Chapter 8.
Tool path control

In this chapter, you will become familiar with options that are useful for tool path control, its editing and transformation.

Tool path status control

The tool path status control in NX CAM is an extremely useful tool in a programming technology engineer's daily work.

1. Open the *Toolpath_status.prt* file from the …*r08\NX_plilki\Status\…*folder.

In order to discuss these problems, first generate the program from the two operations defined so far.

2. From the *Operation Navigator* position — click *MB3* on the folder's name — here `PROGRAM` — and select the *Post Process* option — figure 8.1 (1).

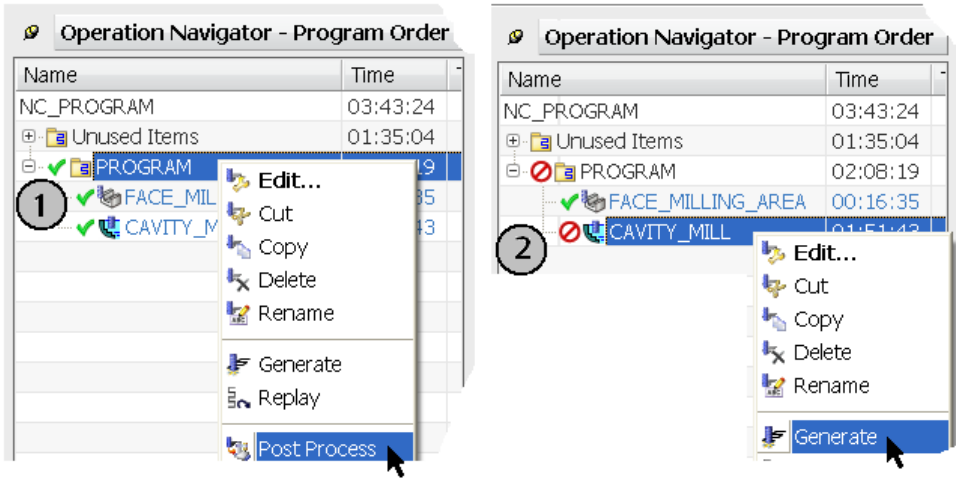

Figure 8.1 Code generation and tool path recalculation

3. After generating the code, pay attention to the ***green*** markers, located close to the operation icons – figure 8.1 (1).

Tip

The green marker – it informs that the specific operation is correct and the NC code has been generated.

4. Edit, e.g. the `CAVITY_MILL` operation and change e.g. its *Cut Levels* parameter to any value, other than the one seen in the dialog box and different from the existing one, and exit the operation using the *OK* buttons.

Caution: Do not generate the tool path again.

5. The ***red*** marker will appear close to the operation (a crossed-out circle) — figure 8.1 (2).

Tip

The red marker – it informs that a specific operation is out-of-date or incorrect – in this case it does not correspond with parameters in the dialog box. The path has not been recalculated after modifications made in the dialog box.

If parameters of consecutive operations depend on this operation (e.g. while using *IPW*, rest milling…), the red marker will also appear by the next ones. A similar effect gives e.g. a change of geometrical parameters of the tool, operation definition and exit from the operation - without the tool path recalculation.

6. Click on the *MB3* `CAVITY_Mill` operation and calculate it again using the *Generate* command.
7. A ***yellow*** marker (an exclamation mark) will appear next to the operation — figure 8.2 (3).

Tip

The yellow marker informs that a specific given operation is correct but the NC code has not been generated.

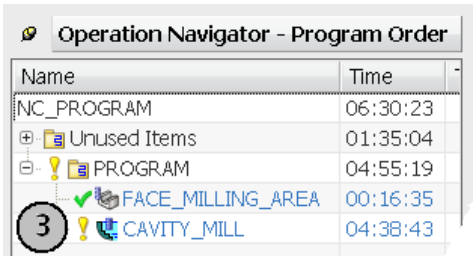

Figure 8.2 Marker's view after the recalculation of the tool path

Editing of the MCS position

Let's assume that we need to change the MCS position in relation to the position defined at the beginning. How will it impact the tool path status?

8. In the *Navigator Operation* go to *Geometry View* — figure 8.3 (a).

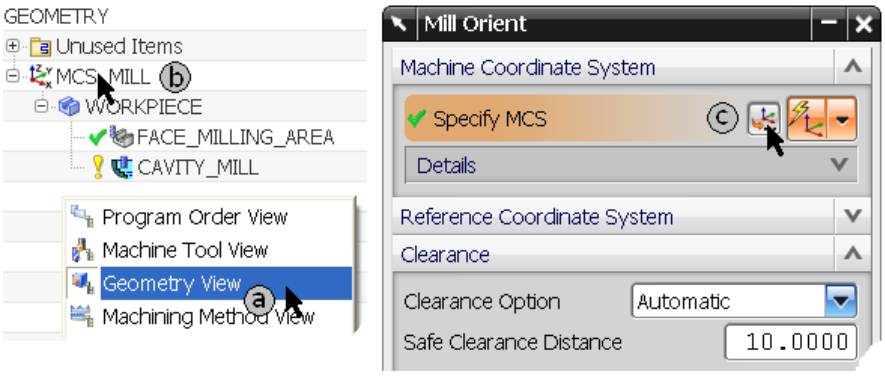

Figure 8.3 Consecutive steps in changing the MCS

9. Double-click on the `MCS_MILL` icon — figure 8.3 (b).

10. The *Mill Orient* dialog box appears. Click on the *CSYS Dialog* icon — figure 8.3 (c).
11. Dynamically move the `MCS` to a randomly chosen place — e.g. downward from the upper part and additionally change the direction of the X or Y axis — figure 8.4.

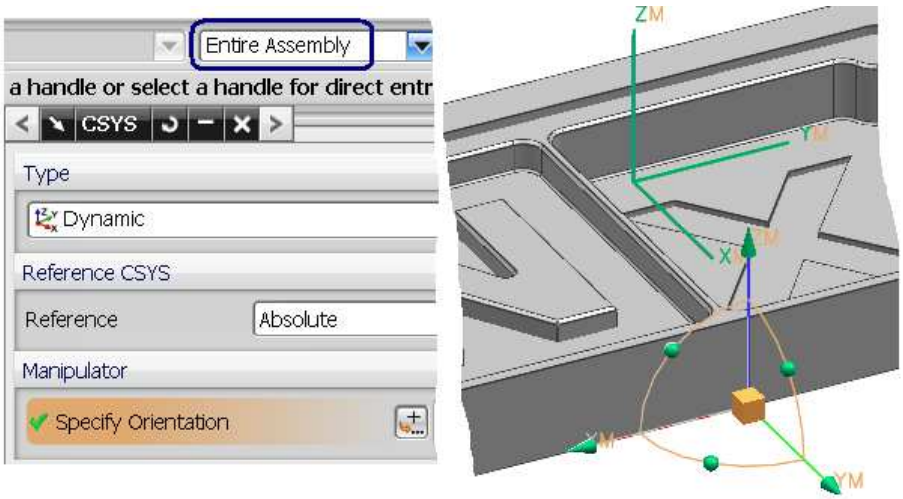

Figure 8.4 Editing of the MCS position

Due to the character of 3-axis machining, do not change the direction of the Z-axis.

12. As we are working in an assembly, remember to set the *Selection Scope* as `Entire Assembly`.

After the confirmation of the new position, the marker by operations will change into a yellow exclamation mark.

Tip

The yellow marker informs that a specific given operation is correct and ready to generate an NC code. You do not have to calculate the tool path again but generate the code immediately, with coordinates relating to the new MCS position.

Feed edition

Let's assume that there is a necessity to change the feed values. How will it impact the tool path status?

13. Notice what values of the machining time are currently in the *Time* column in the *Operation Navigator* –the total time on the upper side of the column and at specific operations.
14. Edit the feed value e.g. in the `CAVITY_Mill` operation.
15. Click on the *Feeds and Speeds* icon in the dialog box – figure 8.5 on the right-hand side.

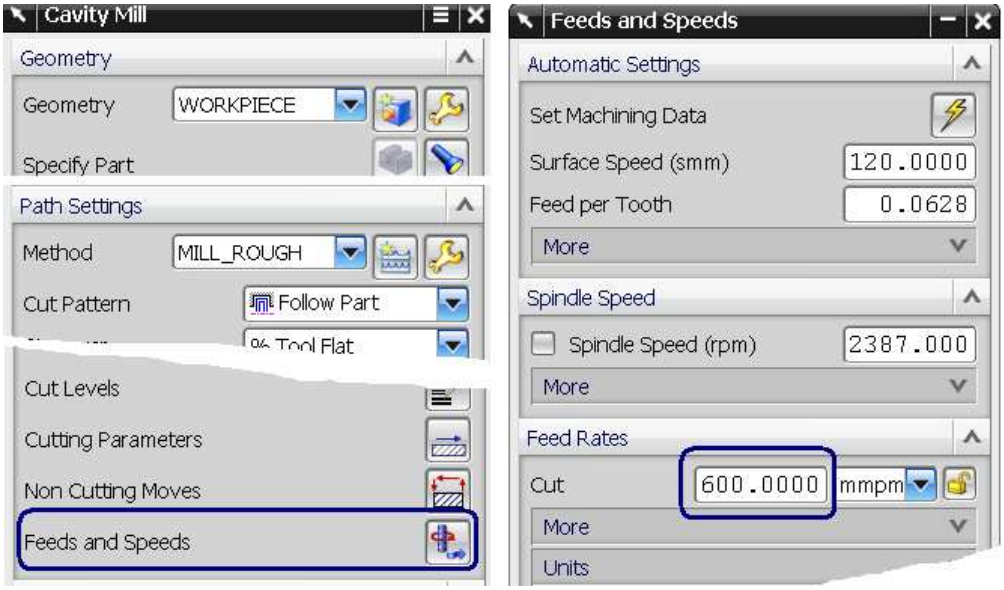

Figure 8.5 Feed value editing.

16. Change the *Cut* feed value (main feed rate) — figure 8.5 on the right-hand side — to a randomly chosen value, other than the current one and exit the operation using the *OK* buttons.

    Caution: Do not generate the tool path again.

17. In the *Time* column, an update of the edited operation's machining time and total time will occur.

Tip

The yellow marker will remain by the operation – you do not have to calculate the tool path again, but immediately generate the code with modified feed values.

Editing of the handle position

Let's assume that we need to change the handle position in relation to the position defined at the beginning. How will it impact the tool path status?

18. Click on the *Move Component* icon (or *Assemblies /Components* menu…) — figure 8.6 (a).

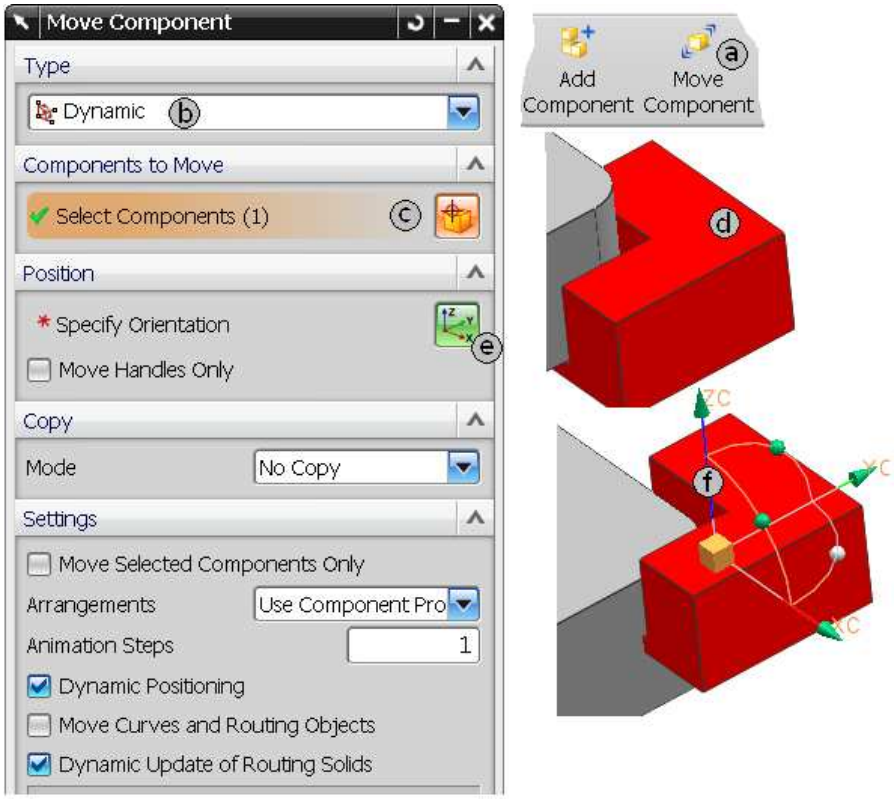

Figure 8.6 Shift of the handle component

19. A dialog box will appear and you will have to choose the *Dynamic* option (from this dialog box) — figure 8.6 (b).
20. Click on the *Components to Move* icon — figure 8.6 (c).
21. Click on one of the handles — figure 8.6 (d).
22. Click on the *Position* icon — figure 8.6 (e).
23. A dynamic system of coordinates will appear by the handle — figure 8.6 (f).
24. Change the handle position – e.g. drag the `ZC` axis so that the handle can protrude outside the component's surface and finish the operation.
25. Exit the operation using the *OK* buttons.

    Caution: Do not generate the tool path again.

Tip

The red marker will appear by the operations. You have to calculate the tool paths again so that they can take into account the modification of the component fixation.

Editing lock

After the completion of tool path definition modification, we can give it a lock status for subsequent editing. It does not mean that the tool path cannot be changed at all, but we will be informed about such an intention with an appropriate message.

26. Click on *MB3* e.g. on the `FACE_MILLING_AREA` operation.
27. Go to the *Tool Path* option — figure 8.7 (1).

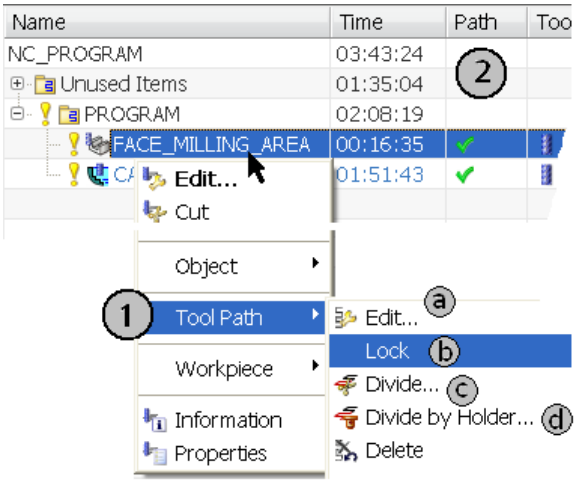

Figure 8.7 Tool path editing options

28. Select the *Lock* option — figure 8.7 (b).
29. A padlock icon will appear in the *Path* column in *Operation Navigator* — figure 8.7 (2).
30. Now, edit this operation – change any parameter and try to calculate the tool path.
31. A dialog box will appear (as in figure 8.8).

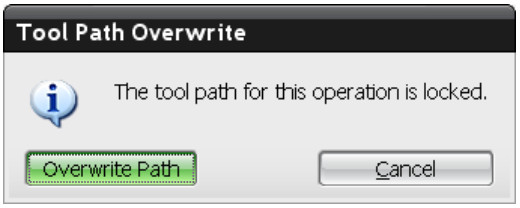

Figure 8.8. Message concerning tool path overwrite

If you click on *Overwrite Path*, the tool path will be calculated with new parameters. If you click *Cancel*, the modifications will be ignored.

Tip

The Edit… command — figure 8.7 (a) — enables launching the tool path editor, by means of which it is possible to execute deep manual interventions into the tool path segments — the description of *Tool Path Editor* can be found later in this chapter.

Tool path division — machining time…

When bulky elements are to be machined, e.g. models of components, the roughing time often extends to over 20 hours. Then, it is necessary to divide the machining, taking into consideration the tool wear or its control measurement. In NX CAM, there is the possibility to make such a division of the tool path by means of the option shown in figure 8.7 (c).

32. Click on *MB3* on the `DIVIDE` operation.
33. Go to the *Tool Path* option — figure 8.7 (1).
34. Select the *Divide* option — figure 8.7 (c).
35. A dialog box for the division definition will appear — figure 8.9.

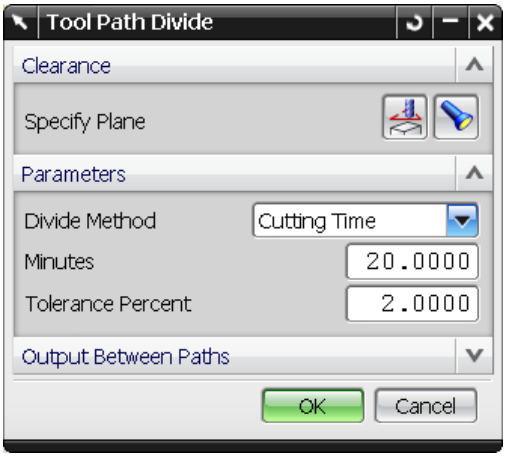

Figure 8.9. Options of tool path division

The tool path may be divided according to the available *Divide Method* :

- *Cutting Time* — measured in minutes;
- *Cutting Distance* — measured in millimetres;
- *Selected Motions* — on selected tool paths.

36. Select e.g. *Cutting Time* method and enter such a time value in the *Minutes* field, after which the tool path division should be executed.
37. By using the *Specify Plane* icon, it is possible to define *Clearance*, within which the tool will be extended after the tool path division.
38. The tool path will be divided according to specified criteria, and the consecutive divided tool paths will be marked with `_DIV_` index and a consecutive number — figure 8.10.

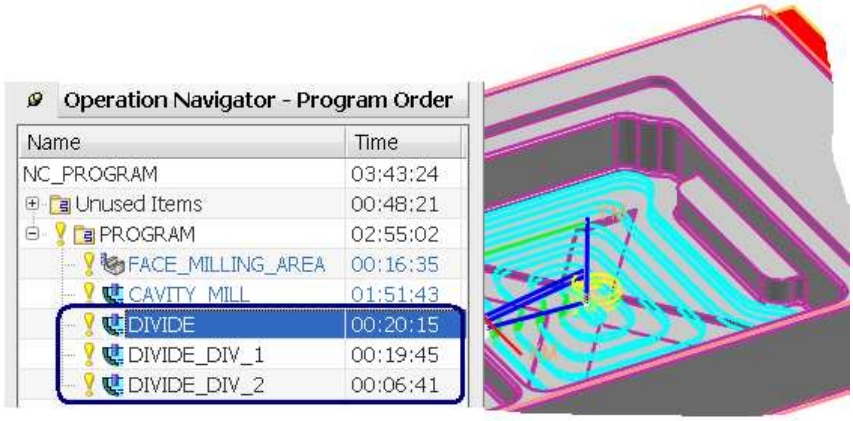

Figure 8.10. Tool path divided according to time

Tip

In order to eliminate the division (or another type of edition), just calculate the operation again.

Tool path division according to the holder

The possibility of dividing the tool path according to the holder interference is useful when we want to machine the maximal height of the component with the optimal extension distance (rigidity) of the tool. The remaining component can be machined on the longer extension distance with smaller parameters of machine cutting.

39. Click on*MB3* on the `HOLDER` operation — there is a tool with a short extension distance.
40. Go to the *Tool Path* option — figure 8.7 (1).
41. Select the *Divide by Holder…* option — figure 8.7 (d).
42. A dialog box with a definition of division will appear — click on *OK* — the tool path will be divided.
43. Under the `HOLDER` operation, the next one with `_DIV` index will appear — as in figure 8.11.

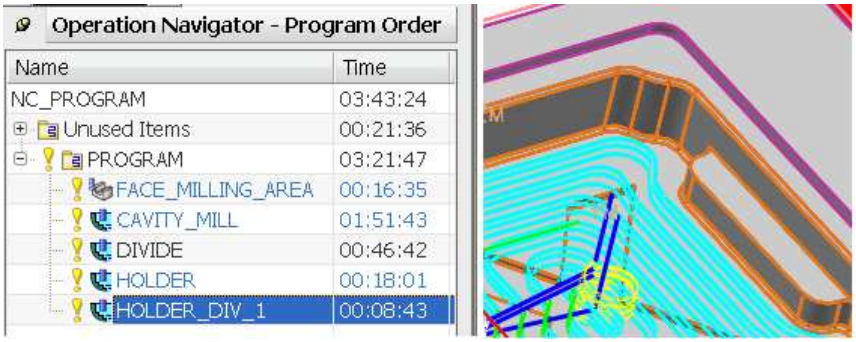

Figure 8.11. The tool path divided according to the holder interference

44. Another tool, with a longer extension distance may now be assigned to the divided (rejected) tool path.

Tool path transformations

Operation tool paths may be put to transformations, similar to geometrical transformations, such as reflections, copying by arrays etc.

45. Click on *MB3* on `TRANSFORM` operation.
46. Go to the *Object* option — figure 8.12 (1) and select the *Transform…* command.

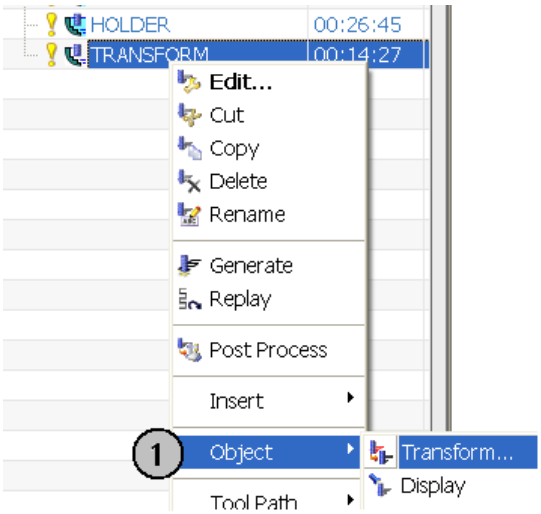

Figure 8.12. ”Transform…” command activation

47. A dialog box for the tool path transformation definition will appear — figure 8.13.

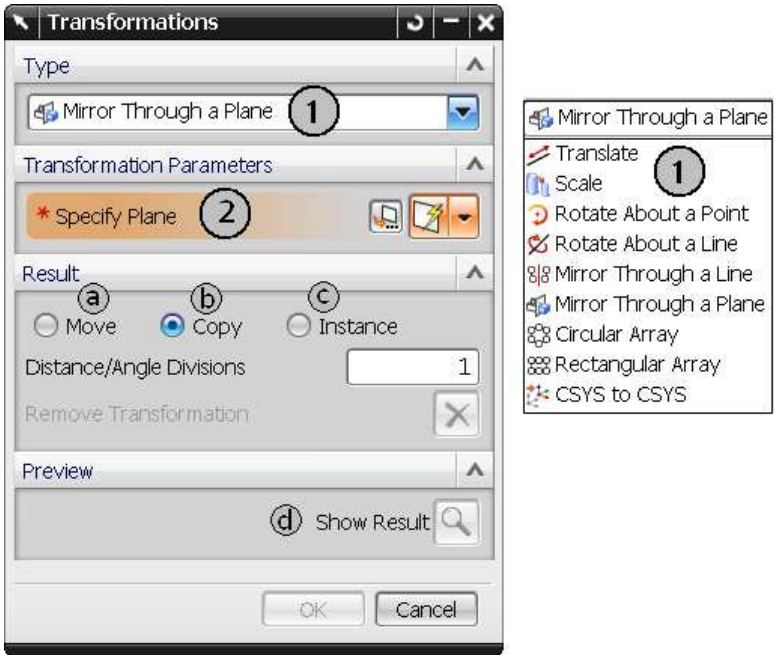

Figure 8.13. Transformation dialog box

Transformation options

48. In the upper part of the dialog box — figure 8.13 (1) — we select *Type* for the transformation. On the right-hand side of the figure, there are available commands. In this case, let's select *Mirror Through a Plane*.
49. Depending on the selected command, *Transformation Parameters* will be available — figure 8.13 (2). Here, click on the *Inferred* icon and select any vertical face from the model.

Tip

If the command cannot see component model faces, check if the `Entire Assembly` option is selected in the *Selection Scope*.

50. In the *Result* group, we determine the type of relationship between the transformed tool path and the original one:
    - *Move* — this option transforms the tool path and it does not leave the original one — figure 8.13 (a).
    - *Copy* — this option transforms the tool path and it leaves the original one but later it is independent of it — figure 8.13 (b). Changes in the original tool path are not reflected in the transformed tool path.
    - *Instance* — this option transforms the tool path and it leaves the original one, but later it will be dependent on the original tool path — figure 8.13 (c). Changes in the original tool path are reflected in the transformed tool path.
51. In this case, we set the *Copy* option.
52. Switch on the tool path view by the *Show Result* icon — figure 8.13 (d).
53. If the view is as required, click on *OK*.
54. In the *Operation Navigator* dialog box, the next tool path with `_COPY` reference will appear — e.g. as in figure 8.14.

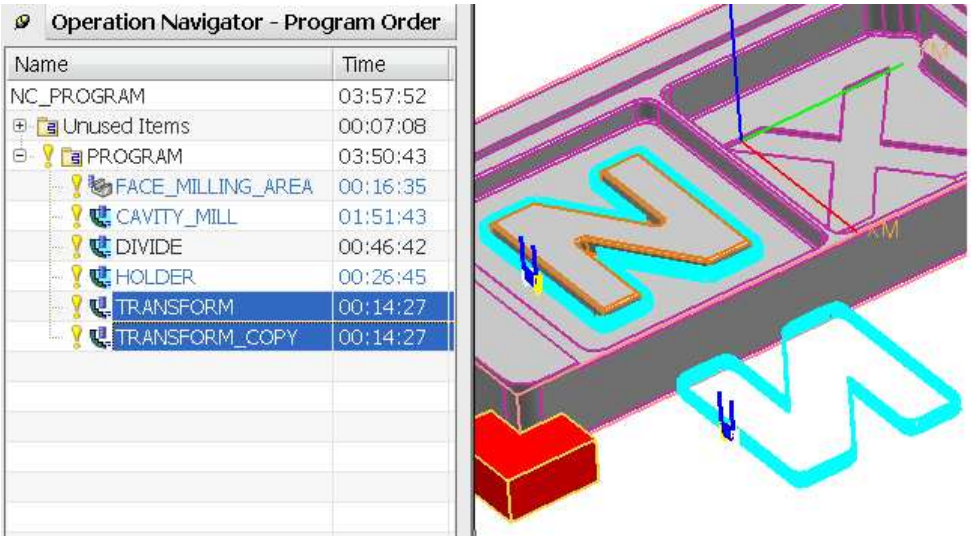

Figure 8.14. Post-reflection view of the tool path

55. Press the *Ctrl* key and select both tool paths with the cursor in order to view them simultaneously on the screen.

Tip

The TRANSRORM_COPY tool path is independent and you can edit it. In the operation dialog box you can change particular options for machining, e.g. machining direction (concurrent, backward), cutting depth etc.

Tool Path Editor

The Edit Command — figure 8.15 (1) — allows a "deep" intervention in the tool path. With the available commands you can for example:

- *Add New Path Event* — figure 8.15 (2);
- *Move. Extend* a specified segment of the tool path or *Reverse* the direction of the whole tool path — figure 8.15 (3);
- To delete particular segments of tool paths you can use the *Delete* command — figure 8.15 (a);
- A frequently used option is the possibility to trim the path to a specified surface using the *Trim* option — as in figure 8.15 (b, 4).

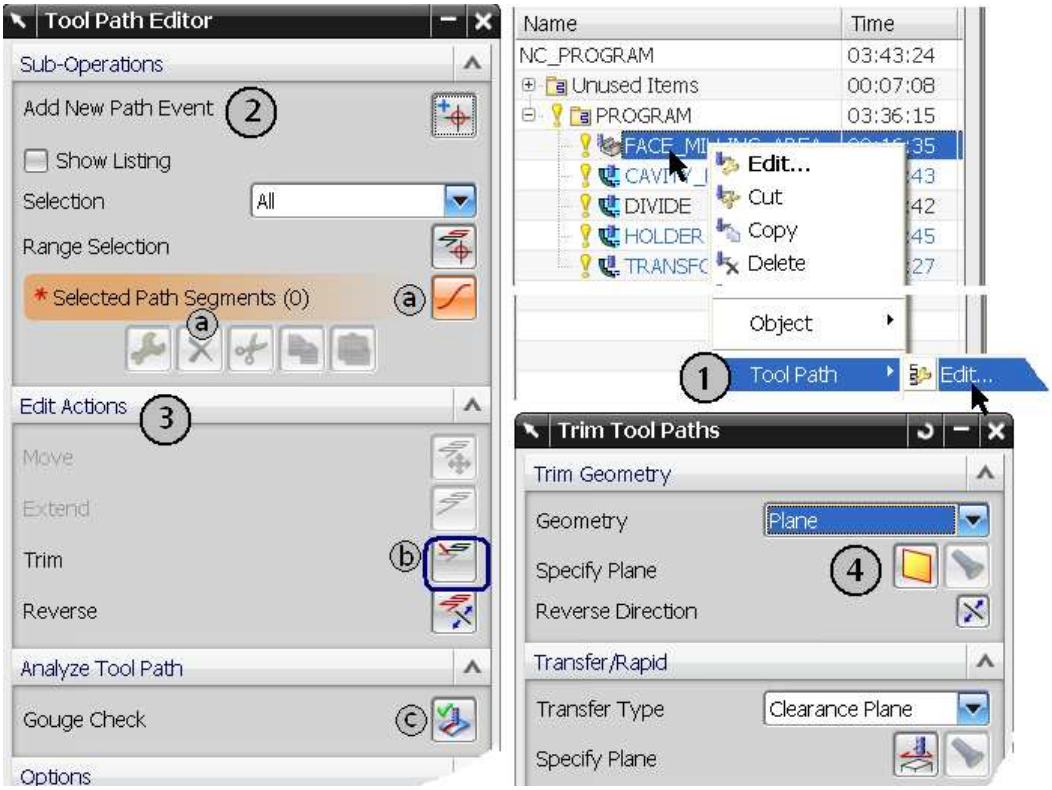

Figure 8.15. View of the Tool Path Editor window

You can submit introduced changes to an examination aimed at the detection of collisions with the component model — this option can be seen in figure 8.15 (c).

Feed control

Depending on his/her needs, the programmer may use one main working feed rate *Cut* and rapid moves `G0` (`FMAX`) or divide all the approach feeds, inputs, etc. into different values.

Setting the feed values

Feed values may be set automatically (for reference, see chapter 4) or determined manually.

56. Edit the `FEEDS` operation.
57. Click on the *Feeds and Speeds* icon — figure 8.16 (1).

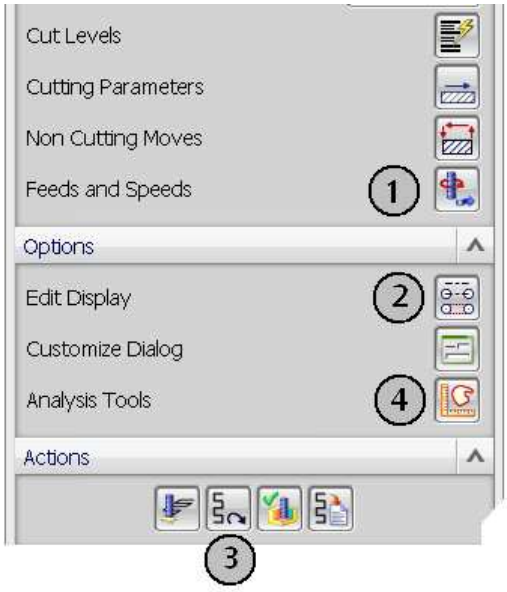

Figure 8.16. Options for the operation dialog box

58. A dialog box appears. Here you can only see the default, filled value of the feed rate *Cut* in the *Feeds* group — figure 8.17.

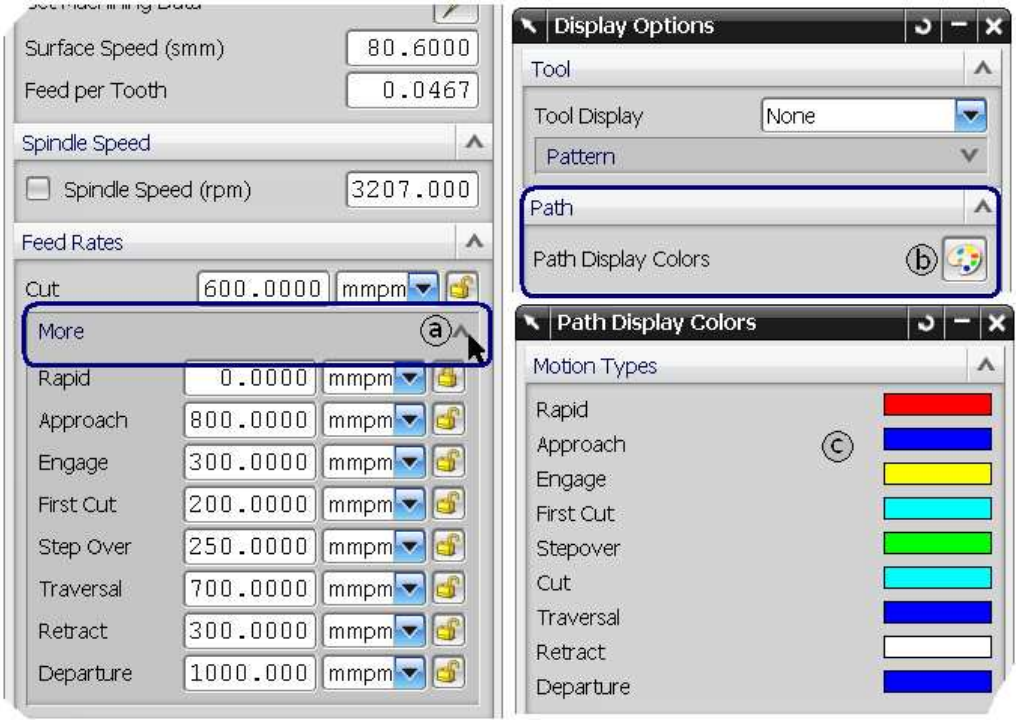

Figure 8.17. Table of feeds and colours

59. If you cannot see other values of feeds, click on the *More* arrow — figure 8.17 (a).

Particular names of feeds and their order in the table are related to the sequence of moves, which usually takes place from the beginning up to the end of the operation, or passage of one machining level through the tool.

Division of feeds

Due to the character of contact with material, moves (and feeds assigned to them) can be divided into:

- cutting — in contact with the machined material — marked on the screen with a solid line;
- dead movement – not in contact with the machined material – marked on the screen with a dashed line; realized by rapid move `G0` or `FMAX`.

In order to check which tool path colour is assigned to the specific type of feed by default:

- Click on the *Edit Display* icon — figure 8.16 (2).
- A dialog box will appear. Here, in the *Path* group, click on the *Path Display Color* icon — figure 8.17 (b).
- A dialog box will appear. Here you can see individual colours, assigned to specific moves.

Move sequence in the operation

Watching the nomenclature of feeds in table 8.17, in practice, their sequence looks as follows:

- *Rapid* — this is a rapid move feed, realized by the `G0` or `FMAX` function — dead movement.

  Instead of the `0` value, you can enter the maximum feed value, e.g. `8000` — then, this move will be realized as `G1 F8000`.

  Tool paths on the screen are shown in red.

- *Approach* — this is a move feed realized from the start point to *Engage* — dead movement.

  In operations at constant levels Z (e.g. *Cavity Milling*, *Zlevel Profile*) with its help, you can realize the move from one level onto another. If its value in the table equals `0`, then it is realized as rapid move `G0`.

  Tool paths on the screen are shown in navy blue.

- *Engage* — this is a move feed penetrating into the material — cutting movement.

  If its value in the table equals `0`, then it receives the value of the feed rate *Cut*.

  Tool paths on the screen are shown in yellow.

- *First Cut* — this is the feed of first path sections in the material — cutting movement.

  If its value in the table equals `0`, then it receives the value of the feed rate *Cut*.

  Tool paths on the screen are shown in cyan.

- *Step Over* — this is the feed connecting two parallel, adjacent tool paths — cutting movement.

  It exists, if the *Cut Pattern* is set to *Zig-Zag*.

  If its value in the table equals `0`, then it receives the value of the feed rate *Cut*.

  Tool paths on the screen are shown in green.

- *Cut* — this is the feed of the main machining move — cutting movement.

  Tool paths on the screen are shown in cyan.

- *Traversal* — this is the feed of rapid, horizontal move — dead movement.

  It is applicable if in *Non Cutting Moves* onto the *Transfer/Rapid* card the *Transfer Type* option is selected as *Direct*. If its value in the table equals `0`, then it is realized as move `G0`.

  Tool paths on the screen are shown in navy blue.

- *Retract* — this is a move feed that outputs the material — cutting movement.

  If its value in the table equals `0`, then it receives the value of the feed rate *Cut*.

  Tool paths on the screen are shown in white.

- *Departure* — is a final move feed, which is realized from the last *Retract* — dead movement. If its value in the table equals `0`, then it is realized as rapid move `G0`.

  Tool paths on the screen are shown in navy blue.

Displaying feed values

In order to display individual feed values on the tool path onto the screen:

- Click on the *Edit Display* icon — figure 8.16 (2).
- A dialog box will appear. Here, in the *Path* group, click on the arrow icon next to *More* — figure 8.18.

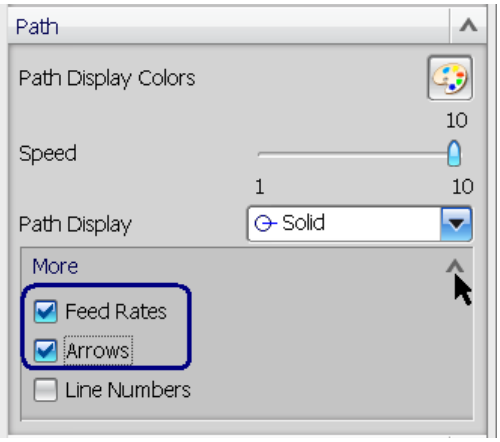

Figure 8.18. Switching on the feed display

- Select the *Feed Rates* option — displaying feed values on the tool path during their change.
- Select the *Arrows* option — displaying arrows on the change of the tool path direction.
- Click on *OK*.
- In the main dialog box of the operation, click on the *Replay* icon in order to refresh the view of tool paths — figure 8.16 (3).

On the screen, on the tool paths of specific operations, feed values appear — figure 8.19 at the bottom — and arrows of the machining direction. Feed values are also displayed when deceleration at corners occurs — figure 8.19 at the top.

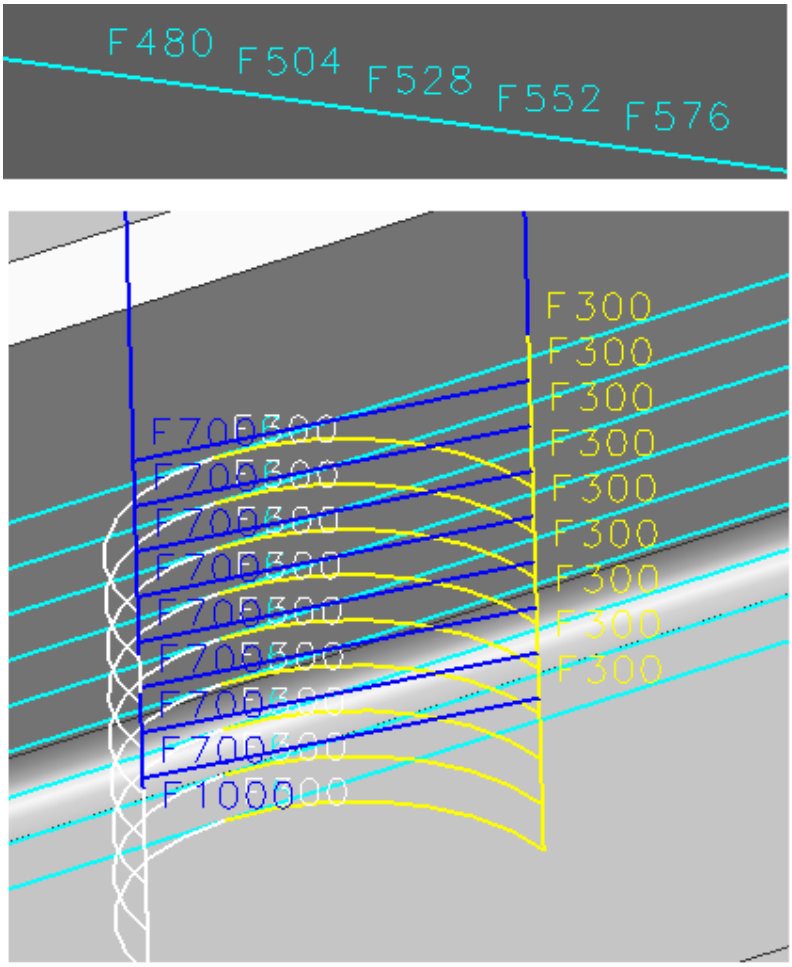

Figure 8.19. View of feed changes on the tool path

Tool path analysis

By using the *Analysis Tools* command — figure 8.16 (4) — it is possible to analyse the path level by level with the display of the feed, machining area etc.

Copying the machining operation

The application of *Synchronous Technology* (more information to be found in chapter 11.) in NX 6.0, resulted not only in an easiness of editing and transfer of modelling operations between files, but also of copying of machining operations to a different file. It enables rapid transfers of one or more operations, or of a whole process to a file with different geometry. Together with the operation settings, tools are also moved.

Copying within the scope of one file

This is a method of operation copying, which is well-known from the previous versions of NX.

Principles and course of the process:

- In order to fully control the copying process, you have to execute it in *Geometry View*.
- With the cursor, select one or more operations (if more, then you have to press the *Ctrl* button).
- Click on *MB3* on operation (operations) and select the *Copy* option — figure 8.20 (a).

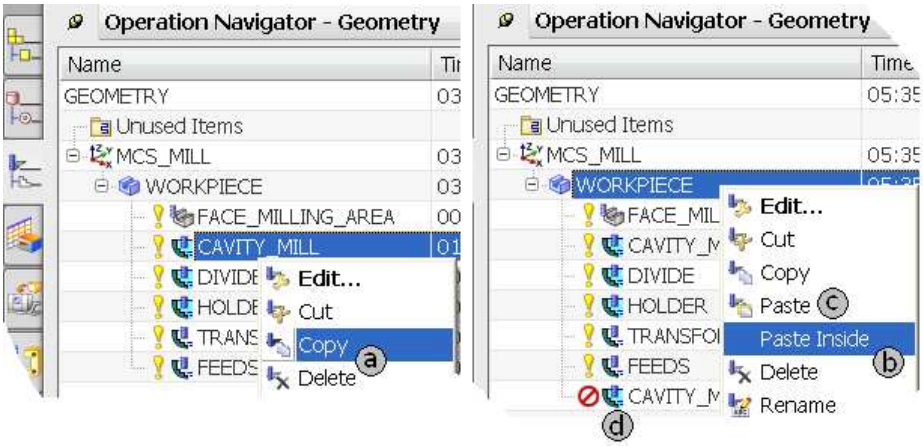

Figure 8.20. Copying operations within the same file

- Click on *MB3* on a specific `WORKPIECE`, onto which you want to copy the operation, and select the *Paste Inside* option — figure 8.20 (b).
- You can also click on another operation, after which you want to insert the copied operation – after its selection, choose the *Paste* option — figure 8.20 (c).
- The copied operation will have an out-of-date status (red marker) and it will be marked with `_COPY` index — as in figure 8.20 (d).
- In the case of using the *Paste Inside* option, the operation is always placed at the end of the tree. You can drag the operation to the top with the cursor. After moving the cursor over another operation, it will be placed beyond the indicated one.

Copying the operation into another file

Copying the operation into another file does not quite differ from the previously presented process.

Principles and course of the process:

- Open two files simultaneously: one, which you want to copy the operation from and the other, into which operations will be copied.
- With the cursor, mark one or more operations in the file, from which you will copy the operations.
- Click on *MB3* on the operation (operations) and select the *Copy* option — figure 8.20 (a).
- Go to the other file — *Window*… menu.
- Click on *MB3* on the specific `WORKPIECE`, onto which you want to copy the operation, and select the *Paste Inside* option — figure 8.20 (b).

Tip

Copying between files works only in *Geometry View*, where the `WORKPIECE` icon is visible.

Here we finish this short description of the most frequent cases of tool path control. In the next chapter we will concentrate on rest roughing.