Modelling of CAM systems to simulate of a milling machine work

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Abstract. Mechatronic is combination of three main disciplines: mechanics, electronics and theory of control. Part of them are CAD/CAM/CAE systems. This work presents the facilitating impact of CAD software on the design process. Basic modeling methods will be presented, including the creation of objects that are the composition of several smaller parts. There are many programs being CAD / CAM / CAE software, however, in order to achieve the assumed goals, Siemens software - NX will be used. It is used to create a simple CAD object and a more complex model that is a milling machine. The simple detail will be presented to describe the basic functions useful during modeling 3D objects, such as sketch-based generation and further modification of the created model. Describing the created milling machine is related with the presentation of the types of constraints. They were used and the way in which selected machine axes were generated, including kinematic and geometrical parameters.

1. CAD/CAM/CAE programs as an engineering tools
Computer Aided Design (CAD) means software that can be used to model objects. There are many programs of this type like Solid Works, Inventor, Solid Edge or Siemens NX. Each of these programs allows to model 2D and 3D objects and assign them the appropriate technological and kinematic attributes [4-8]. Example of a CAD model is shown in figure 1.

Figure 1. The example of a CAD model [1].
Programs of this type are very often implemented as a CAM module, which allows to simulate production processes [4-8]. The example of utilization such program to simulate workpiece machining is shown in figure 2,

![Figure 2](image1.png)

**Figure 2.** The example of the CAM model [2].

In addition to modeling, programs of this type also have CEA modules, which allows to calculate the appropriate strength properties of tested objects [4-8]. The example of the CAE model is shown in figure 3.

![Figure 3](image2.png)

**Figure 3.** The example of the CAE model [3].
2. **NX as a multifunctional tool for engineering tasks**

Siemens NX is a multifunctional tool for engineering applications which is utilized in CAD/CAM/CAE tasks. It allows to create models which could be used for simulation analyzing as well as for strength analysis. [8-9]

Modeling in the NX program can be started by generating a simple solid or drawing a sketch. Then the obtained surface can be drawn with the Extrude option, after selecting the plane on which the sketch will be created. The sketch can be created in this way that after applying the extrusion command simple figure is created. It also allows applying appropriate corrections to the 3D solid using appropriate options.

For products that are usually the result of turning, their geometry can be more easily obtained by rotating the edge. The better option is to use the Revolve function. The whole process is done in the same way as in the case of Extrude, but the axis of rotation, around which the object should be created, should be selected and the angular value of the beginning and the end of rotation should be specified. Then the created simple model, can be modified e.g. by introducing drilling holes, rounding or cutting edges, etc. When making holes, one can select several types, starting from straight, through dredged to conical ones. Another parameter that should be set during drilling holes is their depth and diameter. In order to insert the hole, it is enough to generate a sketch with one point which determines the location of the expected modification. The example of a simple model after applying the Revolve function, and then modified through the Edge Blend functions is shown in figure 4

![Figure 4](image1.png)

**Figure 4.** The example of modeling the object in the NX program utilizing the Revolve function, then rounded off the edges.

Then, holes were made in a simple model and they were duplicated with a circular pattern using the Pattern Feature. The detail view after using the described functions is shown in figure 5.

There are also other functions that allow to modify the created detail, such as: [9]

- **Trim** – allows to cut one object in relation to another object, which is often utilized to create a shear plane curve,
- **Chamfer** – works analogically like Edge Blend with the difference that instead of rounding of selected edges, it chamfers them,
- **Shell** – used to hollow a selected surface with the expected wall thickness,
- **Draft** – allows to add additional material with the behavior of the declared angle.
Figure 5. The example of further modification using the hole generator and Pattern Feature function to generate a pattern

The other modification options and the option allowing to create simple solids without creating a sketch can be found in the Insert tab (Menu bar).

3. Model of the analysed milling center

The model that is the purpose of this work is a 5-axis milling machine. The created milling object is shown in figure 7. In order to model the presented object, it was necessary to combine the modeled parts with appropriate constraints.

In order to create the object a number of different constraints have been used, such as:

- Fix – responsible for establishing the object relative to the coordinate system. It was used to fix the machine frame,
- Align – generates connections to one plane. The example of utilization is shown in figure 6,
- Touch – related objects of this type constrains must have a contact point / surface,
- Distance – it sets the right distance between the details used in order to keep the distance between the X-axis and the end of the machine,
- Parallel – defines the position of objects as parallel to each other.

Figure 6. The example of assembly constrains – align.
Figure 7. Virtual model of a milling machine.

Using “Machine Tool Navigator” the appropriate connections between the axes have been created.

Figure 8. The example of parameters setting of the exemplar axis - maximum value in the Y axis.
As it can be seen during the modeling of any axis, the dimensional parameters and motion parameters should be considered. After assigning the appropriate parameters, a simple simulation can be carried out to check whether the parameters of the selected axis have been properly selected. In figure 9 is shown the interrelationships between the X and Z axis. The parameters for each axis have been defined separately in a manner analogous to the Y axis.

![Figure 9. The example of assigning the axis interrelations](image)

4. Conclusion
The work presents the most important modeling methods illustrations them with appropriate examples. By the example of the NX program it shows the possibilities offered by modeling using CAD programs. It is illustrated how using a few functions, could be obtained a model, that more precisely suits to the actual one.

By using many modeling possibilities NX is the ideal tool for creating 3D models. The purpose of further work will be to continue the implementation of CAD software and to simulate the created model to conduct, investigations of a mechatronics function.

5. References
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