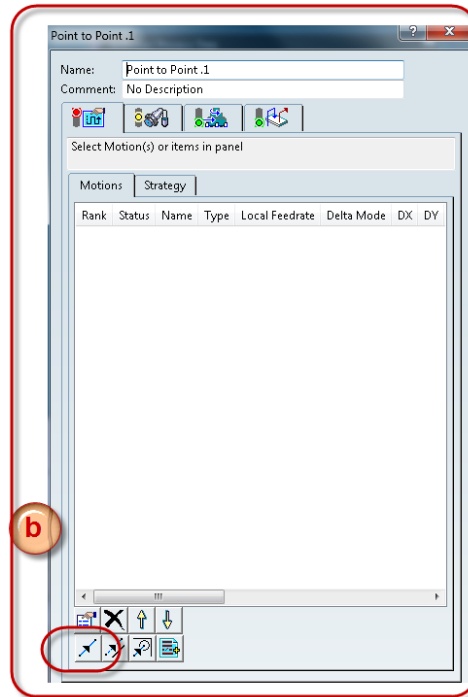
*Click **OK** in the Tool path dialog box and then OK in the Operation window. The Operation is placed In the **Activities Process Tree***



2. Point to Point

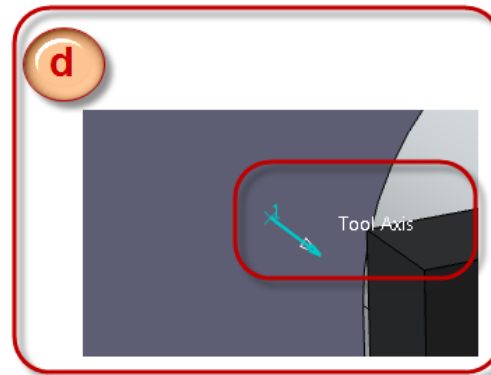
Because of the 3-axis operations, the machine needs space to change side on the part to avoid collision with the tool. Therefore a point has been created that will give the rotation axis enough time to switch side.

- a. Select **Point to Point** operation
- b. Select **Goto Point**
- c. Select the **point**



A tool axis appears on the point. To make the transition for the next operation easier this axis needs to be rotated.

d. Double click on the **Tool Axis** text



e. Change the parameters to

I: 0

J: 0

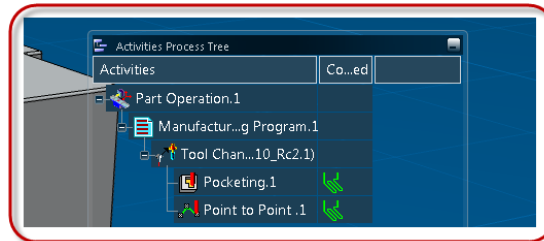
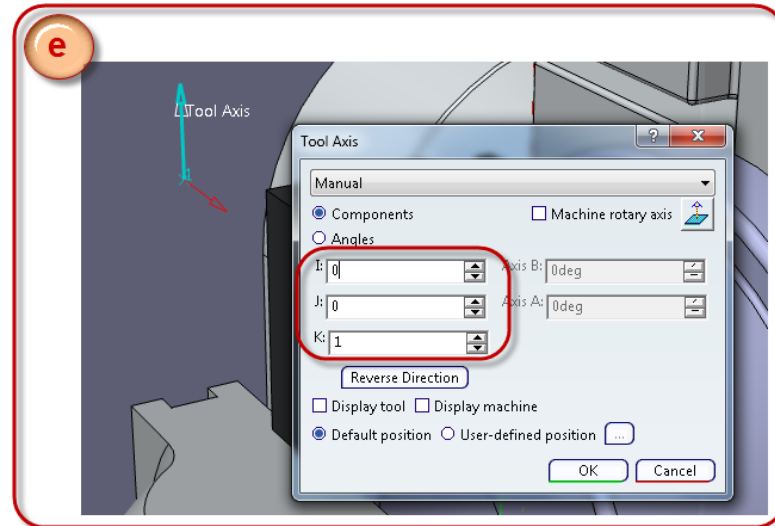
K: 1

Select **OK**,

Compute and accept.

The operation is placed

in the **Activities Process Tree**



3. Profile contouring.1

Select **Profile Contouring**



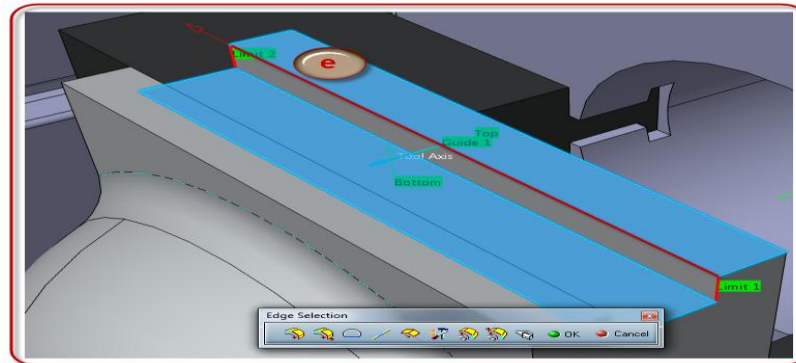
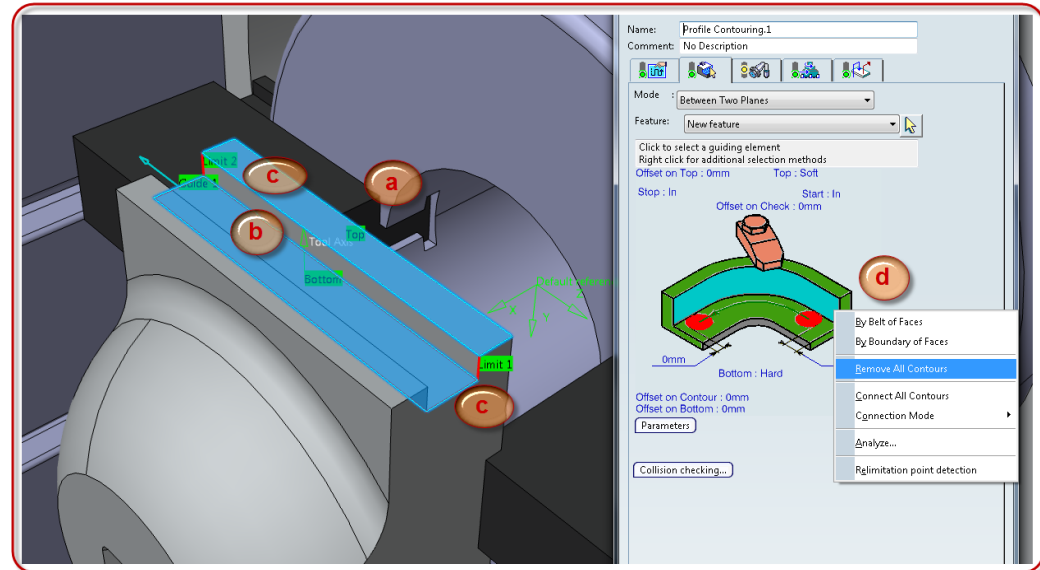
a. Select **top**

b. Select **bottom**

c. Select **limit lines**

d. Right click on the **wall** and
select **Remove All Contours**

e. Click on the **wall** to select
Guiding element.
Select the **guide** as you see
in the picture



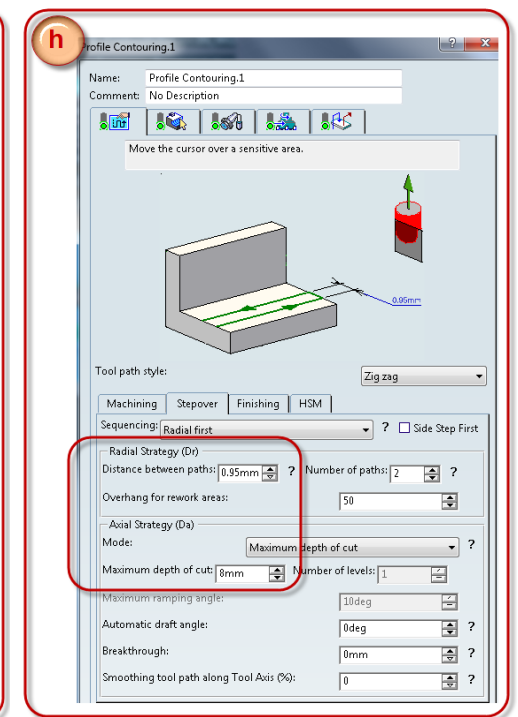
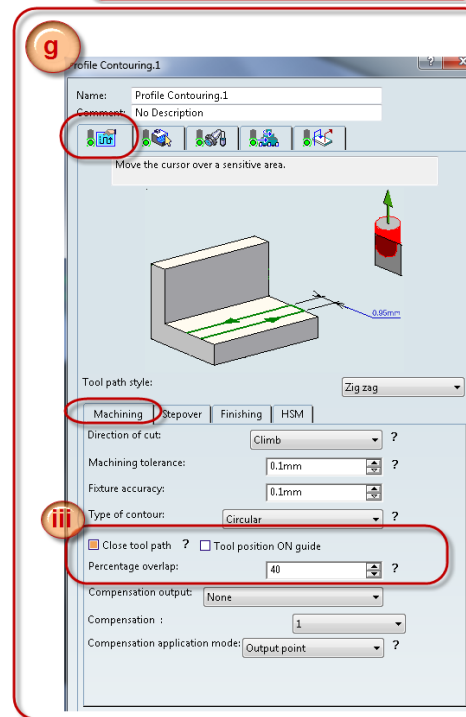
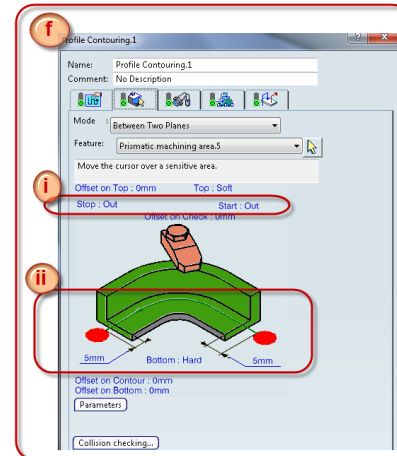
f. Edit parameters as you see in the picture

- i. **Stop: Out**
Start: Out
- ii. **Offset on start and stop: 5mm**

g. Continue to **Strategy** parameters
edit parameters if needed

- iii. **Mark Close tool path**
Percentage overlap: 40

h. Move on to **Stepover** strategy edit
Distance between paths: 0.95 mm
Maximum depth of cut: 8 mm



i. Use the **same tool** as in the **Pocketing operation** earlier

j. **Activate Approach and Retract** macro

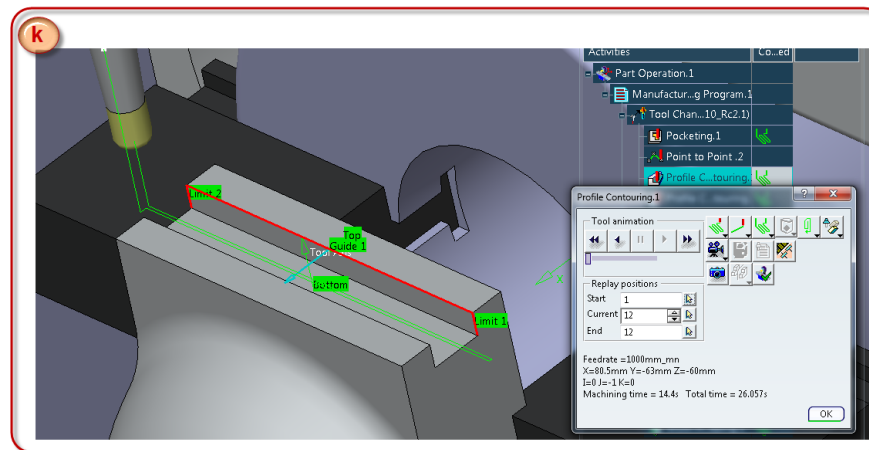
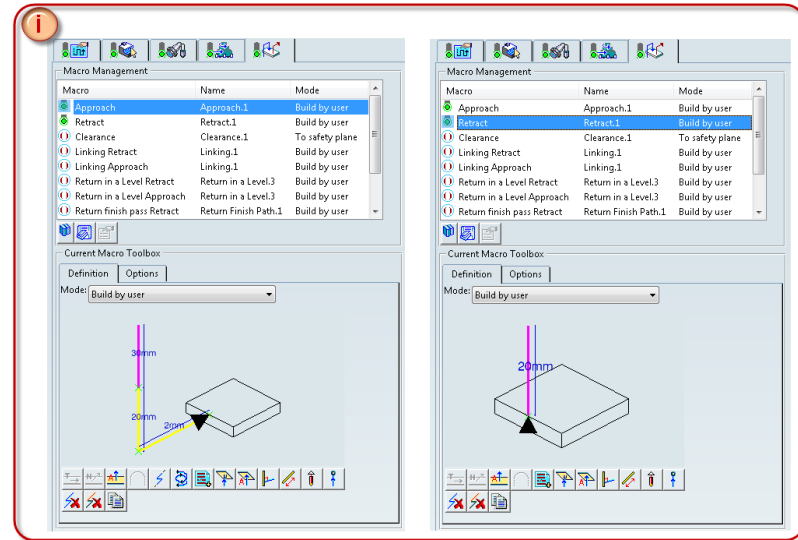
Use parameters as in the picture.

Approach: **Horizontal: 20 mm**

Axial: 50 mm

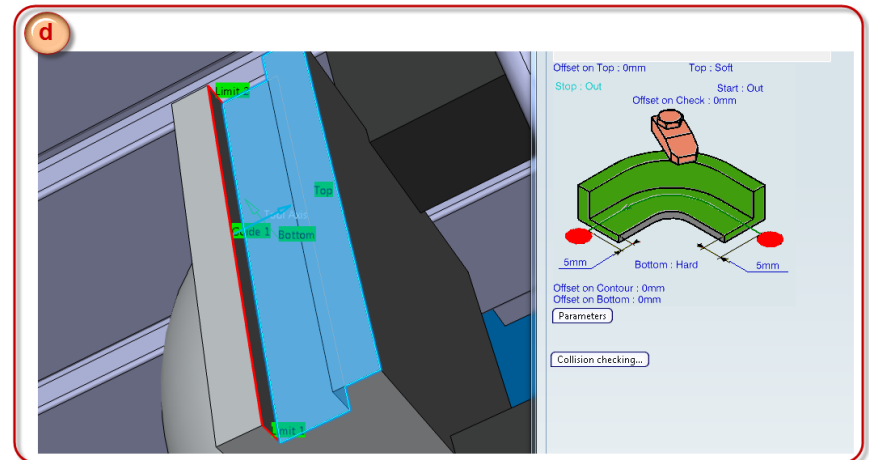
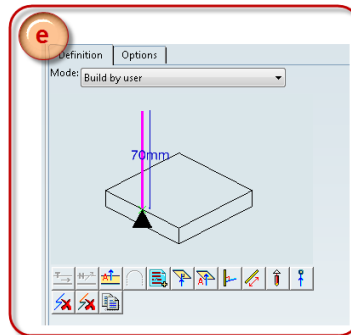
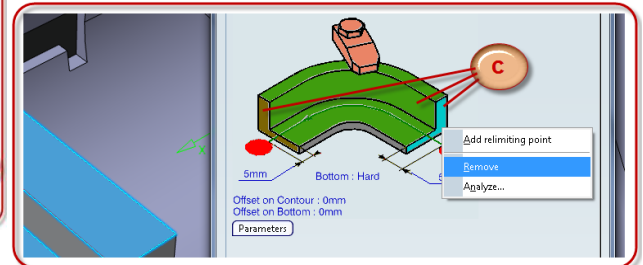
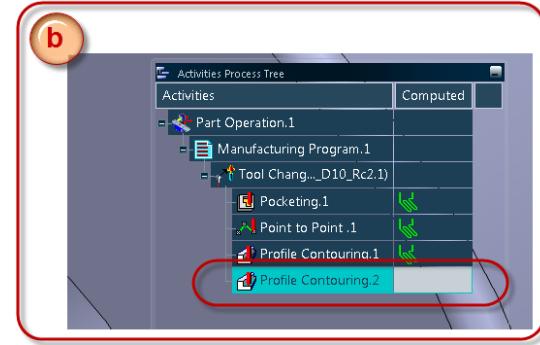
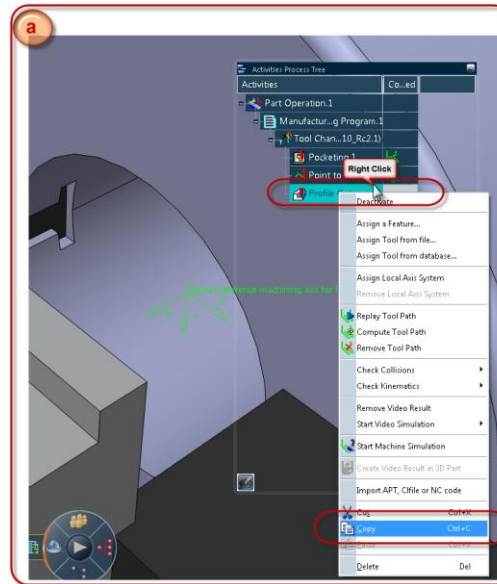
Retract: **Axial 20 mm**

k. **Compute** to see **Tool path**



4. Profile contouring.2

- Right click on the **Profile contouring** you just created, **select copy**
Then right click on the same, **select paste**
- Double click on **Profile Contouring.2**
- Remove the **Guiding contour** and the **limiting lines**
- Select **new Guiding contour** and **limiting lines** as you see in the picture
- Change the **Retract macro**
Axial: 70 mm
- Compute** and select **OK**



5. Pocketing.2

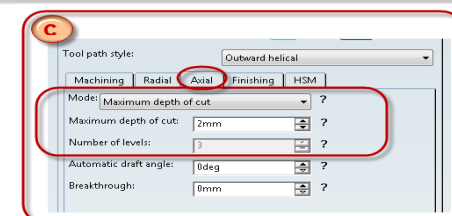
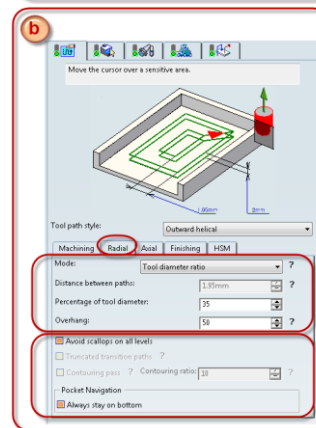
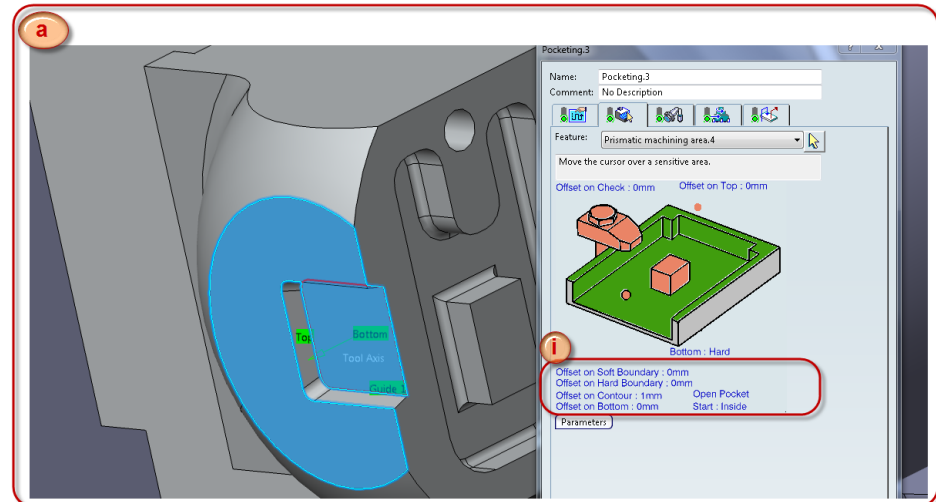
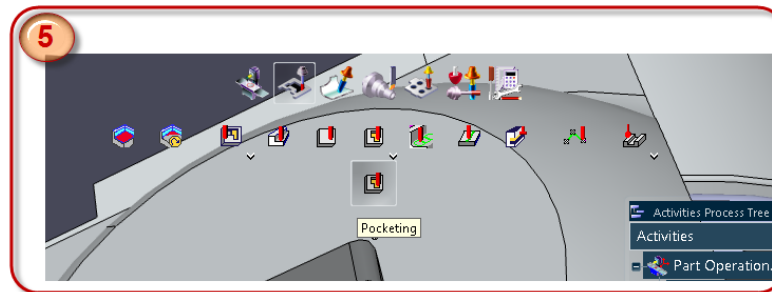
Select **Pocketing** operation

- i. Select the **Top** and **Bottom** as shown in the picture
 - i. **Offset on Contour: 1 mm**

- ii. In the **Radial Strategy** tab select
Mode: Tool diameter ratio
Percentage of tool diameter: 35
Overhang: 50
 Mark **Avoid scallops** on all levels
 and mark **Always stay on bottom**

- iii. In the **Axial strategy** tab select
Mode: Maximum depth of cut
Maximum depth of cut: 2 mm

- iv. In the **High speed milling (HSM)** Strategy tab select
Corner tab:
Corner radius: 1 mm
Limit angle: 10 deg
Extra Segment overhang: 0.5 mm



- v. In the **HSM strategy tab** select
Transition tab:

Transition radius: 1mm

Transition angle: 45 deg

Transition length: 0 mm

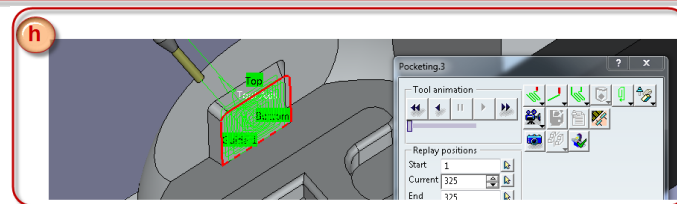
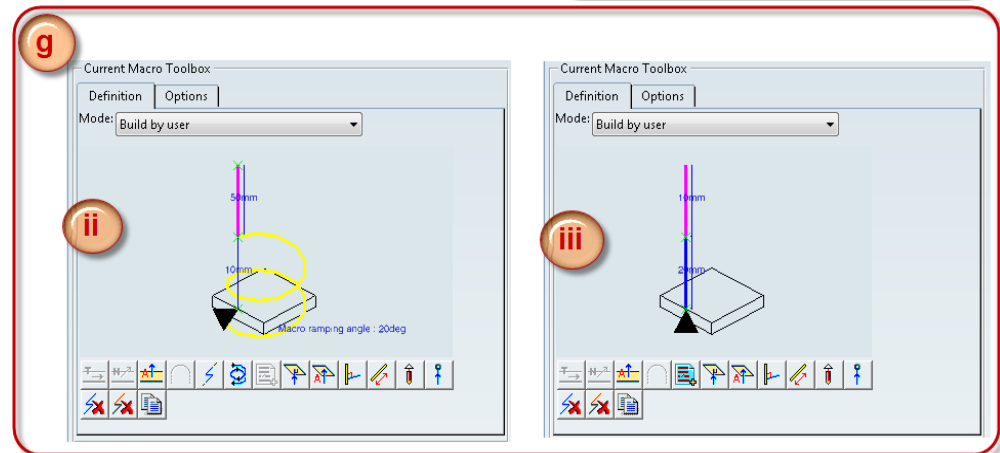
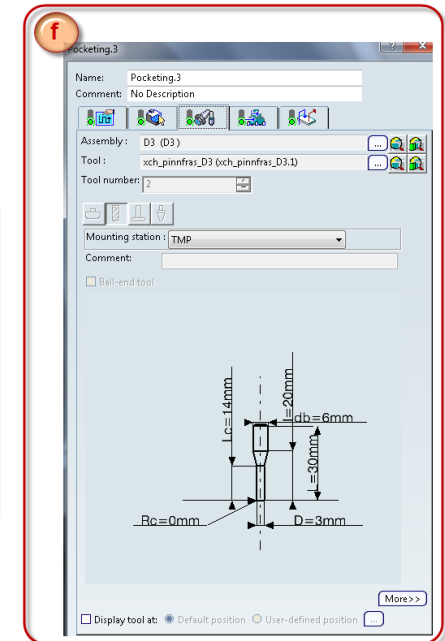
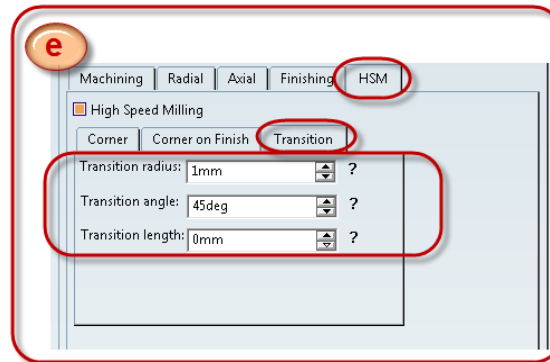
- vi. In the Tool Change tab
Select **Tool assembly** from current
Machining Cell

Select the **Tool Assembly D3**

- vii. In the Macro tab, activate
Approach and Retract macro

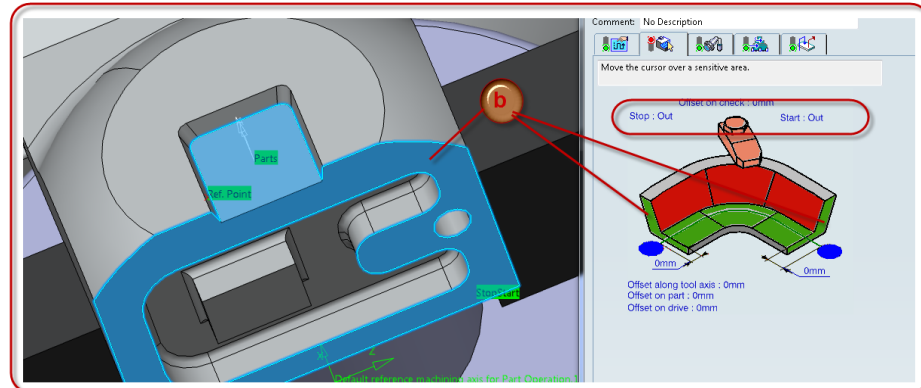
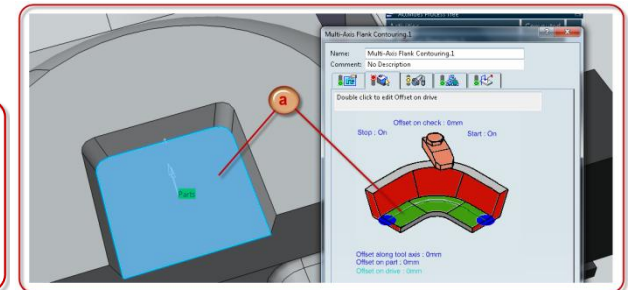
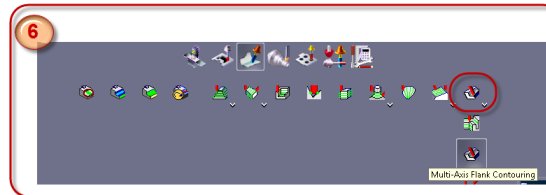
- ii. **Approach:**
Ramping Angle: 20 deg
Ramping Axial: 10 mm
Axial: 50
- iii. **Retract:**
Axial: 30 mm

- viii. **Compute** and select **OK**

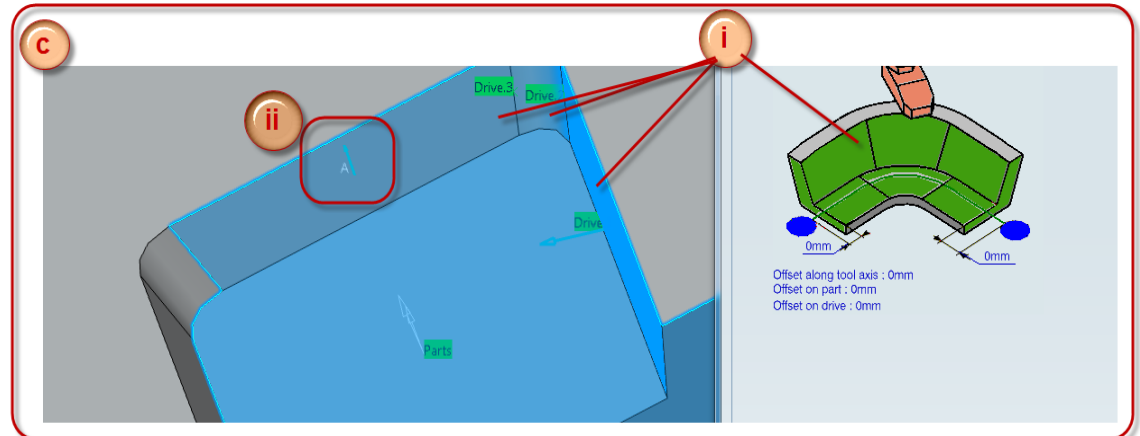


6. Multi-axis Flank Contouring

- i. In the **Geometry** tab select **bottom of the pocket** as **part**
- ii. Select the **side** as both **Start** and **Stop**
Change the **Start** and **stop** to **Out**
- iii. Select **Drive Elements**

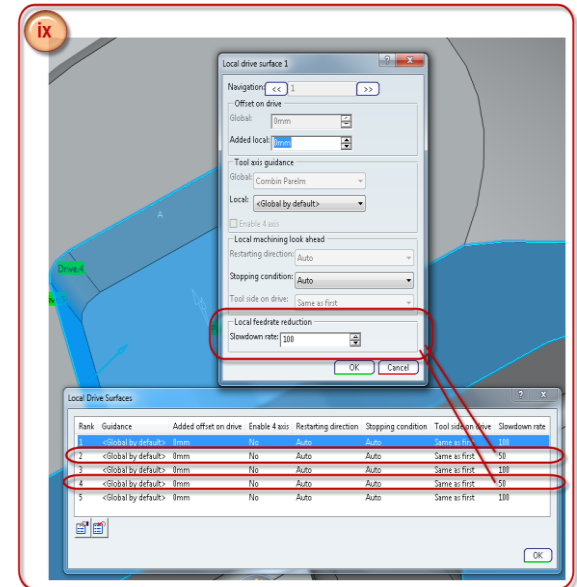
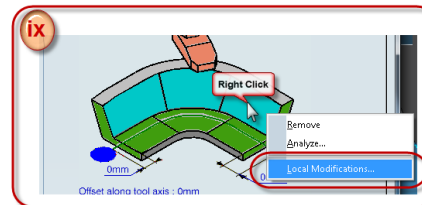
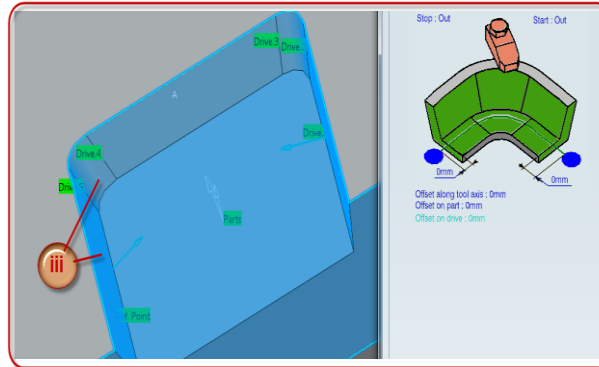


- i. First select the **three surfaces** as **Drive Elements**
- ii. Click on the **arrow** to **change direction** on the **Drive Elements**



iii. After **step ii**. Select **Drive.4** and **Drive.5**

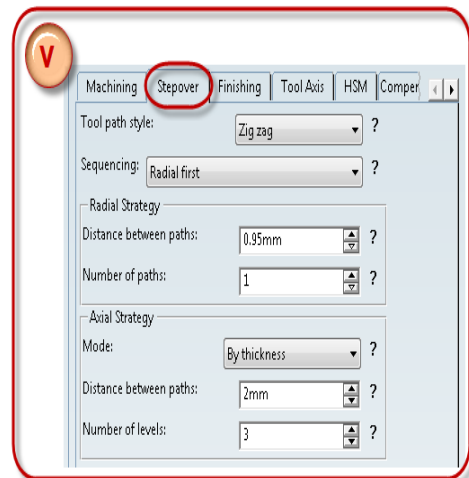
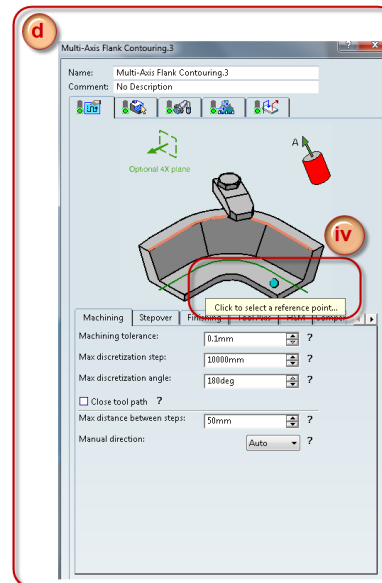
- ix. To change parameters for each **Drive** right click on the **Drive Elements** selection and select **Local Modifications**. Double click on **Drive 2** and **Drive 3** and change the **Slowdown rate: 50**



- iv. On the **Strategy Parameters** tab
iv. Click on the **reference point** and then click on any of the **Drive surfaces**

- v. In the **Stepover** tab Edit parameters
Radial strategy:
Distance between paths: 0.95 mm

Axial strategy:
Mode: By thickness
Distance between paths: 2 mm



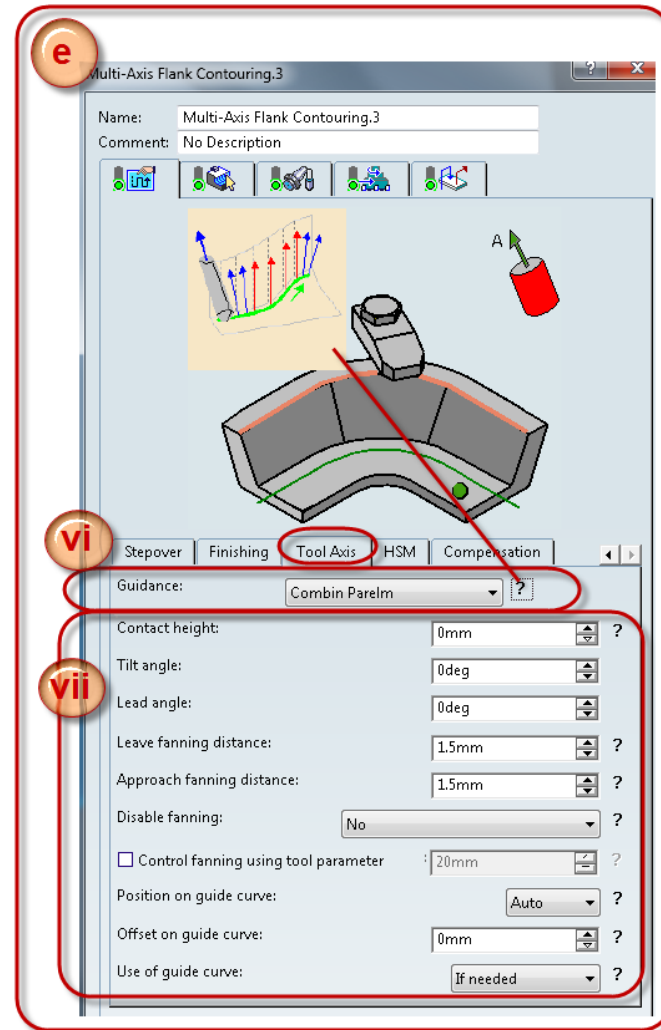
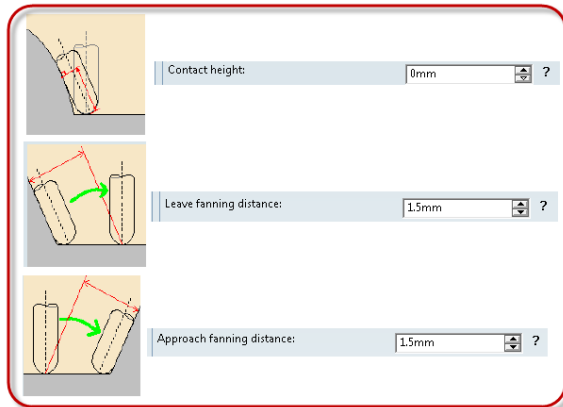
v. Select the **Tool Axis** tab

vi. For **Tool Axis Guidance**:
Select **Combin Parelm**

*Definition of **Combin Parelm** is shown by clicking on the question mark as you see in the picture*

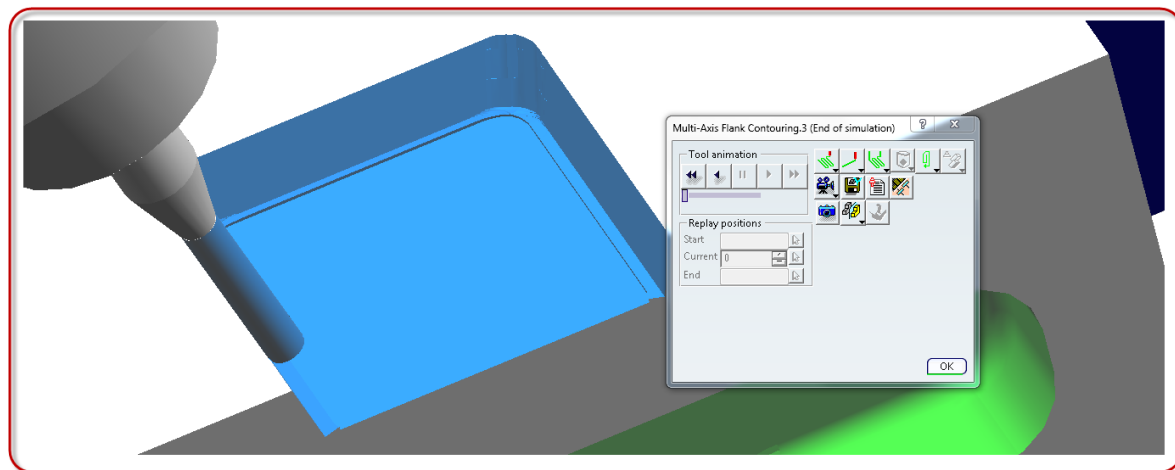
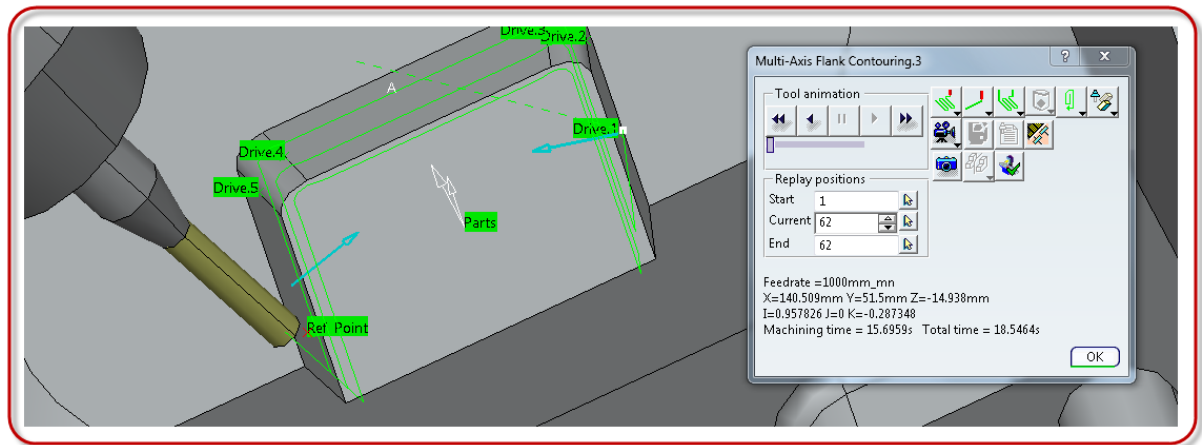
vii. Edit the parameters to those you see in the picture

The definition to the parameters is in the Picture below



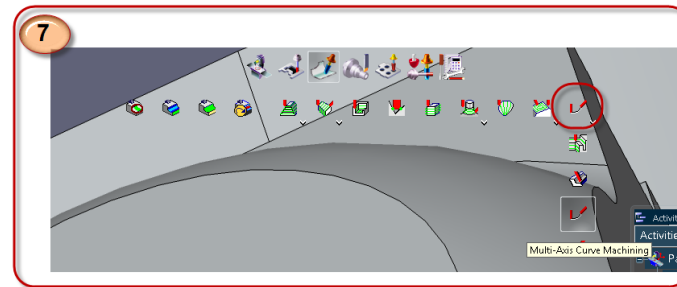
- vi. Continue to **the Macro parameters**
activate the **Approach** and **Retract**
Macros to:
Axial: 10 mm for both

- vii. **Compute** and select **OK**



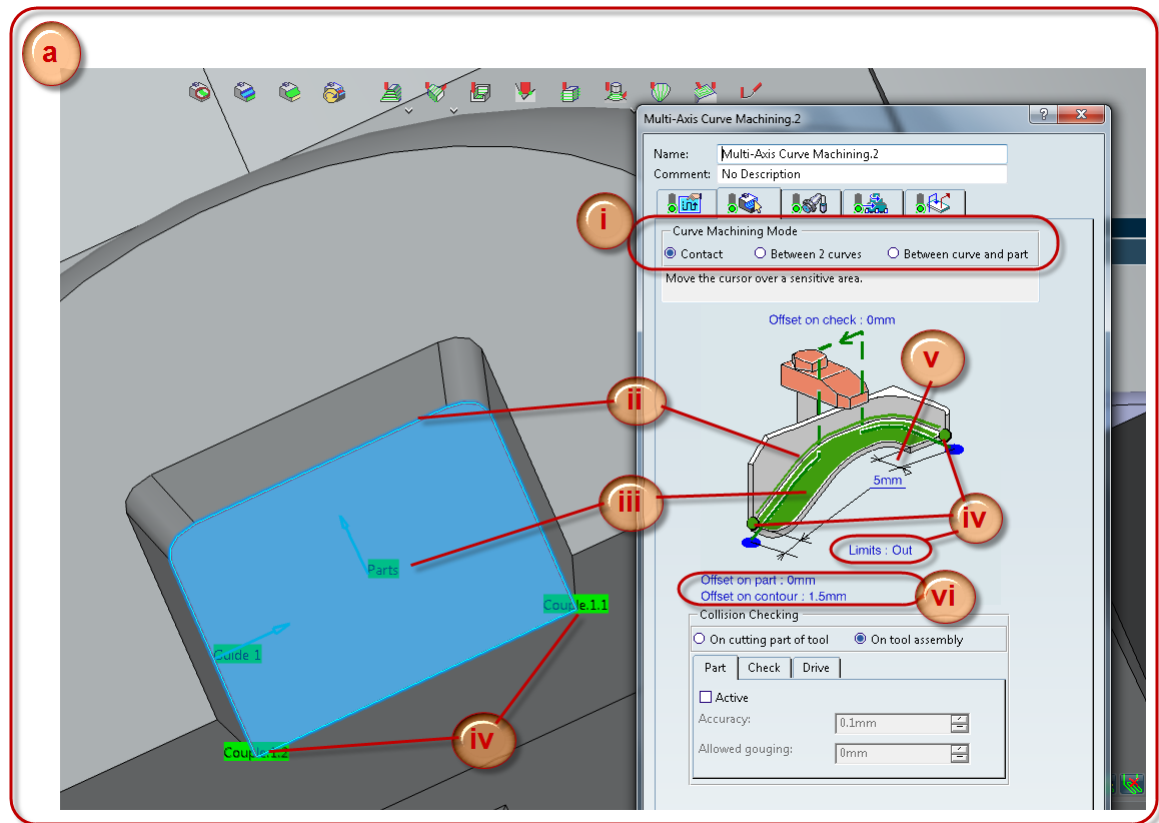
7. Multi-Axis Curve Machining

Select Multi-axis Curve Machining

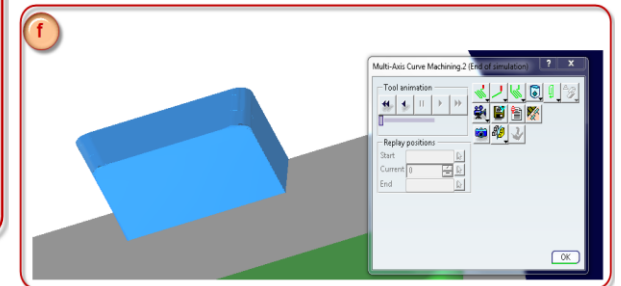
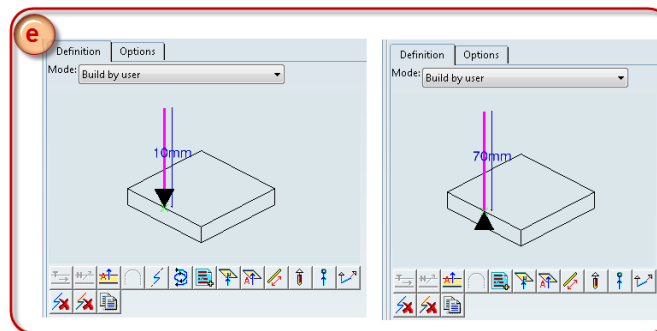
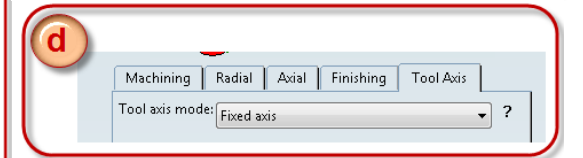
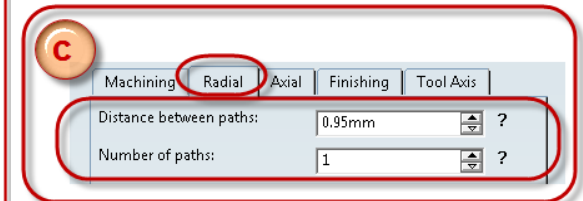
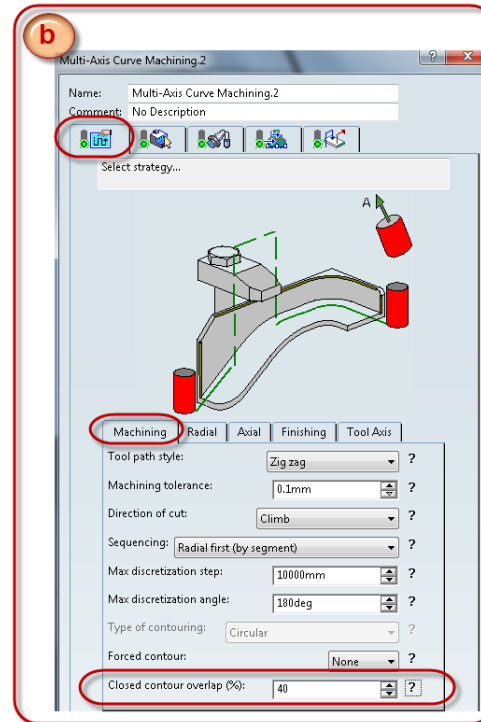


a. Select Geometry parameters Tab

- i. **Curve Machining Mode**
Choose **Contact**
- ii. Select **guiding element**
choose the **bottom contour**
- iii. Select the **bottom** of the pocket as the **Part**
- iv. Select **limit points** by clicking on the **edges** of the **bottom contour**
Set **Limits: Out**
- v. Set **value to 5 mm**
- vi. Set value on the **Offset on contour: 1.5 mm**



- b. Continue to **Strategy parameters**
And edit parameter
Closed contour overlap (%): 40
- c. Select **Radial Strategy** tab
Distance between paths: 0.95 mm
- d. **Tool Axis** strategy tab
Tool axis mode: Fixed Axis
- e. Continue to the **Macro Parameters**
Activate Approach and Retract
Edit Approach motion to:
Axial: 10 mm
Retract motion to:
Axial: 70 mm



- f. **Compute** and select **OK**

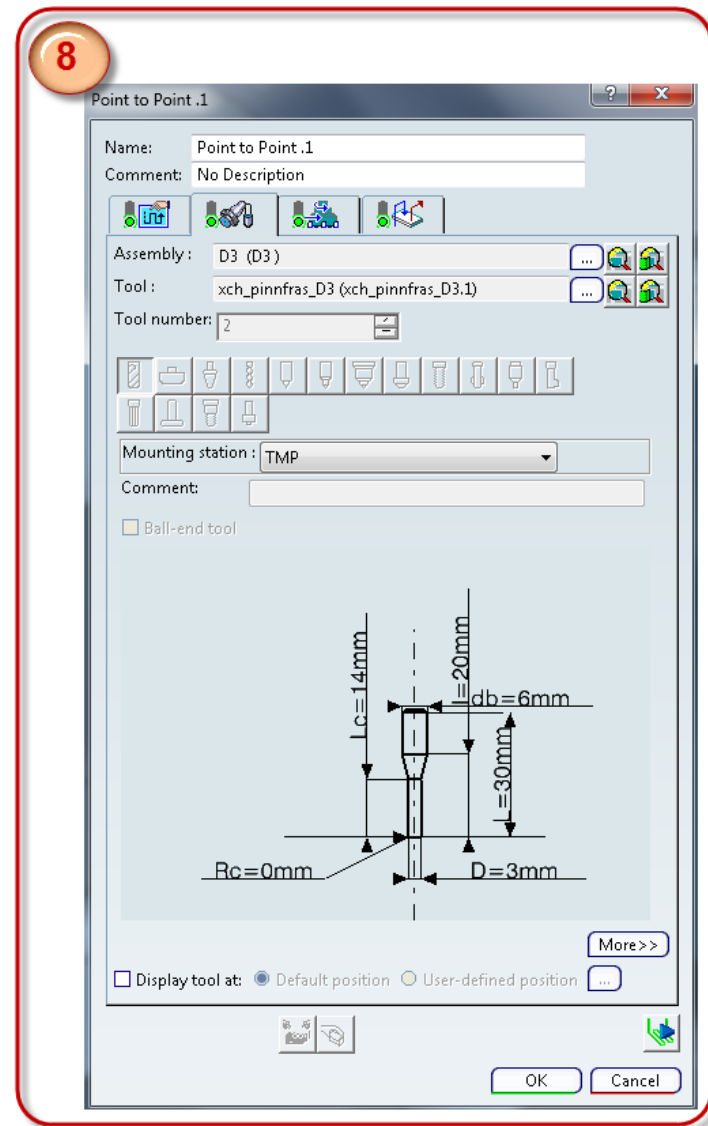
8. Point to Point

Copy the **Point to Point Operation** from **step 2**

Right click on the **Multi-axis curve machining Operation** in the **Activities Process tree**, the one that was created previously, select **Paste**

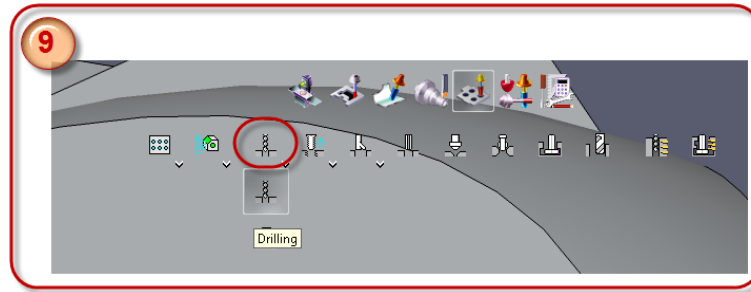
Double click the **Operation** that was just created. Select the **Tool Change** tab and choose the **D3 tool assembly**

Compute and select **OK**



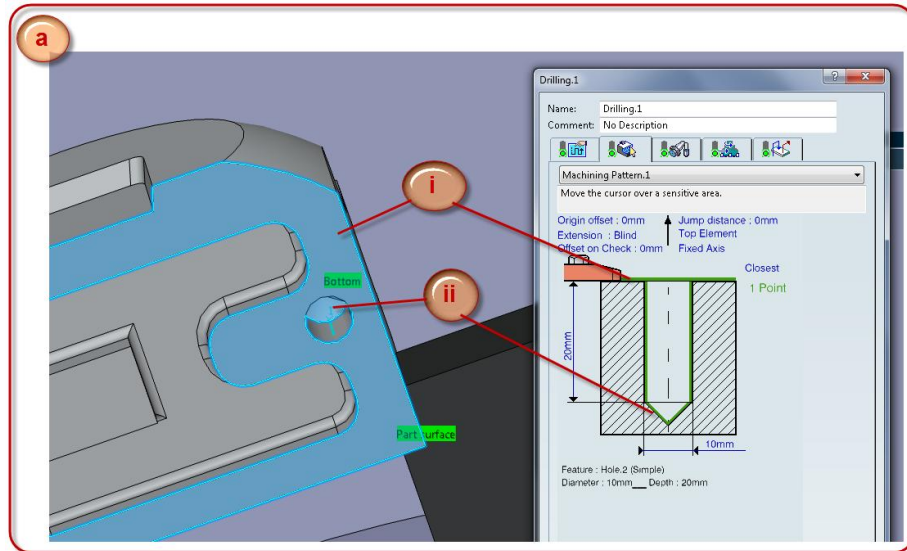
9. Drilling

Select **Axial Machining Operations** and then select **Drilling**



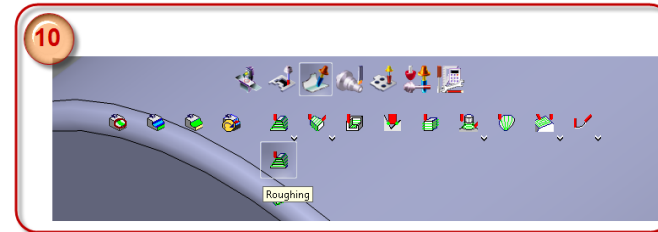
a. In the Geometry Parameter tab

- i. Select the **top** as **Part surface**
- ii. Select the **bottom** by clicking on the **bottom face** of the hole



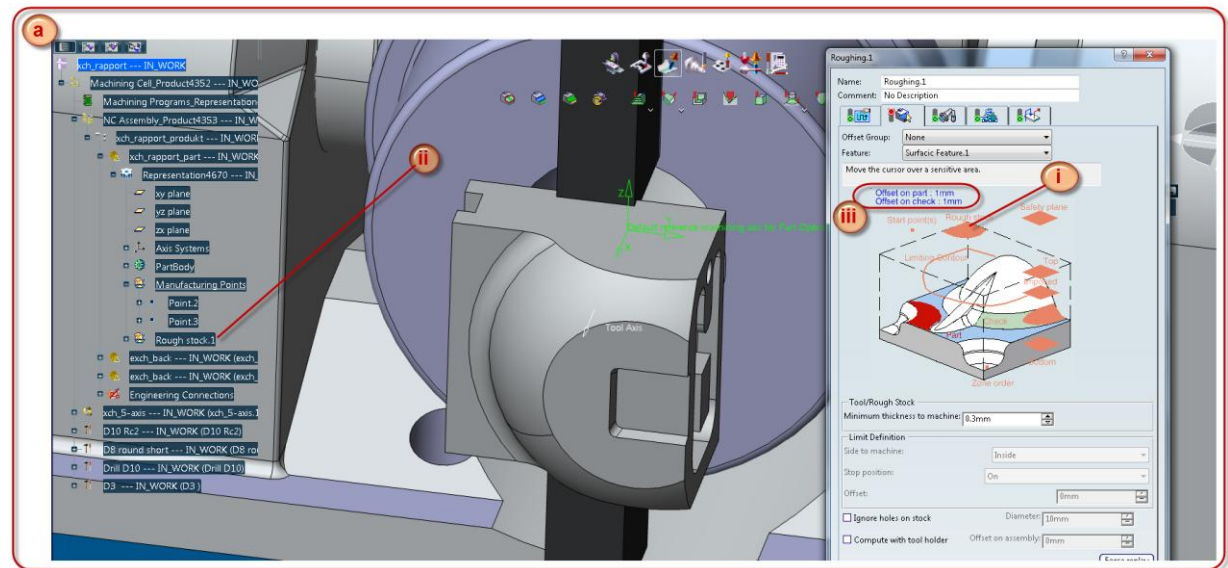
10. Roughing

Click on **Surface Machining Operations** and select **Roughing**



a. Roughing operation window Geometry strategy

- i. Select rough stock
- ii. Select the rough stock
From the PPR tree
- iii. Edit the **offset**
parameter on **part** and
check: 1 mm



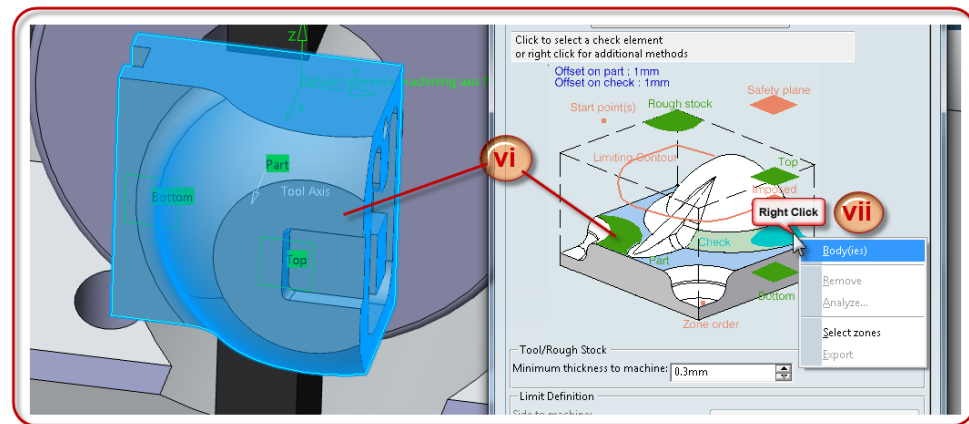
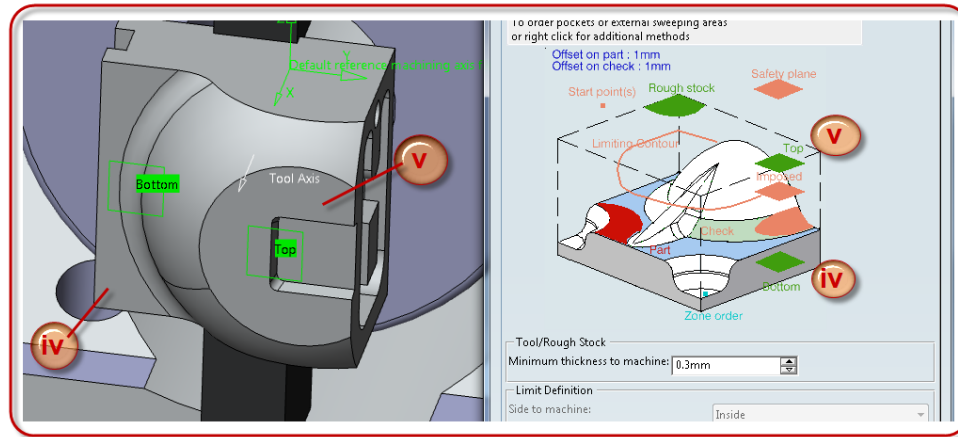
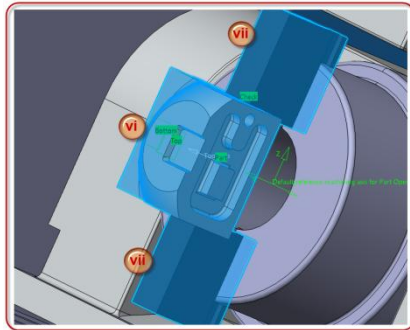
iv. Select **Bottom**

v. Select **Top**

vi. Select **Part**

When you select the part, the whole Feature is marked. With the other Selections as Top and Bottom the Desired geometry is recognized automatically

vii. Right click on **Check**, select **Body(ies)** and then select the **two fixtures** as check element



b. In the Strategy Parameters tab

i. Machining mode: By Area

Tool path style: Helical

You can see the explanation of the tool path style on the picture

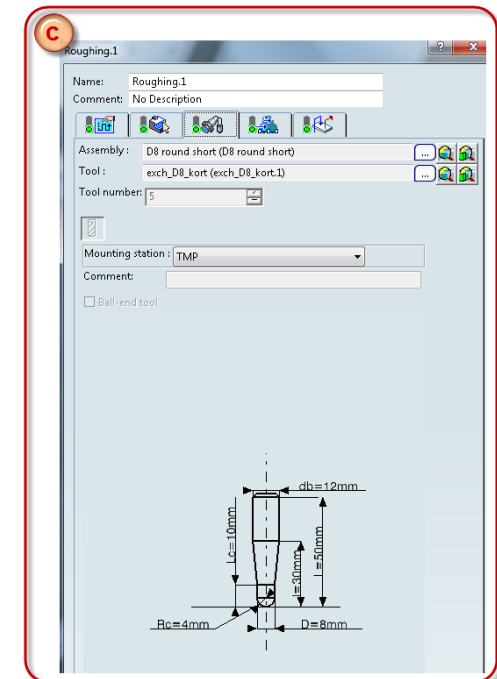
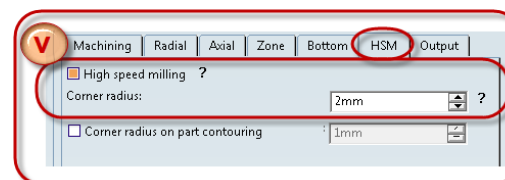
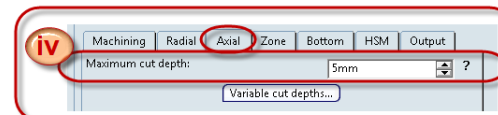
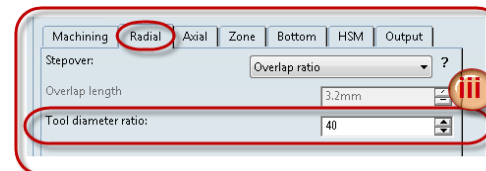
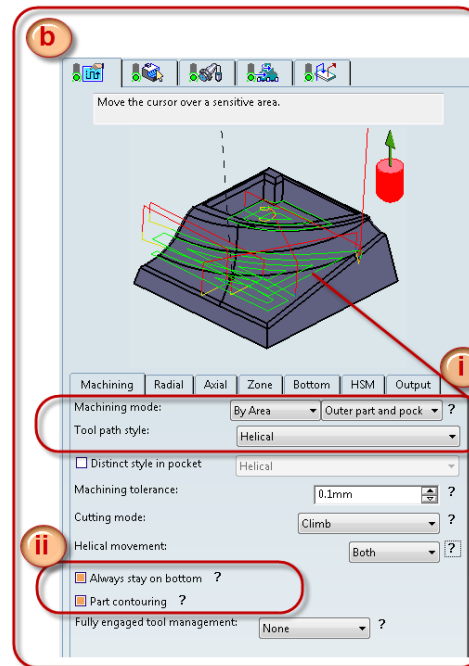
ii. Mark Always stay on bottom and Part contouring

**iii. In Radial strategy tab
Stepover: Overlap ratio
Tool diameter ratio: 40**

**iv. In Axial strategy tab
Maximum cut depth: 5 mm**

**v. In the HSM tab
mark High speed milling
Corner radius: 2 mm**

c. In the Tool Change tab
Select the **D8 Tool Assembly** from the **machining cell**



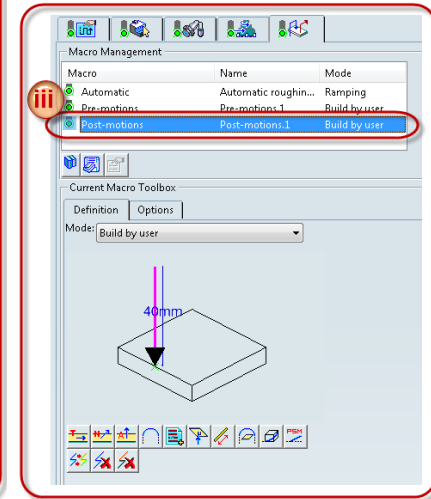
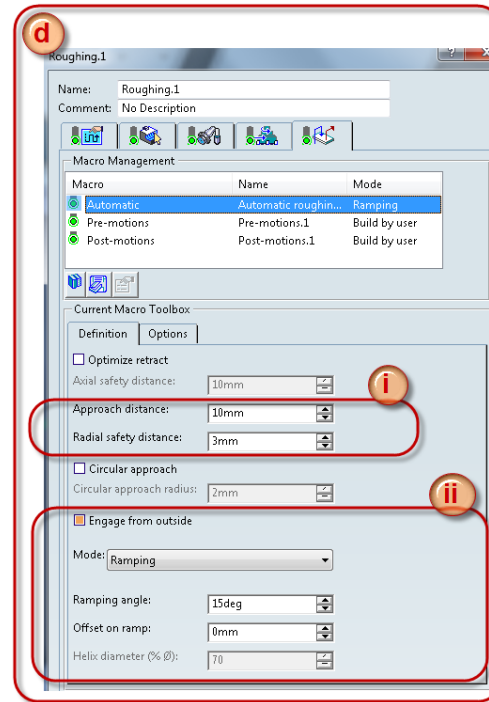
d. In the **Macro Parameter** tab

These are automatic

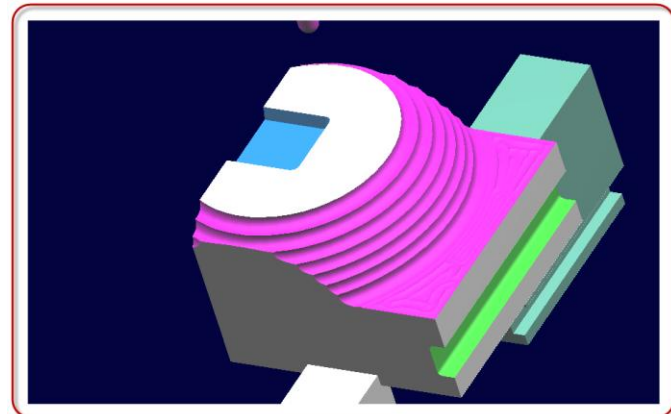
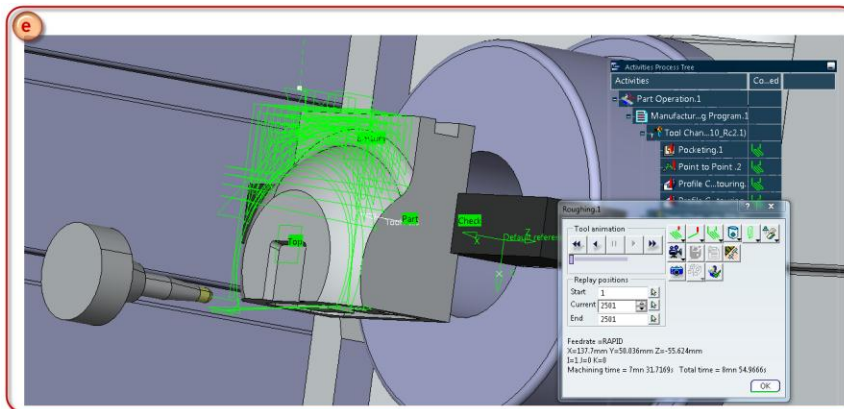
i. **Approach distance: 10 mm**
Radial safety distance: 3 mm

ii. **Mark Engage from outside**
Mode: Ramping
Ramping angle: 15 deg

iii. Select **Post-motions** and add
Axial: 40 mm

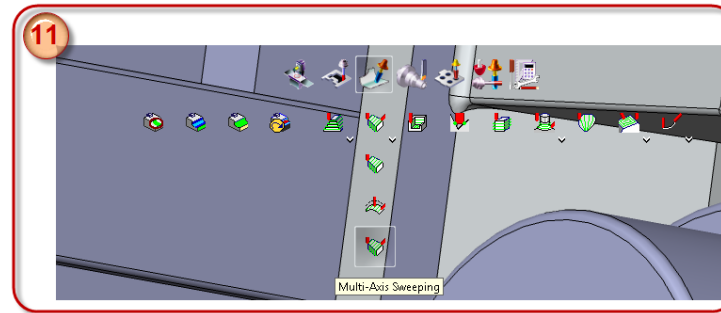


e. **Compute** and select **OK**



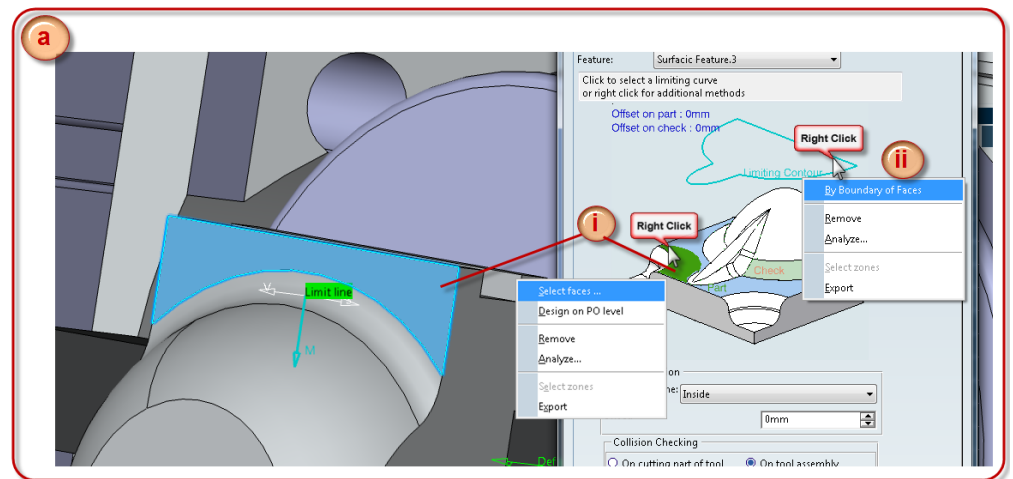
11. Multi-Axis Sweeping.1

Click on **Surface Machining Operations** and select **Multi-Axis Sweeping**



a. In the **Geometry Parameter** tab

- i. Right click on **Part** and select **Select faces..** then select the **surface** as you see in the picture
- ii. Right click on **Limiting Contour** and select **By Boundary of Faces** select the same surface as in **step i**.



b. In the Strategy Parameter tab

- i. Click on **Start Direction** and edit the parameters to

I: 0

J: 0

K: 1

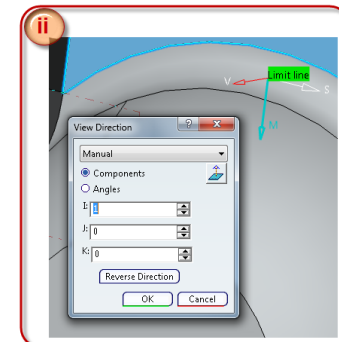
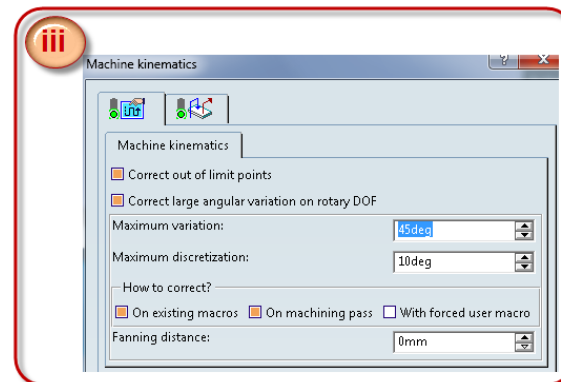
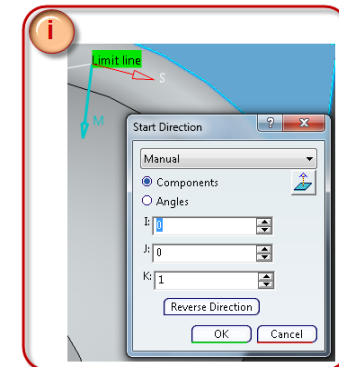
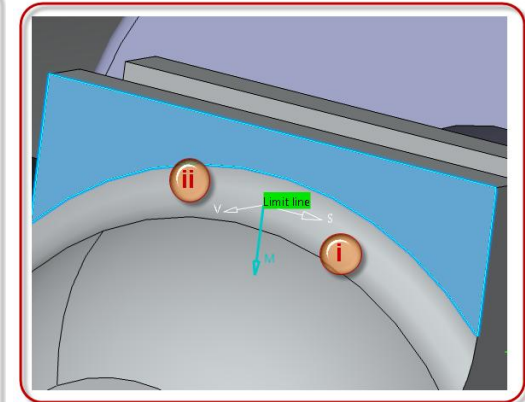
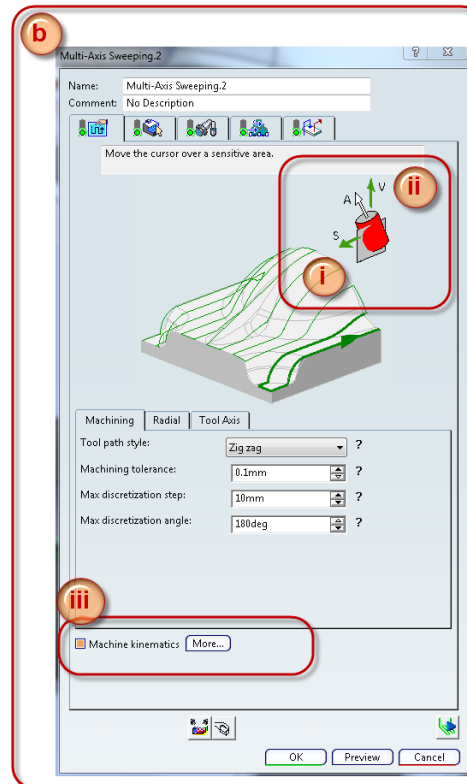
- ii. Click on **View Direction** and edit the parameters to

I: 1

J: 0

K: 0

- iii. Mark **Machine Kinematics** and click on **More..**
mark and select parameters as you see in the picture
press **OK**

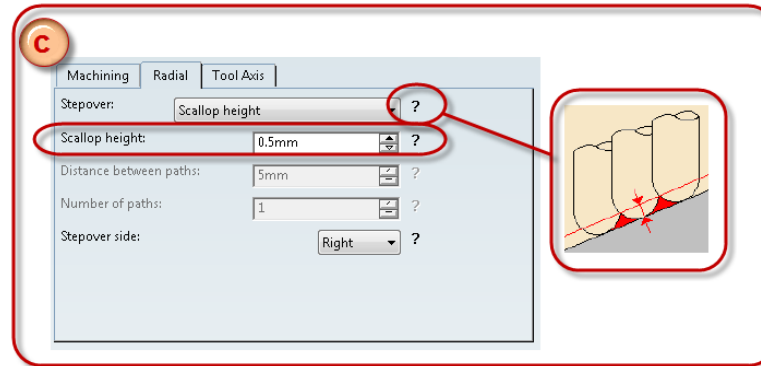


- c. Select the **Radial Strategy** tab

Stepover: Scallop height

Scallop height: 0.5 mm

If you would like a finer surface select a smaller Scallop height, 0.5 mm is just to make the simulation faster



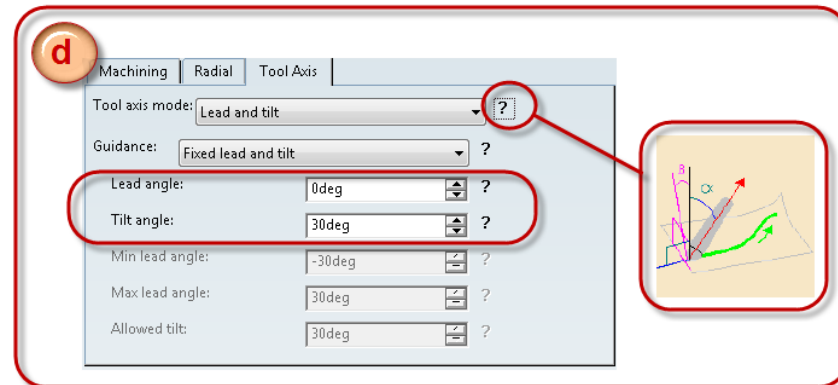
- d. Select the **Tool Axis** tab

Tool axis mode: Lead and tilt

Guidance: Fixed lead and tilt

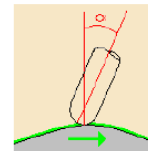
Lead angle: 0 deg

Tilt angle: 30 deg

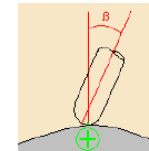


*Lead angle is the **angle along** the cutting Direction (a) in the picture. Tilt angle is the **angle normal** to the cutting Direction (B) in the picture.*

Lead angle



Tilt angle



e. Continue to **Macro Parameter** tab

i. Activate **Approach** and

Retract motion

Approach:

Circular: 10 mm

Axial: 50 mm

Retract:

Circular: 10 mm

Axial: 10 mm

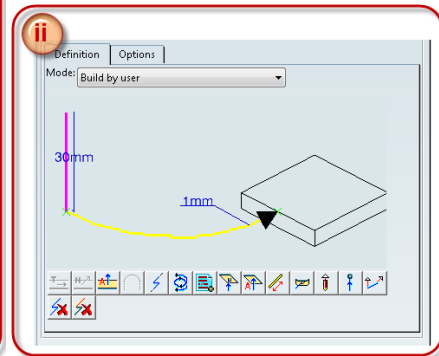
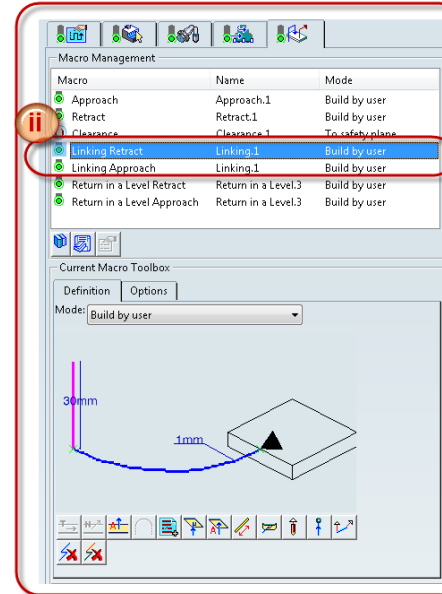
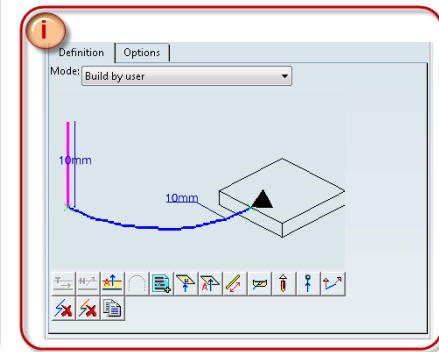
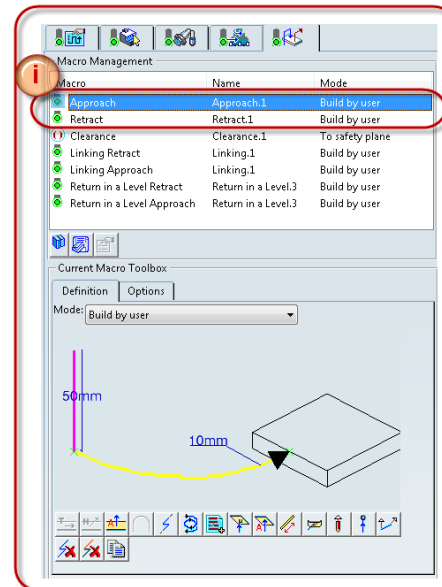
ii. Activate **Linking Retract**

and **Linking Approach**

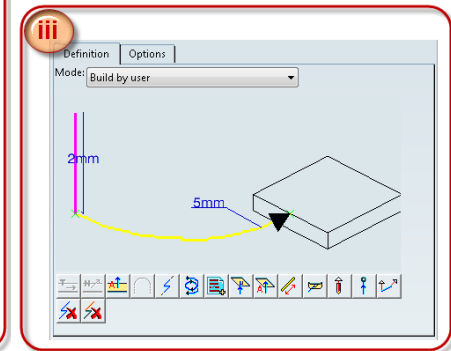
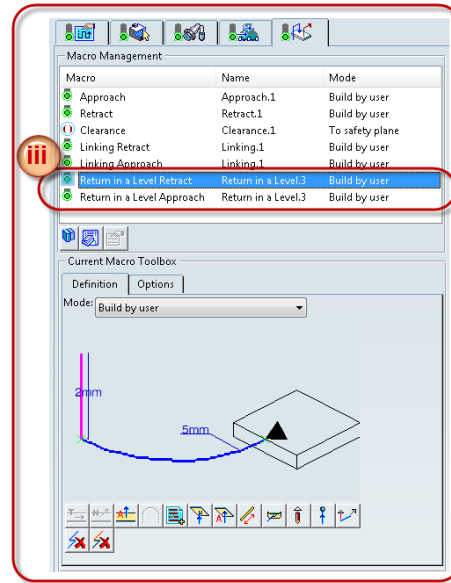
Both:

Circular: 1 mm

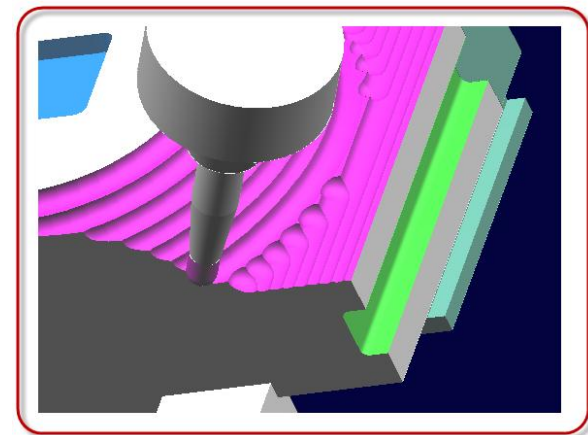
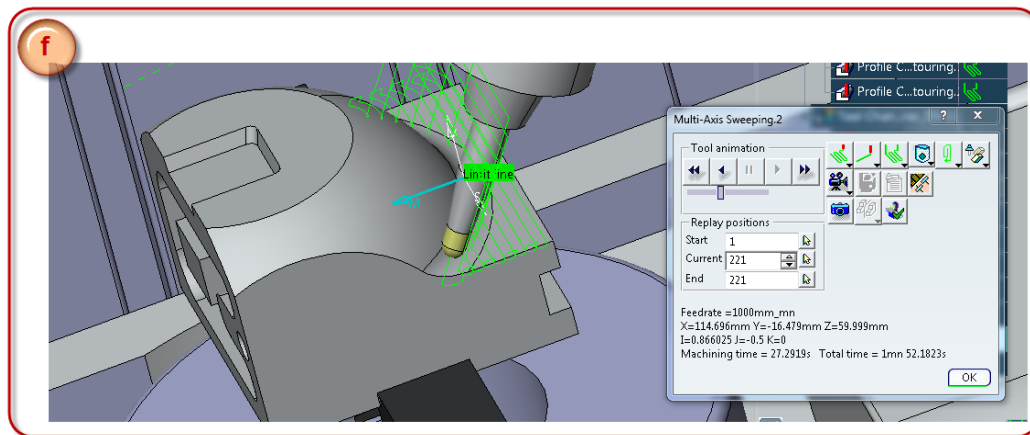
Axial: 30 mm



- iii. Activate **Return in a Level Retract** and **Return in a Level Approach**
Both:
Circular: 5 mm
Axial: 2 mm

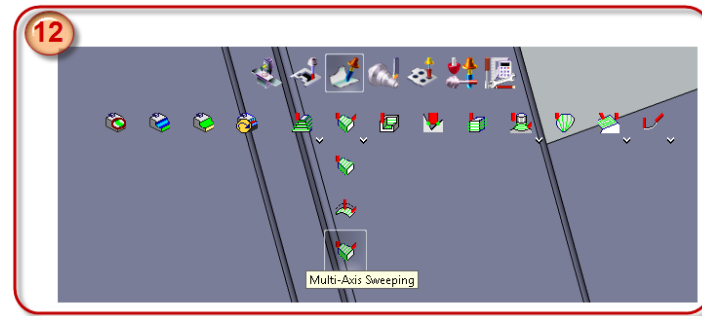


- f. **Compute** and select **OK**



12. Multi-Axis sweeping.2

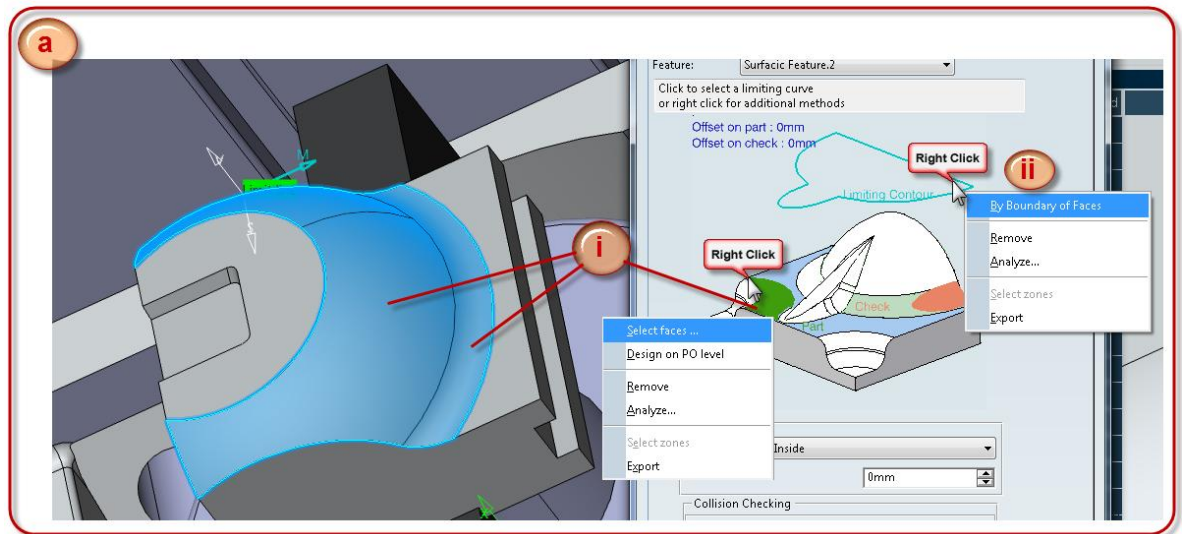
Click on **Surface Machining Operations** and select **Multi-Axis Sweeping**



a. In the Geometry parameters tab

i. As in the previous **step 11**, right click on **Part** and select **Select faces..** then select the **two surfaces** as illustrated in the **picture**

ii. Right click on the **Limiting Contour** and select **By Boundary of Faces**, then select the **same two surfaces** as in **step i**.



b. In the Strategy Parameter tab

- i. Click on **Start Direction** and edit the parameters to

I: 0

J: 0

K: 1

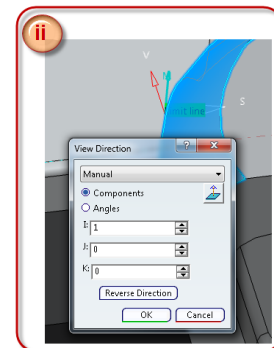
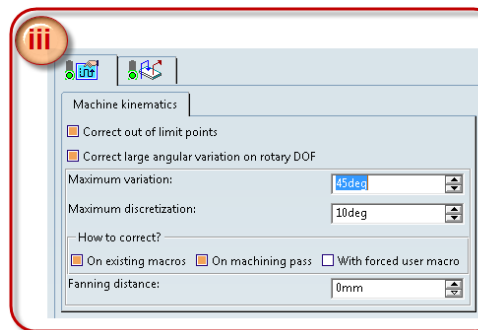
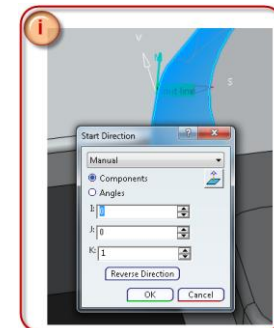
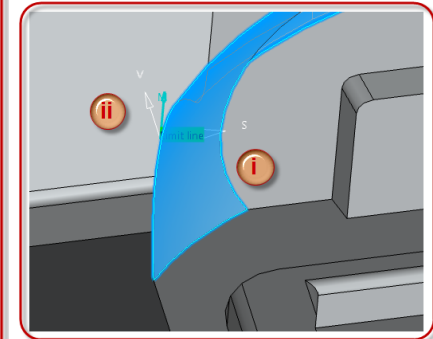
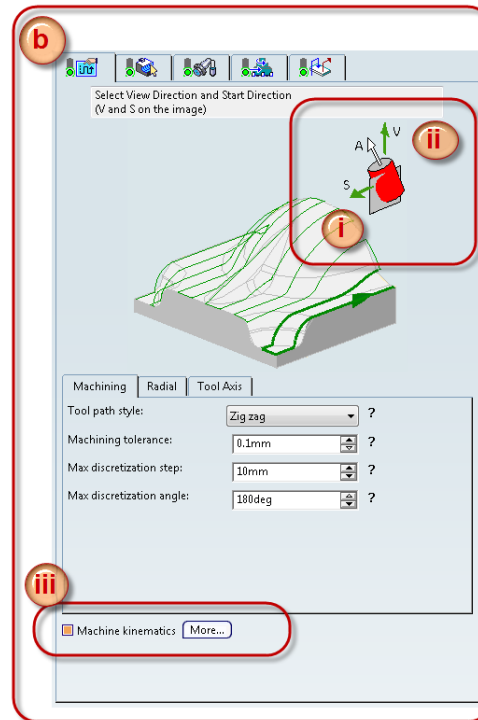
- ii. Click on **View Direction** and edit the parameters to

I: 1

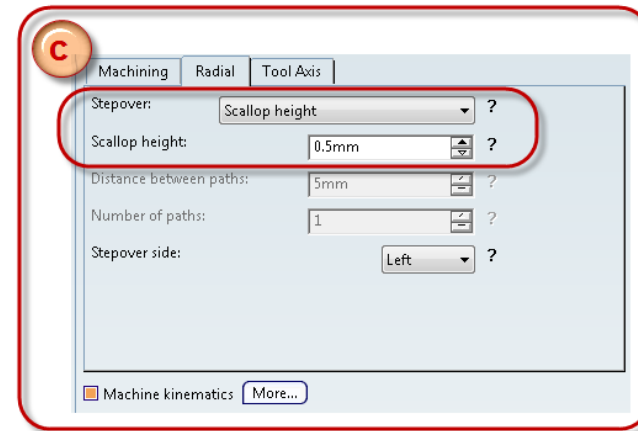
J: 0

K: 0

- iii. Mark **Machine Kinematics** and click on **More..**
mark and select parameters as you see in the picture
press **OK**

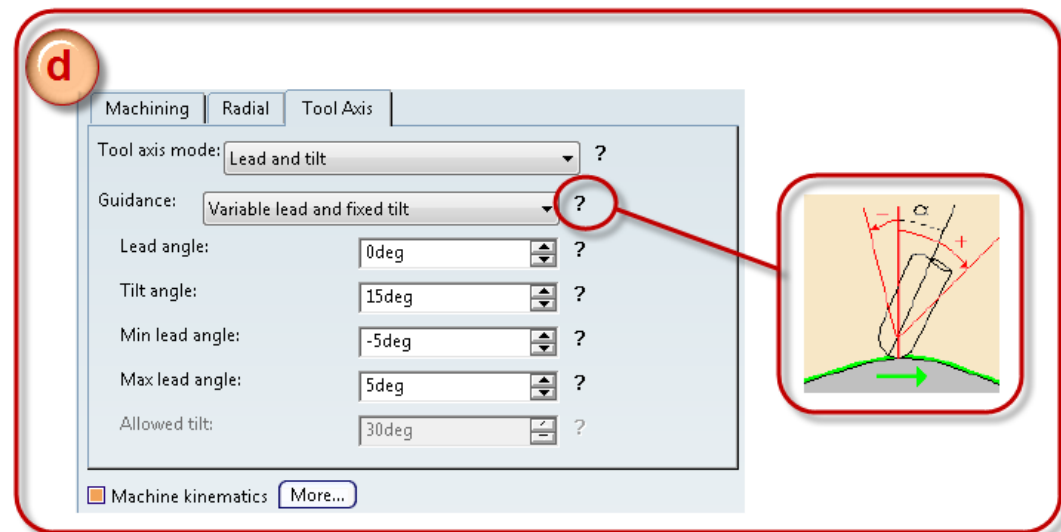


- c. Select the **Radial Strategy** tab
Stepover: Scallop height
Scallop height: 0.5 mm



- d. Select the **Tool Axis** tab
Guidance: Variable lead and fixed tilt
Lead angle: 0 deg
Tilt angle: 15 deg
Min lead angle: -5 deg
Max lead angle: 5 deg

The starting lead angle will be 0 deg but when needed it will automatically go between -5 and 5 degrees.



e. Continue to **Macro parameter** tab

i. Activate **Approach** and **Retract** motion

Approach:

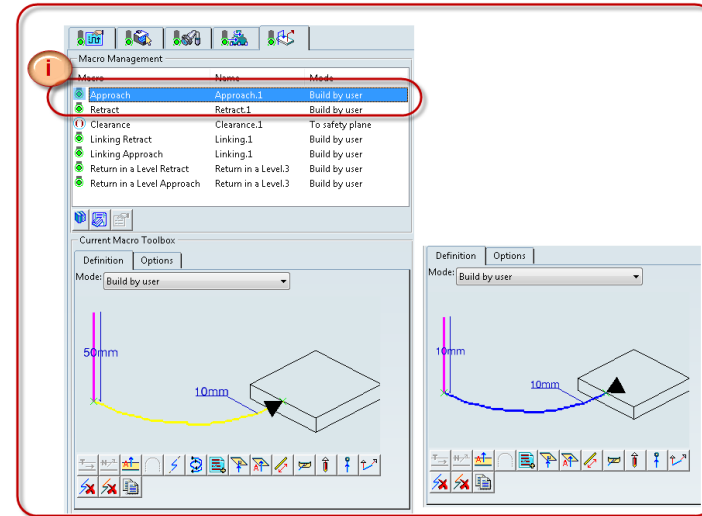
Circular: 10 mm

Axial: 50

Retract:

Circular: 10 mm

Axial: 10 mm

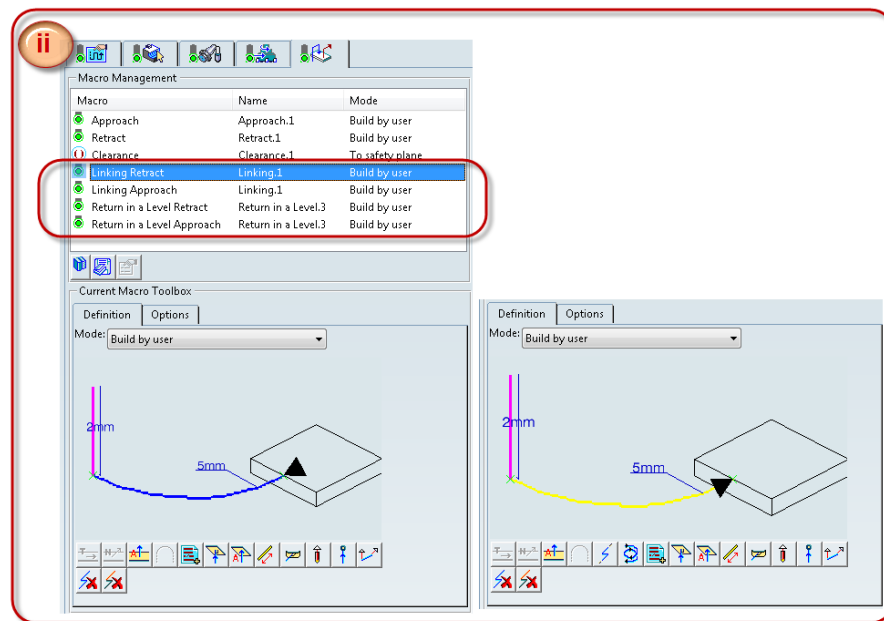


ii. Activate **Linking Retract**,
Linking Approach,
Return in a Level Retract and
Return in a Level Approach

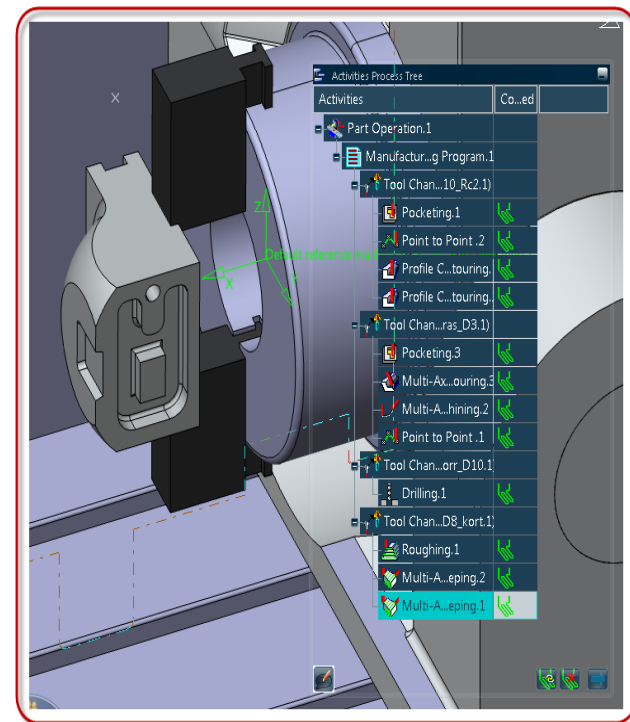
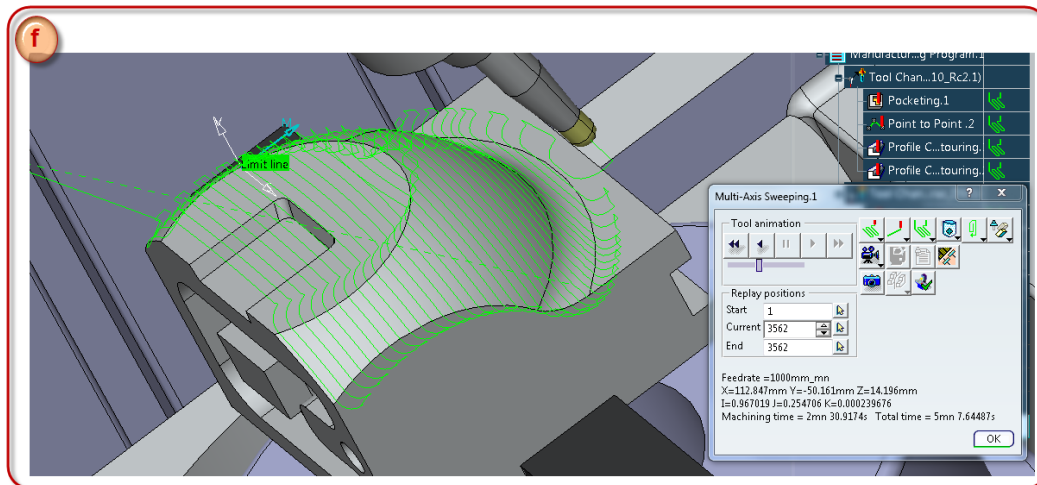
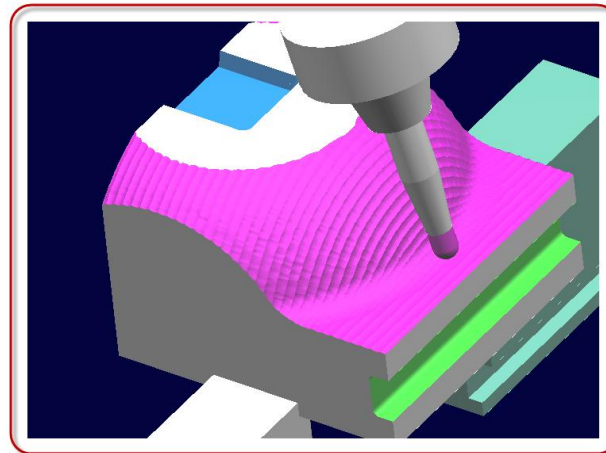
For all four

Circular: 5 mm

Axial: 2 mm



f. **Compute** and select **OK**

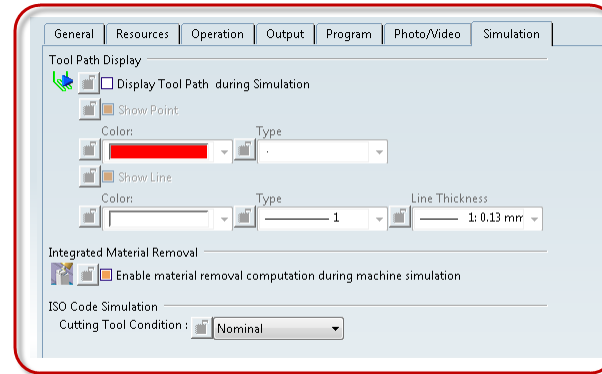


VERIFY AND GENERATE NC-CODE

1. Simulation

Select Tools > Options > Machining >

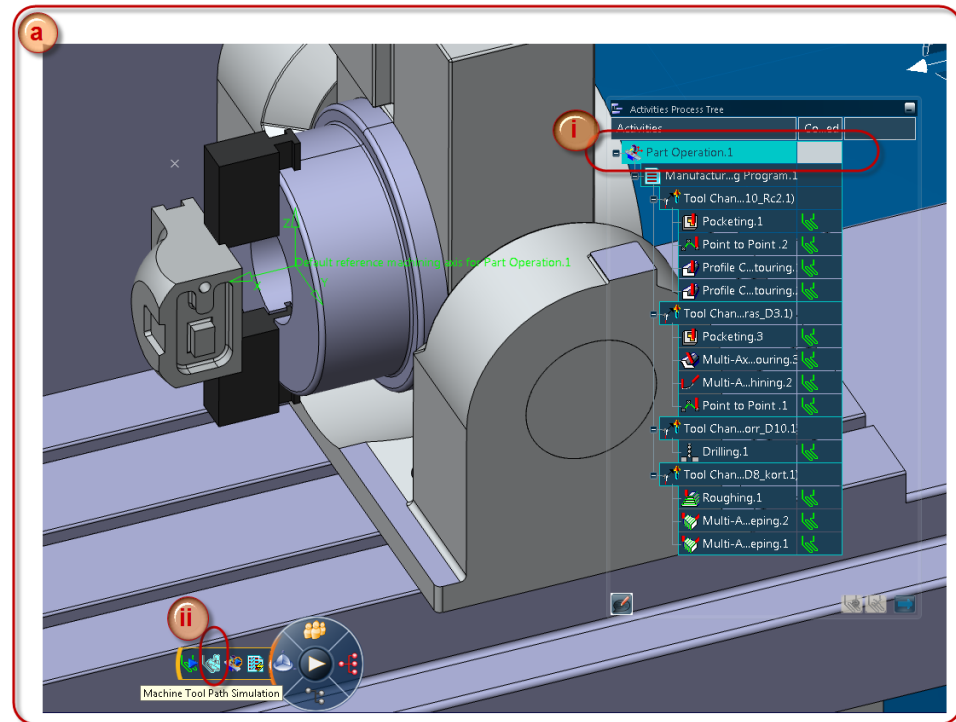
Simulation Tab > Mark **Enable material removal**



a. Machine Tool Path Simulation

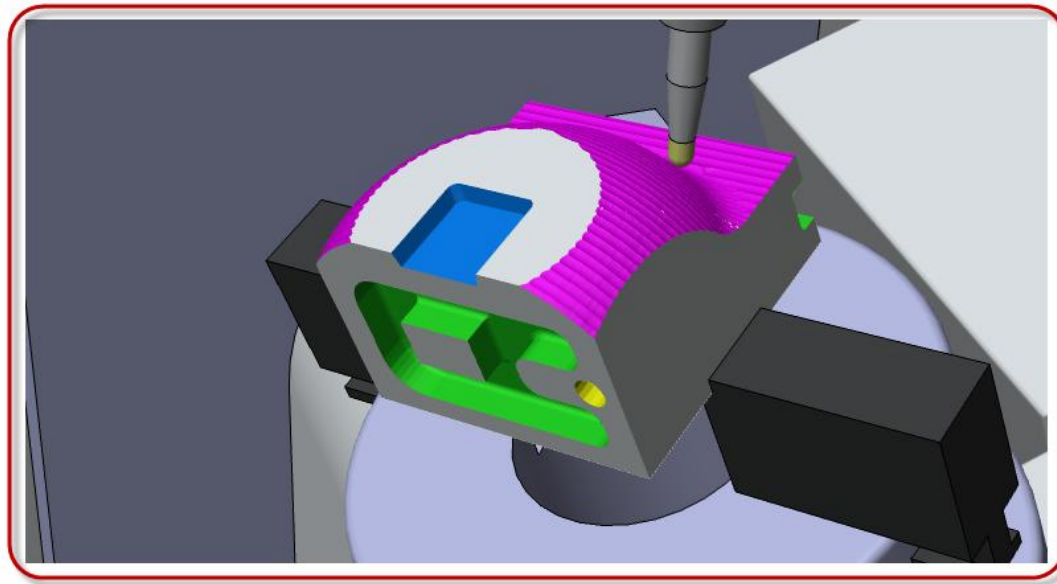
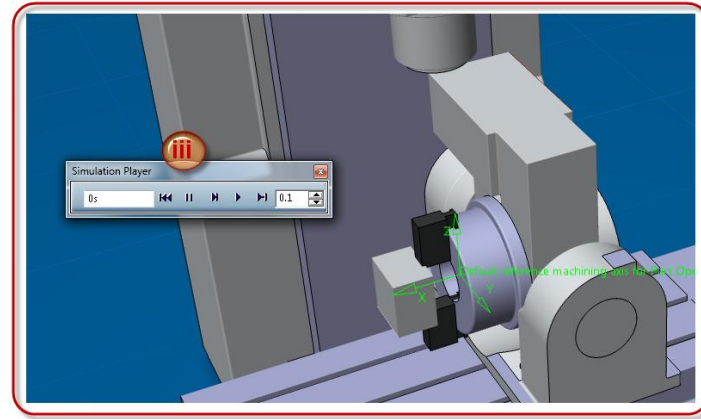
i. Mark **Part Operation.1** in the **Activities Process Tree**

ii. Select **Machine Tool Path Simulation** in the **Compass**



- iii. The **Simulation Player** includes **start, stop, time** and **speed** selection

Press play and see the simulation with material removal and machine movement

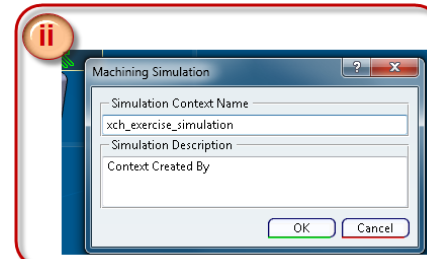


b. Live Simulation

- i. Select the **Launch Live Simulation of NC Machine** button on the compass

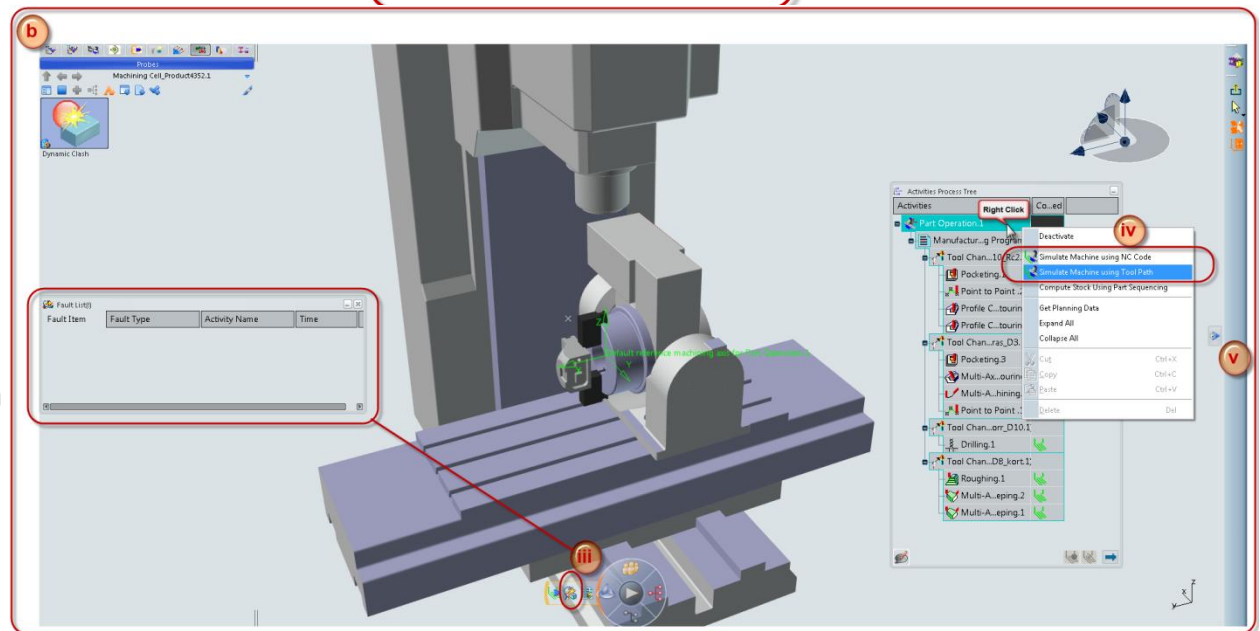


- ii. Name the **Machining Simulation**



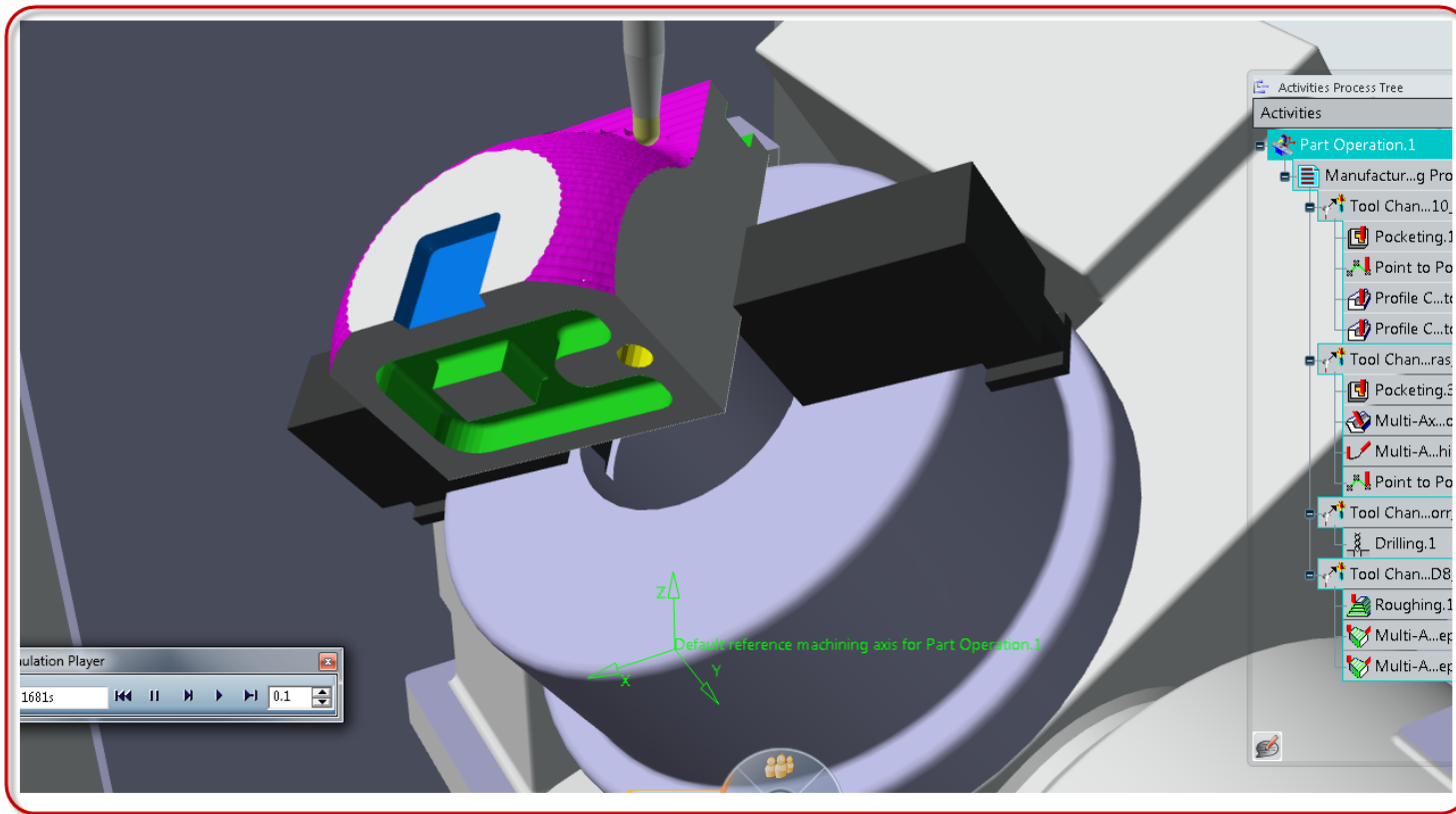
- iii. **Display fault list**

- iv. Right click to choose if you want to simulate using **NC Code or Tool Path**
- Select by **Tool Path**



- v. **Video Simulation options**

Press play and see the Live Simulation



2. Generate NC Code

- a. Click to access the **Generate NC code** window
- b. Select **Execute** to generate the **NC code**
- c. **The NC code is placed in the PPR tree**
Right click on it and select
Export from NC Files Container
Select where to save the file
- d. View the **NC code**

