

KORIŠTENJE I PREDNOSTI CAD/CAM TEHNOLOGIJE U INDUSTRIJI

USAGE AND ADVANTAGES OF CAD/CAM TECHNOLOGY FOR INDUSTRY

Robert Jolić, Đuro Kukec, Tomislav Pavlic

Stručni članak

Sažetak: Ovaj rad opisuje karakteristike CNC tehnologije, njeno korištenje i prednosti korištenja u proizvodnji. Opisana je ideja, izrada 3D modela u CAD programskome alatu, zatim su u CAM programskome alatu definirani svi parametri glodanja i na kraju je sve glodano na stroju. Prikazano je koliko se brže ovom tehnologijom proizvod može koristiti i proizvoditi, u odnosu na način da se sve radi ručno. Sve je modelirano u programskome alatu SolidWorks, parametri glodanja su definirani u programskome alatu SolidCAM, i na kraju je G-kod, dobiven iz SolidCAM-a, prilagođen stroju koristeći postprocesor nazvan Mach3.

Ključne riječi: glodanje, računalom podržan dizajn, računalom podržana proizvodnja, SolidWorks, SolidCam

Professional paper

Abstract: This paper briefly describes the steps of production with the help of CNC technology, to show its usage and advantages in industry. It starts from an idea which is then brought to life in a 3D model designed in the CAD software. It is followed by defining all the parameters for CNC milling using the CAM software. Finally, it is brought to the machine to be milled. In the end, the result is a product that can be used and produced more quickly than if everything had to be done manually. Everything is modelled in SolidWorks, the parameters are defined for milling in SolidCAM, and a G-code got from SolidCAM is loaded using a post processor called Mach3.

Key words: CAD, CAM, SolidWorks, SolidCam, milling

1. INTRODUCTION

Throughout history, before all the vast amount of technology that is used today, people had to do most things manually. Material processing used to take a lot of skill and a profuse amount of time and effort to achieve a desired finished product. Fortunately, the advances in technology that were made make material processing much simpler and less time consuming. Nowadays, CAD/CAM system is used for designing a product and for controlling manufacturing processes. CAD/CAM is an acronym for computer-aided design/computer-aided manufacturing [1]. Naturally, the CAD component comes first. A model has to be designed an imagined/drawn model taken and transferred into modelling software. Some popular software or most commonly used 3D modelling software is for example Catia, AutoCAD, Autodesk and SolidWorks [2] (SolidWorks is used as the example in this article).

After completing the design in CAD software, it can be moved onto the next step, which is setting all the parameters in the CAM software for the unfinished product. Not only is it controlled how the piece is to be machined and processed, but also a control of how the actual CNC machine will work to fulfil the desires for the product is wanted. To clarify – there is a difference between what is wished of the machine to do to the piece being machined and the specific parameters that are

wanted of the machine to have so it can work properly. All of this can be regulated in the CAM software. Some popular CAM software include CATIA, Mastercam, and SolidCAM, which was used in the article [3]. In the example a 2.5D milling/machining or a two-and-a-half-axis mill was used.

2. 3D MODELING THE PIECE IN THE CAD SOFTWARE SOLIDWORKS

The "product" that is produced is more or less a randomly constructed piece. It does not serve much purpose for us in the real world other than to show how well the CAD/CAM software and machine work (with the assumption that the model and the parameters for the piece and machine are set perfectly). As mentioned in the introduction, the CAD software used- to model the piece is SolidWorks. Before even the modelling of the piece can begin few things have to be taken into account, one of the main things being what material is being used and its dimensions. The size of the material available for milling has a dimension of 100 mm x 60 mm with a height of 18 mm. Medium-density fiberboard (MDF) blocks is being used.

Everything is done step-by-step and first comes a 2D drawing which is then extruded into a 3D object that can be manipulated until getting the desired piece.



Figure 1 Sketch toolbar

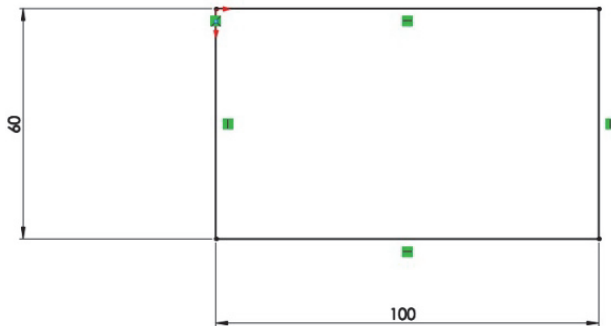


Figure 2 2D drawing of MDF block

After drawing a base sketch, it is turned into a 3D object by using the Boss-Extrude tool. Basically, the 2D sketch is given a third dimension, which is the thickness of the MDF (18mm).

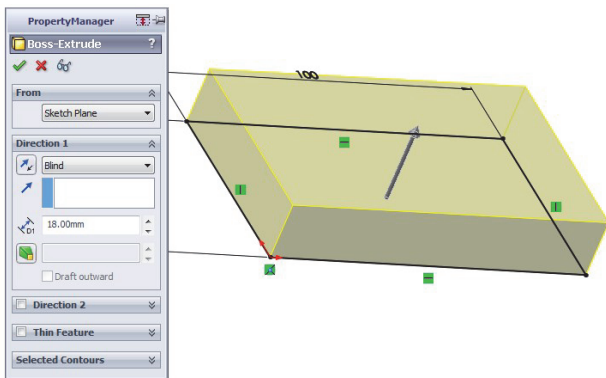


Figure 3 Giving the sketch a third dimension (Boss-Extrude)

Now there is a 3D model of the MDF block and material can be cut away until having a finished piece that will be ready for production (milling).

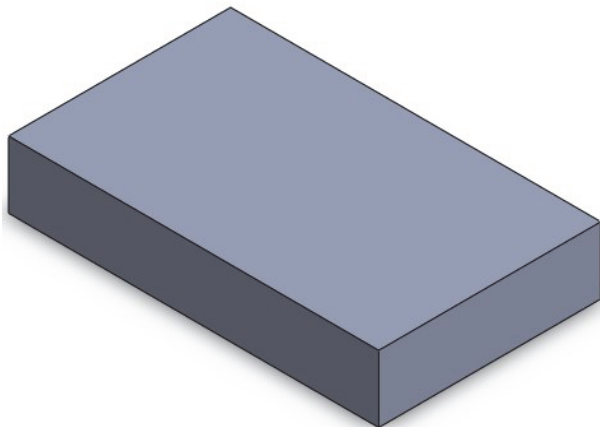


Figure 4 3D model of an MDF block

Next, the block is set to the Normal To view from the toolbars.

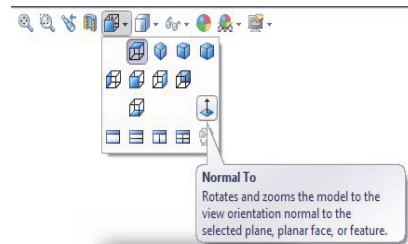


Figure 5 Setting the object Normal To

This gives a view of the front of the face of the object.

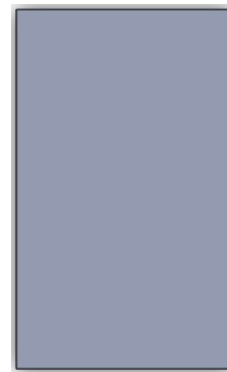


Figure 6 Normal To view of the MDF block

Having the block's front face parallel to us, that face can be selected and a 2D sketch can be drawn on it. The piece is modelled the same way it is desired of the CNC machine to machine the block. The desired shape is drawn and the rest of the material removed, which in SolidWorks would be done with the help of the Extruded Cut tool.

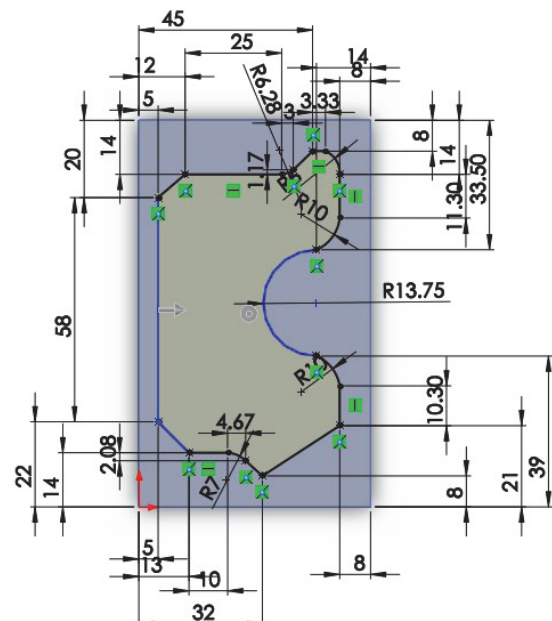


Figure 7 Sketch on the front face of the block (before cutting)

Now that the sketch is complete, the excess material that is not wanted/needed can be cut away. However, the excess material is all the material on the outside of the line. Therefore, to cut away that side, the Extruded Cut tool is chosen, typed in how deep to cut, and it has to be made sure **Flip side to cut** is checked in.

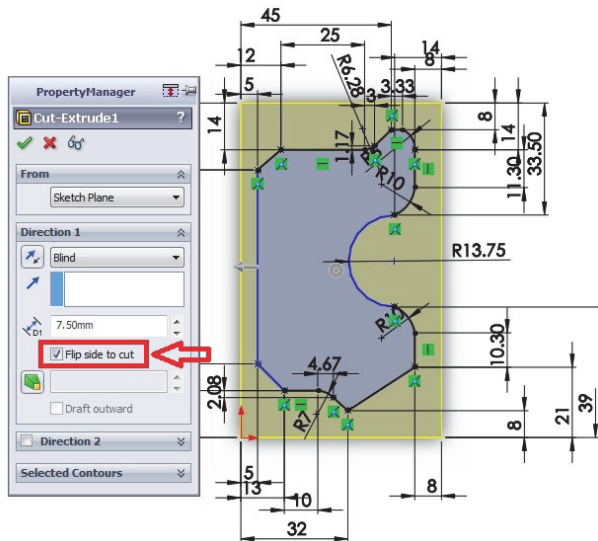


Figure 8 Cut-Extrude with Flip side to cut checked in

When the excess material is cut away, it can be moved onto the next part of the object and some oddly shaped pockets can be added. One thing to keep in mind while modelling the piece is to never make an arc less than half the radius of the end mill of the CNC machine. If this mistake is made, problems may appear when simulating the piece in the CAM software.

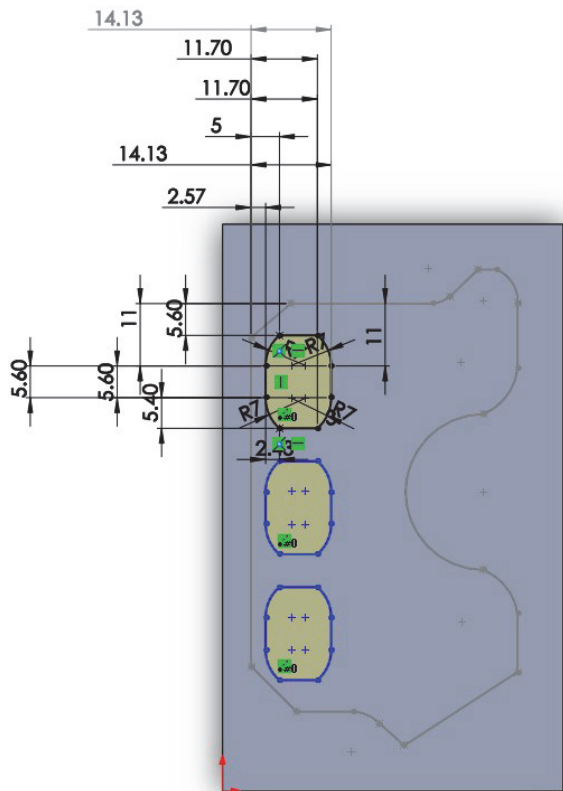


Figure 9 3 pockets to be cut-extruded

One shortcut for duplicating a sketch x-amount of times down or across a linear path is to use the Linear Sketch Pattern tool (also located in the sketch toolbar, refer to (Figure 1)).

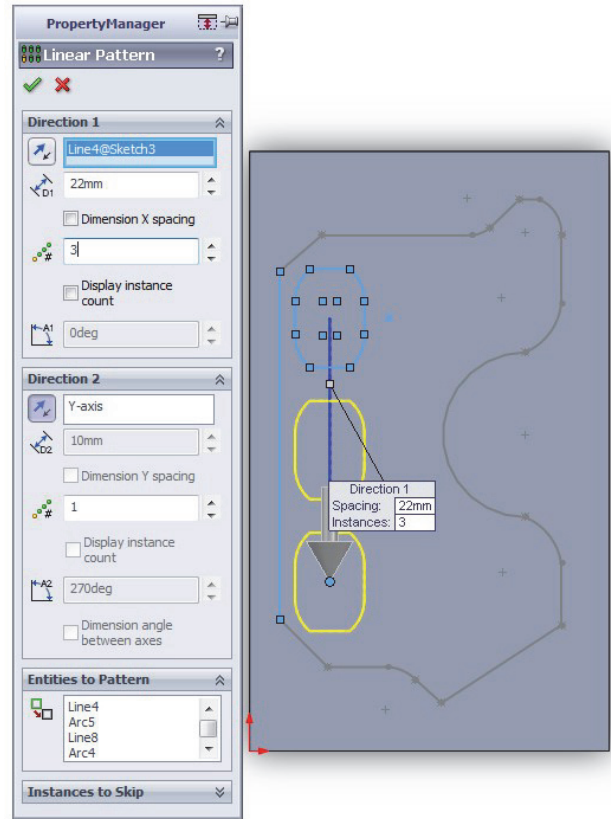


Figure 10 Linear Sketch Pattern tool for duplicating sketches down or across a linear path

Now the Extruded Cut tool is used again (refer to (Figure 8)) however this time the Flip side to cut option is unchecked.

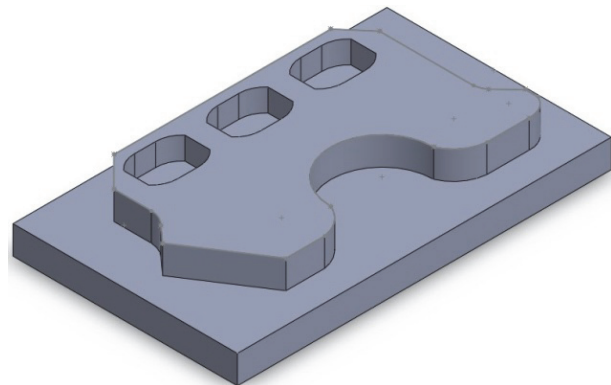


Figure 11 Cut-Extruded pockets

Beside the three pockets, another pocket is going to be added. However, it will be in the shape of an arc slot also called a *Three-Point Arc Slot*. One important thing to do here is to make the arc slot *concentric* to the existing arc. By definition, *concentric* here means that the two arcs will share the same center – this will keep the arc slot an equal distance away from the outer edge (Fig. 11).

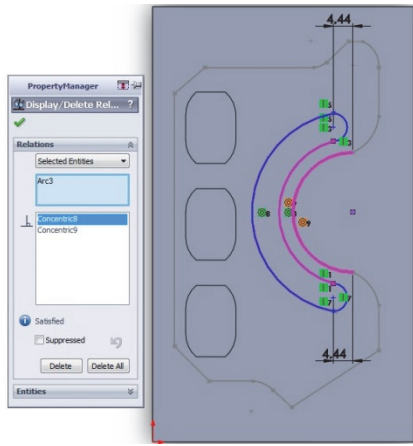


Figure 12 Drawn arc slot *concentric* to outer arc

The slot is extruded, again having the **Flip side to cut** option unchecked (because everything within the sketch lines is being extruded).

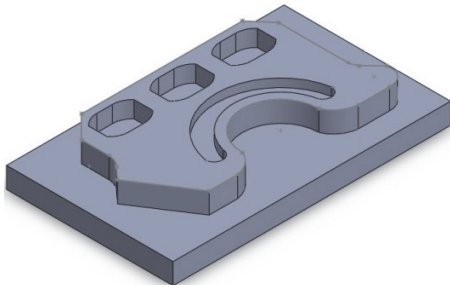


Figure 13 *Cut-Extruded* arc slot

The final step to the model is to add five through holes that are placed one at every corner of the piece and one inside the open space underneath the arc. Here it has to be taken into account the size of the end mill again; the holes can be no smaller than the end mill, either the same diameter or larger than the end mill.

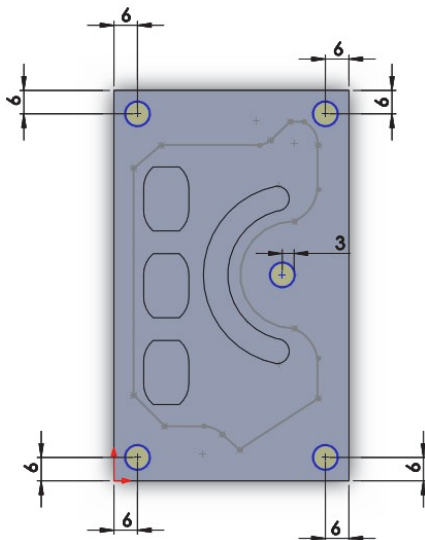


Figure 14 Sketching the holes

One final *Extruded Cut* and the finished 3D model is ready for the CAM software.

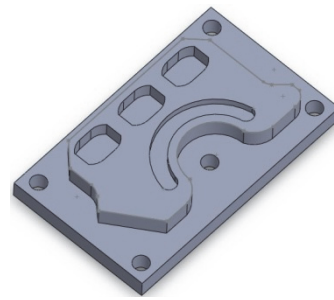


Figure 15 Finished model

3. PREPARING THE PIECE AND DEFINING THE PARAMETERS IN SOLIDCAM

When the model is opened in SolidCam, few starting options are chosen.

First, the CNC machine is chosen, which in this case is Mach3. Next, the coordinate system of the model is defined. The x, y and z axes of the model were chosen. The x axis is chosen along the shorter side of the model (60 mm), the y along the longer side (100 mm), and the z along the height of the model (18 mm)

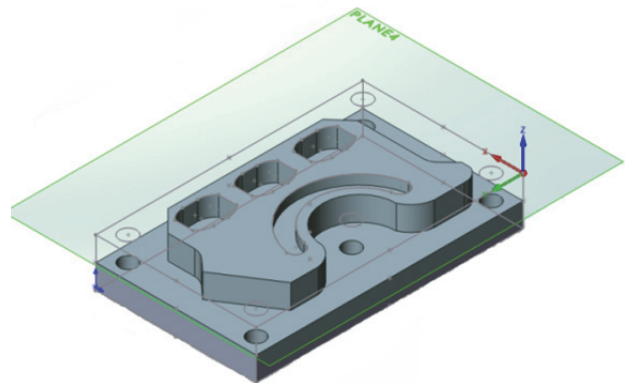


Figure 16 Choosing the coordinate system

Use the right click on 'Operations' and go to Add Operations >> Milling.

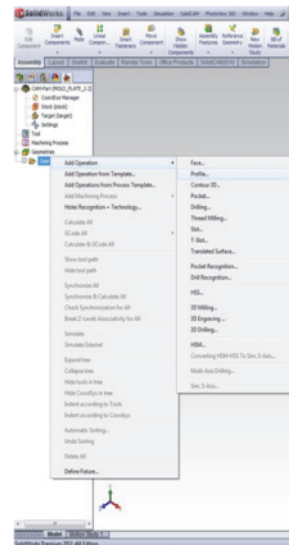


Figure 17 Adding a new operation

Under the *Geometry Edit* option select all the contours and edges of the object. Everything within or outside those contours and edges will be milled or altered somehow. These are called chains. One edge is selected and then chose the 'Auto-Constant z' option and the whole chain/ edge is automatically selected.

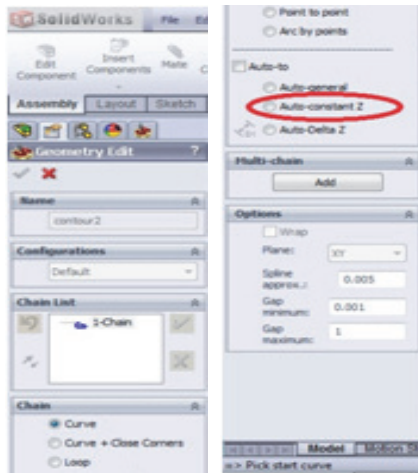


Figure 18 Geometry Edit option

Next the tool is selected, which is called the End Mill. An End Mill with a diameter of 6 mm was used with the following parameters:

- TL: 53 mm
- OHL: 20 mm
- SL: 19 mm
- H Length: 100
- Number of Flutes: 2

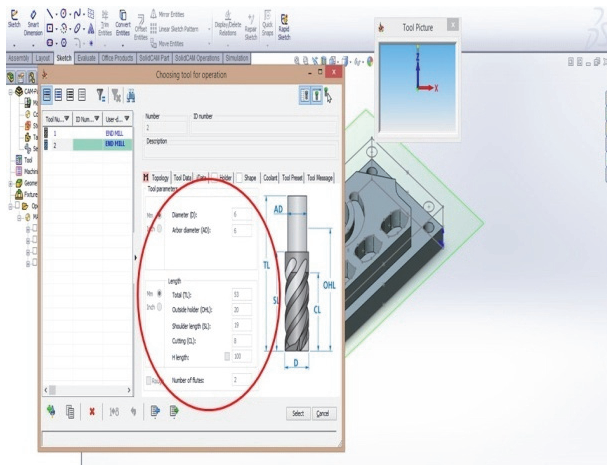


Figure 19 Selecting the end mill

The next important step is the selection of the feed rate and the spinning rate.

Feed rate: the rate at which the cutting tool and the work piece move in relation to one another.

Spin rate: spindle speed in rpm (revolutions per minute).

For MDF (Medium-density fiberboard) and plastic the following parameters were used:

- Feed XY: 1000 mm/tooth
- Feed Z: 330 mm/tooth
- Spin rate: 29,000 rpm

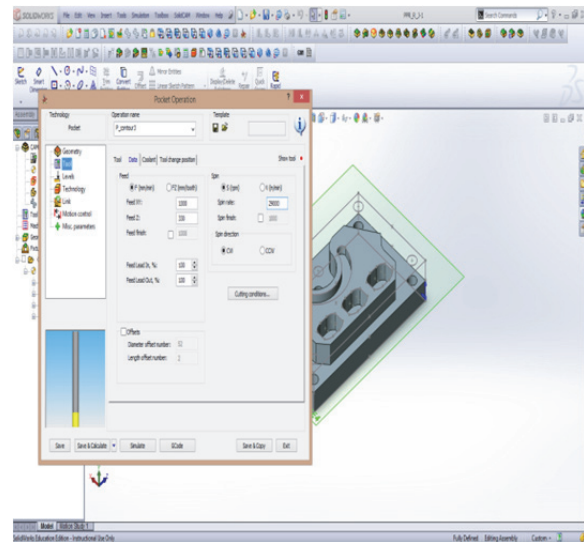


Figure 20 Tool speed parameters

Next the pocket operation is performed. The pocket operation is used even on all the outer edges because there is so much material on the piece to be removed. In the options the upper level is the height at which the End Mill will enter the material. The pocket depth is the depth at which the End Mill stops going into the material. A contour is selected and the required parameters are filled in.

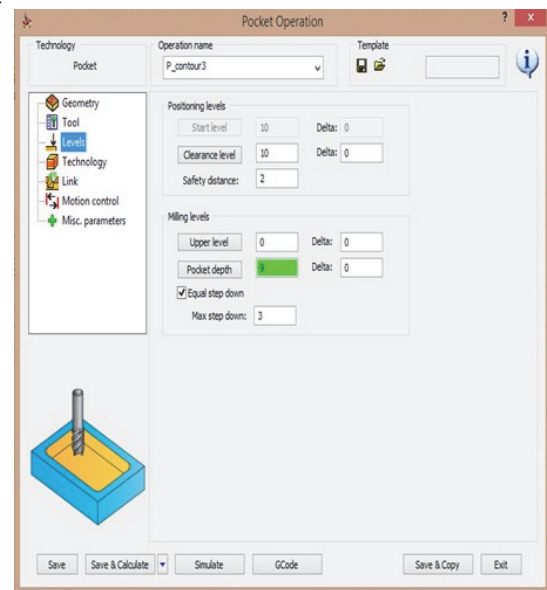


Figure 21 Pocket operation

Also what should be taken from pocket operations (*Figure 21*) is the Max Step Down. With this option it is basically told to the CNC machine how many passes to make until the desired shape based on the pocket depth is milled out. So if my pocket depth is 9 mm I would have the CNC make 3 passes each at a depth of 3 mm. This makes it easier on the End Mill and the CNC, however it increases the work time.

The milling will start from the outside of the block, this way only 50% of the End Mill would enter into the material. The direction of the milling was also chosen, it was chosen to climb, which would be a clock-wise direction. This way all the milled material would be thrown away from the block. After setting up all the

desired options the next step was to simulate the milling job in SolidCam so it can be verified if everything checks out. This piece would take approximately 8:40 minutes to be milled on the CNC.

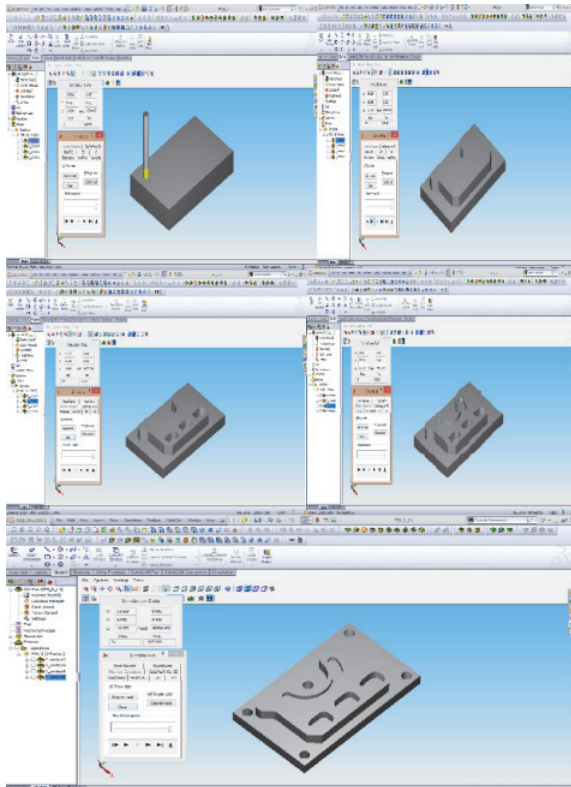


Figure 22 Simulating the milling job in SolidCam

4. MACH3 AND MILLING THE MDF BLOCK INTO THE 3D MODEL

Now the milling job can begin. Opening *Mach3* [4] and loading the G-Code is the first step.

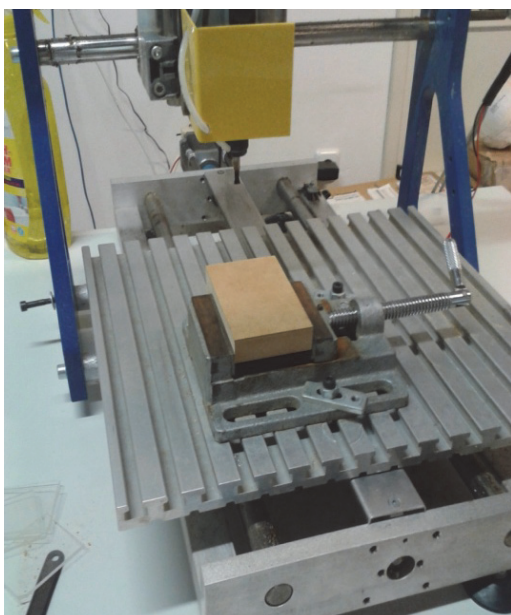


Figure 23 Block in place prepared to find the starting point (the 0 point) [5]

Next it has to be clicked on the reset button to make sure Jog is on. The block is manually set and the center of the End Mill is positioned at the zero point on the block. In *Mach3*, the x, y and z are set to zero when at the starting position. When everything is in position and ready to start, milling is started by selecting the Cycle Start button.

Mach3 is opened and the G-code is loaded. After loading the G-Code the reset button should be clicked on and it needs to be made sure Jog is on. The block is manually set and the End Mill is centered at the zero point on the block. In *Mach3*, the x, y and z are set to zero at the starting position. When everything is in position and ready to start, the Cycle Start button can be clicked on.

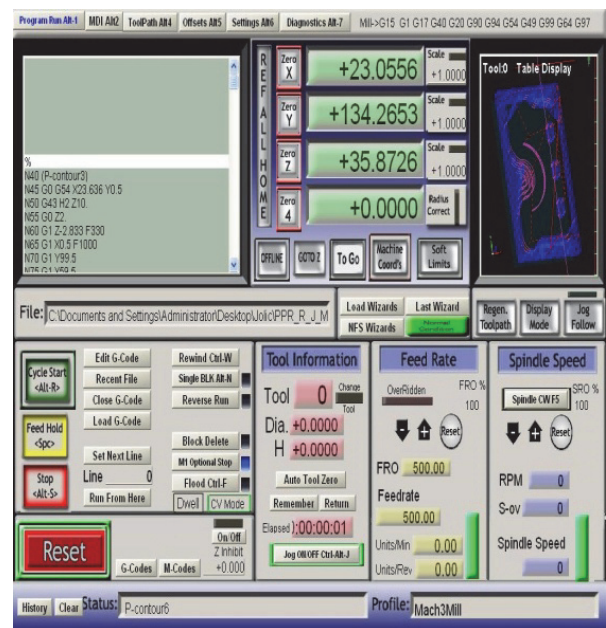


Figure 24 *Mach3* in preparation for milling

Waiting period lasts until the CNC machine completes the milling job and the finished product can be removed.



Figure 25 The finished piece before clean up

It doesn't look to impressive at first, however the excess has to be removed and the piece has to be cleaned off a bit. This type of quality comes when finished

because of the material that is used; the MDF. This does not cause a problem because it is easily cleaned. (Figure 26) shows the final product.



Figure 26 Completed piece, touched up and cleaned off

Kontakt autora:

Robert Jolić

Visoka tehnička škola u Bjelovaru
Trg E. Kvaternika 4, 43000 Bjelovar
robert.jolic@hotmail.com

Đuro Kukec, prof.

Sveučilište Sjever
Sveučilišni centar Varaždin
104. brigade 3, 42000 Varaždin
djuro.kukec@velv.hr

Pavlic Tomislav, mag.ing.mech.

Visoka tehnička škola u Bjelovaru
Trg E. Kvaternika 4, 43000 Bjelovar
tpavlic@vtsbj.hr

5. CONCLUSION

The goal was to test the CAD/CAM system (along with the CNC machine) and see what results could be achieved with an intermediate level of skill. And the result speaks for itself. *SolidWorks* helped us model the piece in 3D. The model was transferred from the CAD software into the CAM software (*SolidCAM*), where all the parameters of the piece and machine were fully defined. This allowed us to generate the G-code so that it could be loaded into *Mach3* and actually mill the piece of MDF block and get the finished product. Overall, it is safe to say that our piece came out fantastic. It came out exactly as imagined, drawn, and defined in the CAD/CAM software. If there wouldn't be such technology, making a piece like this would be much harder and time consuming. The parameters could be probably defined and tweaked even further to get an even better piece; however, for the standards and for what was tried, the piece is completely satisfying.

6. REFERENCES

- [1] http://en.wikipedia.org/wiki/Computer_aided_technologies (Accessed: september 2014.)
- [2] <http://www.solidworks.com/> (Accessed: september 2014.)
- [3] <http://www.solidcam.com/> (Accessed: september 2014.)
- [4] <http://www.machsupport.com/software/mach3/> (Accessed: september 2014.)
- [5] Picture of the CNC machine from VTSBJ college