# How We Machined Boxy


Fred Proctor, NIST

ISO TC 184 SC 4

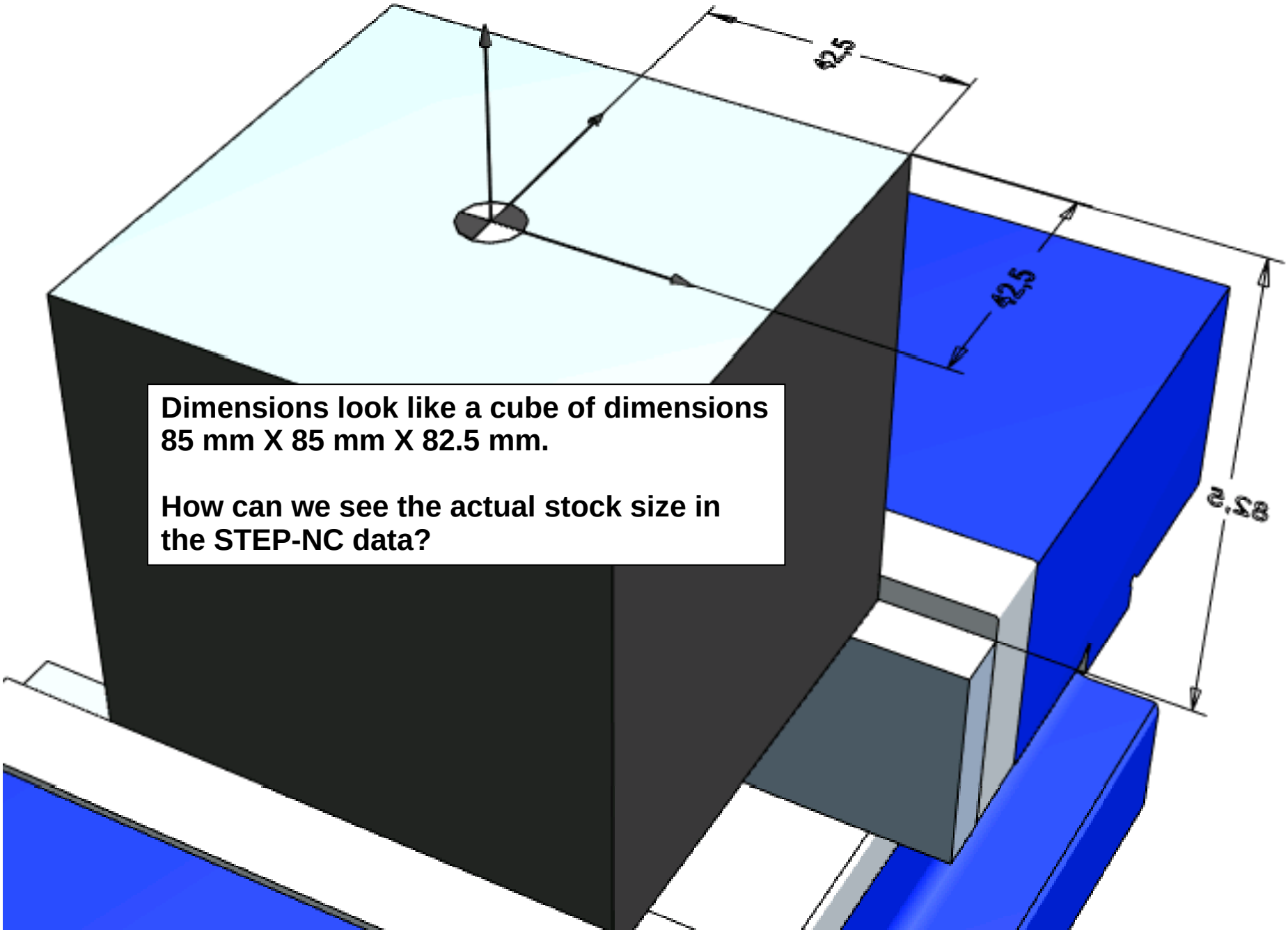
STEP Manufacturing Team Meeting

June 16-18, 2010

NIST, Gaithersburg MD USA

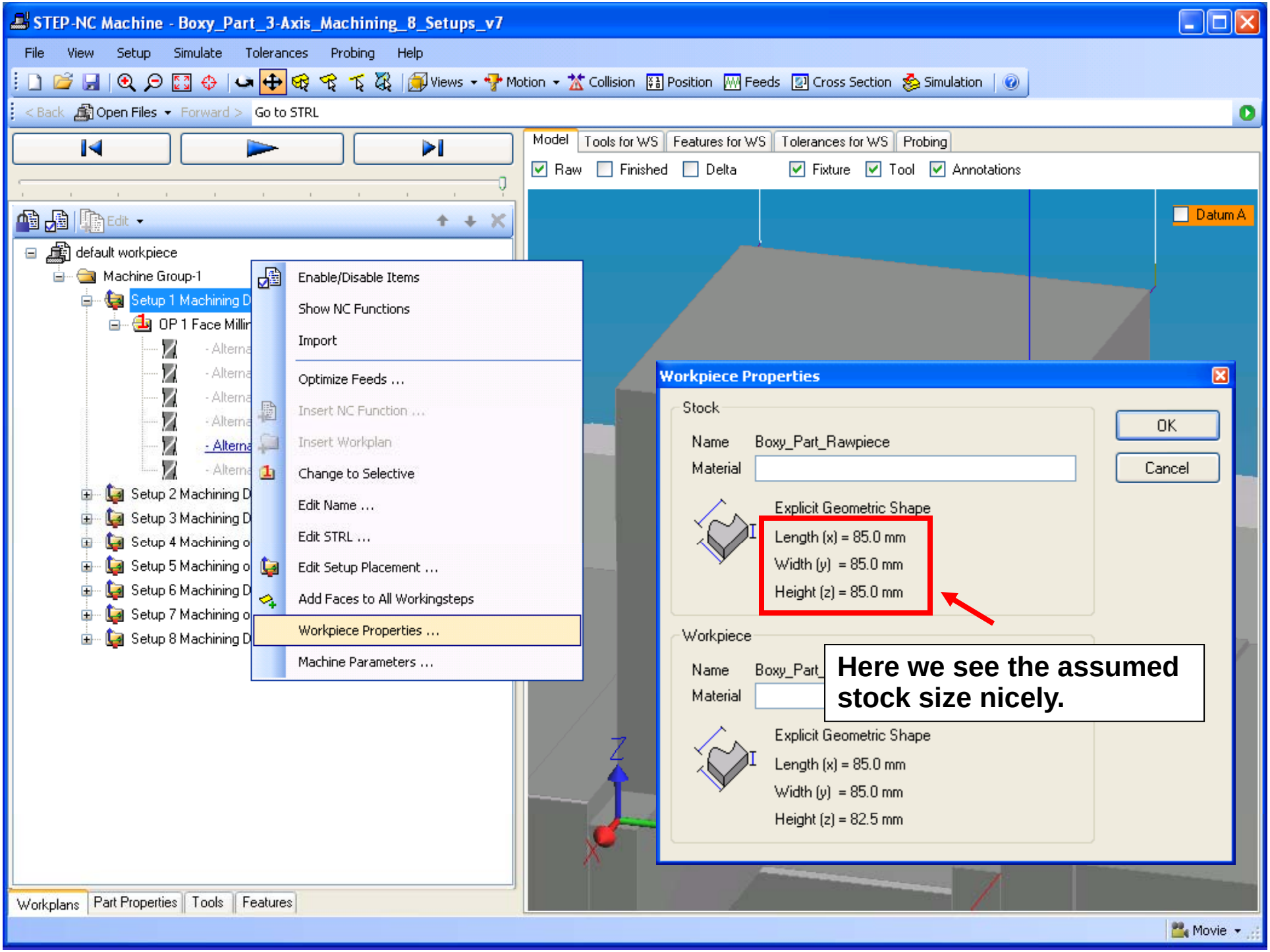
# The Process

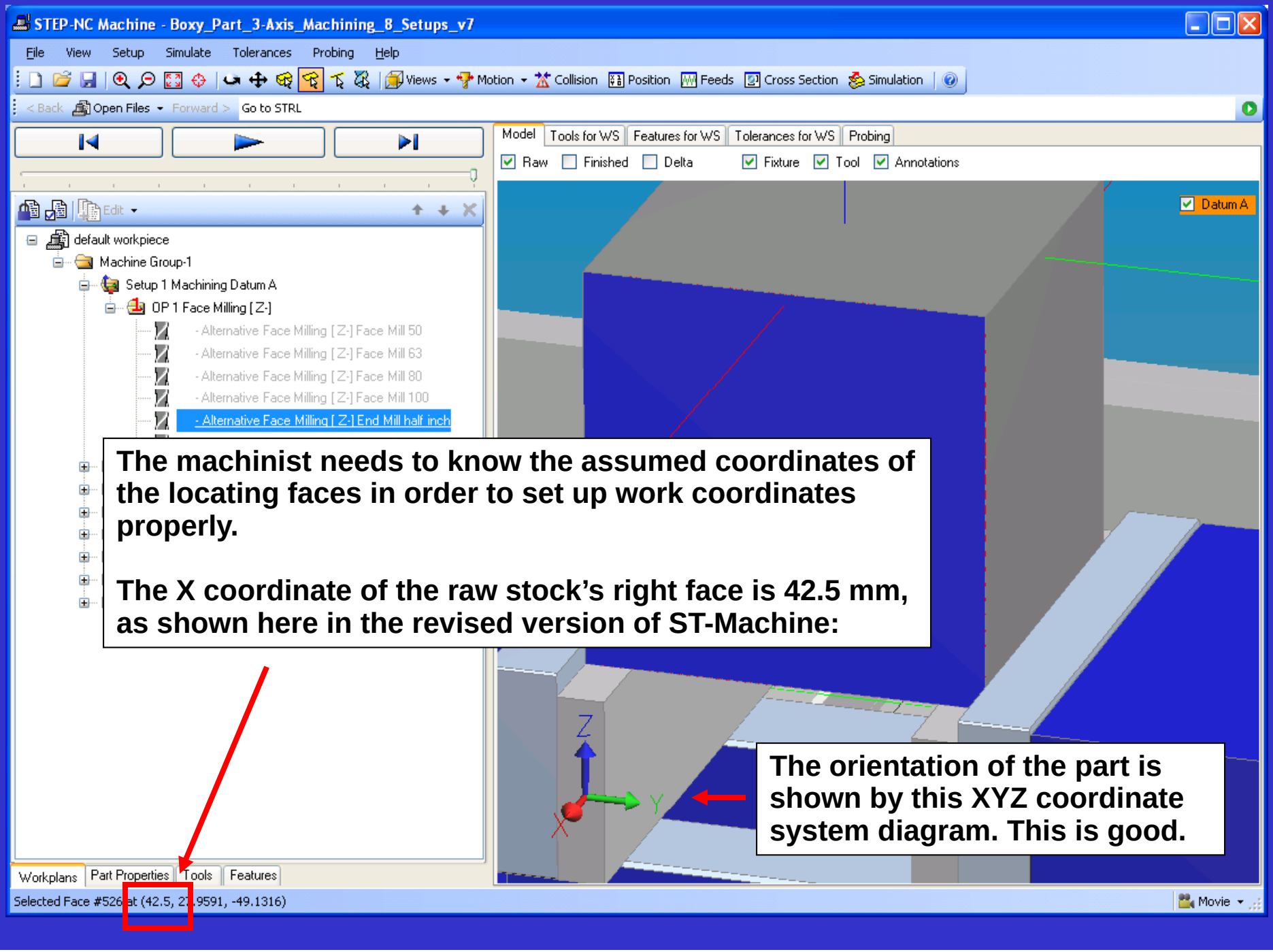
- We started with the 8-setup STEP-NC file with the accompanying 3D PDF that showed the setup assumptions
- We preferred a 7-setup process plan and got one
- We found we needed more information on setup and tooling from the STEP-NC data and got it
- We cut parts in wax and steel and found we had to adjust speeds and feeds to suit our setup
- We looked into cutting forces and spindle power

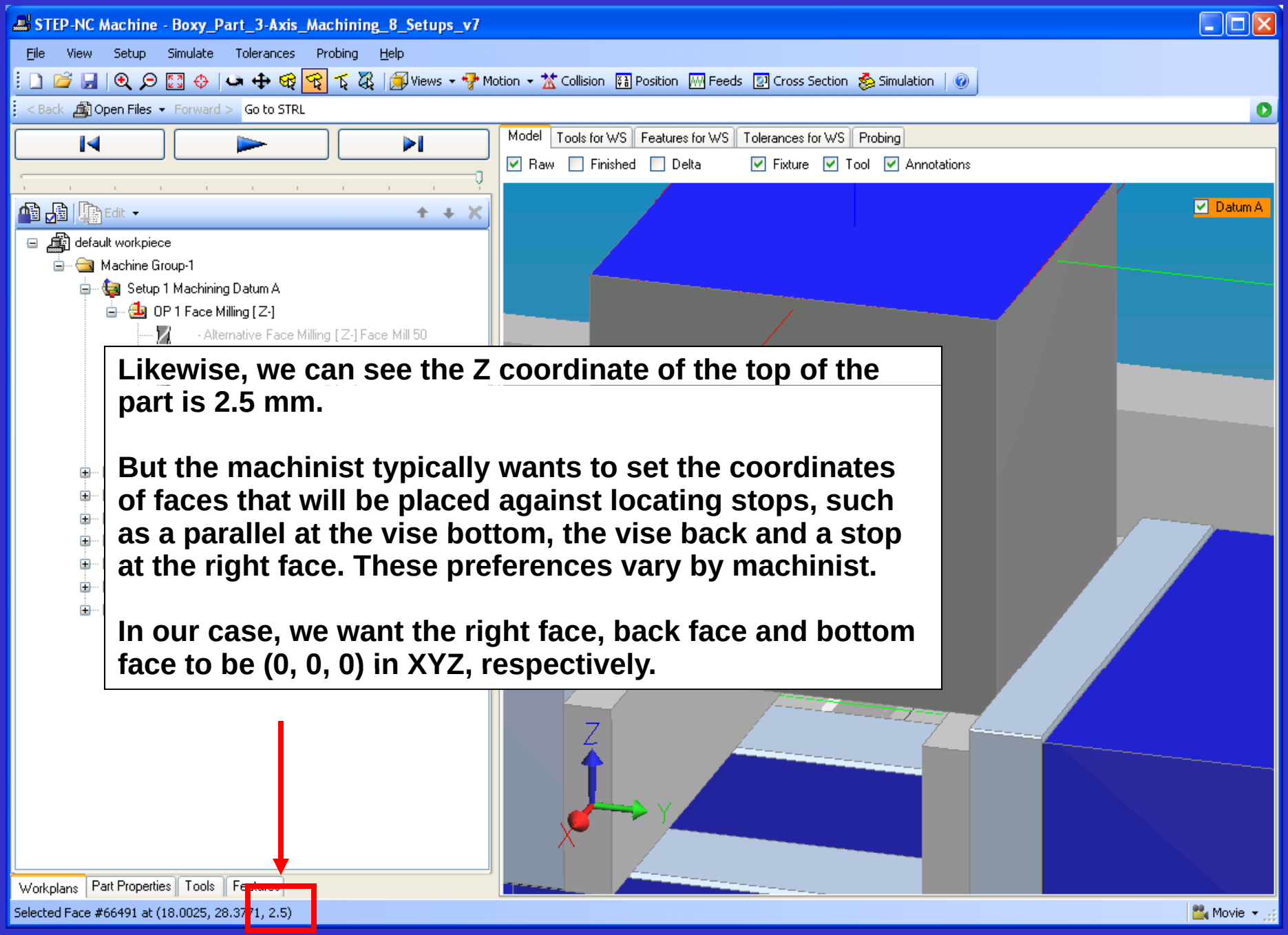



**Dimensions look like a cube of dimensions 85 mm X 85 mm X 82.5 mm.**

**How can we see the actual stock size in the STEP-NC data?**


STEP-NC Machine - Boxy_Part_3-Axis_Machining_8_Setups_v7
File View Setup Simulate Tolerances Probing Help
Views Motion Collision Position Feeds Cross Section Simulation
< Back Open Files Forward > Go to STRL
Edit
default workpiece
Machine Group-1
Setup 1 Machining D
OP 1 Face Millir
Setup 2 Machining D
Setup 3 Machining D
Setup 4 Machining o
Setup 5 Machining o
Setup 6 Machining D
Setup 7 Machining o
Setup 8 Machining D
Enable/Disable Items
Show NC Functions
Import
Optimize Feeds ...
Insert NC Function ...
Insert Workplan
Change to Selective
Edit Name ...
Edit STRL ...
Edit Setup Placement ...
Add Faces to All Workingsteps
Workpiece Properties ...
Machine Parameters ...
Workplans Part Properties Tools Features
Model Tools for WS Features for WS Tolerances for WS Probing
Raw Finished Delta Fixture Tool Annotations
Datum A
Workpiece Properties
Stock
Name Boxy_Part_Rawpiece
Material
Explicit Geometric Shape
Length (x) = 85.0 mm
Width (y) = 85.0 mm
Height (z) = 85.0 mm
OK
Cancel
Workpiece
Name Boxy_Part_
Material
Explicit Geometric Shape
Length (x) = 85.0 mm
Width (y) = 85.0 mm
Height (z) = 82.5 mm
Here we see the assumed stock size nicely.
Movie

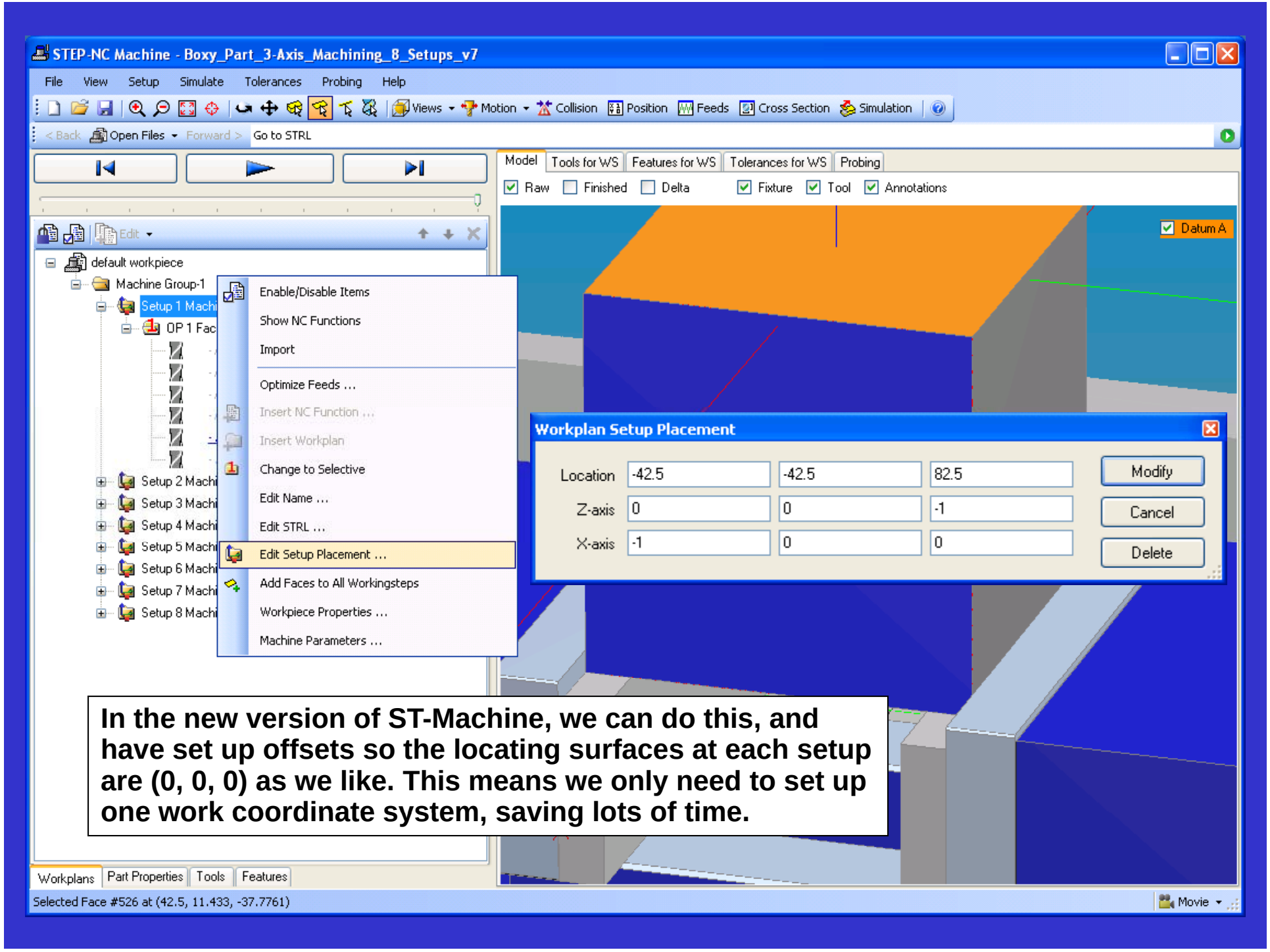

STEP-NC Machine - Boxy_Part_3-Axis_Machining_8_Setups_v7
File View Setup Simulate Tolerances Probing Help
Views Motion Collision Position Feeds Cross Section Simulation
< Back Open Files Forward > Go to STRL
Edit
default workpiece
Machine Group-1
Setup 1 Machining Datum A
OP 1 Face Milling [Z-]
- Alternative Face Milling [Z-] Face Mill 50
- Alternative Face Milling [Z-] Face Mill 63
- Alternative Face Milling [Z-] Face Mill 80
- Alternative Face Milling [Z-] Face Mill 100
- Alternative Face Milling [Z-] End Mill half inch
Model Tools for WS Features for WS Tolerances for WS Probing
Raw Finished Delta Fixture Tool Annotations
Datum A
Z
Y
X
Workplans Part Properties Tools Features
Selected Face #526 at (42.5, 2 .9591, -49.1316)
Movie
The machinist needs to know the assumed coordinates of the locating faces in order to set up work coordinates properly.
The X coordinate of the raw stock's right face is 42.5 mm, as shown here in the revised version of ST-Machine:
The orientation of the part is shown by this XYZ coordinate system diagram. This is good.

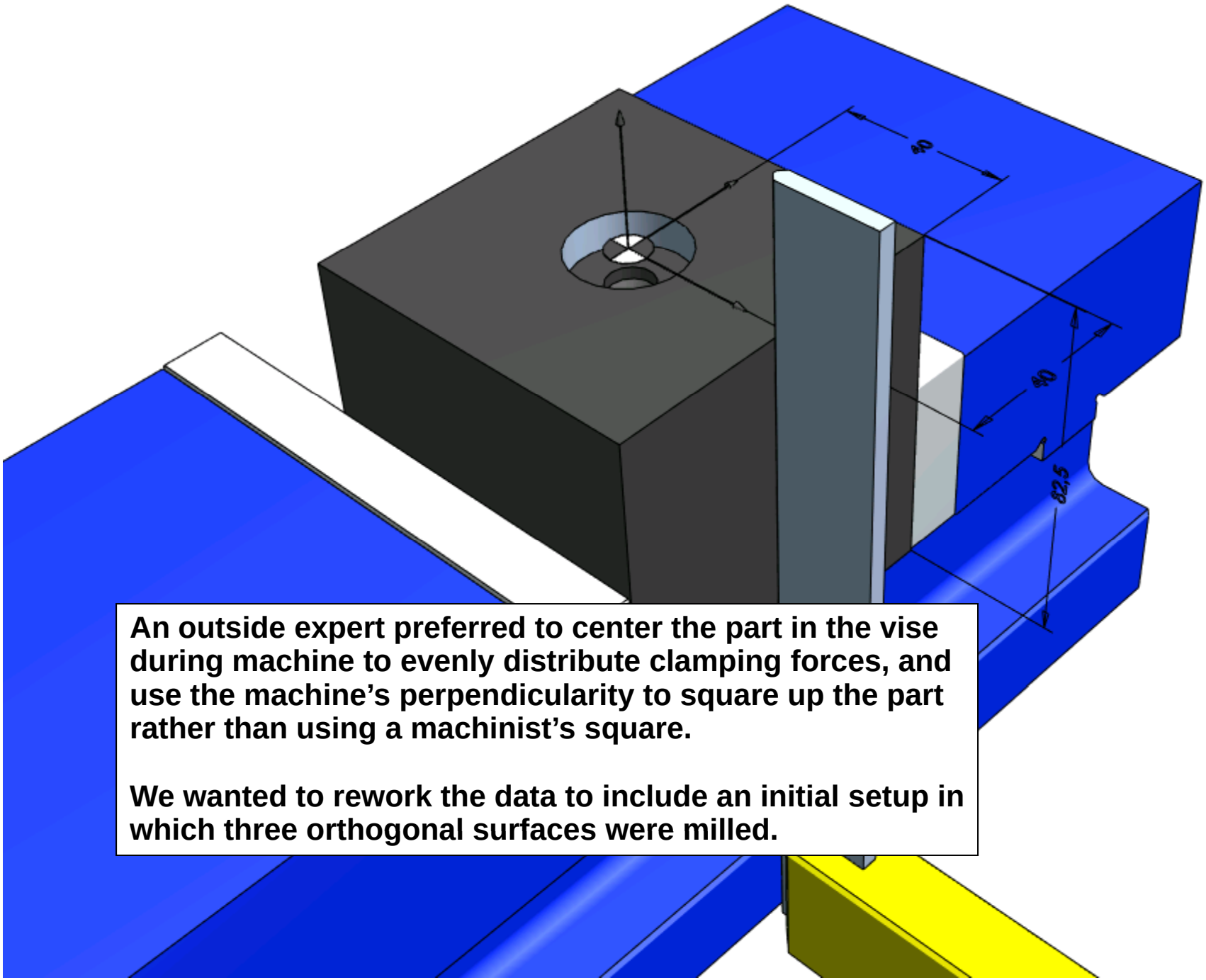



**Likewise, we can see the Z coordinate of the top of the part is 2.5 mm.**

**But the machinist typically wants to set the coordinates of faces that will be placed against locating stops, such as a parallel at the vise bottom, the vise back and a stop at the right face. These preferences vary by machinist.**

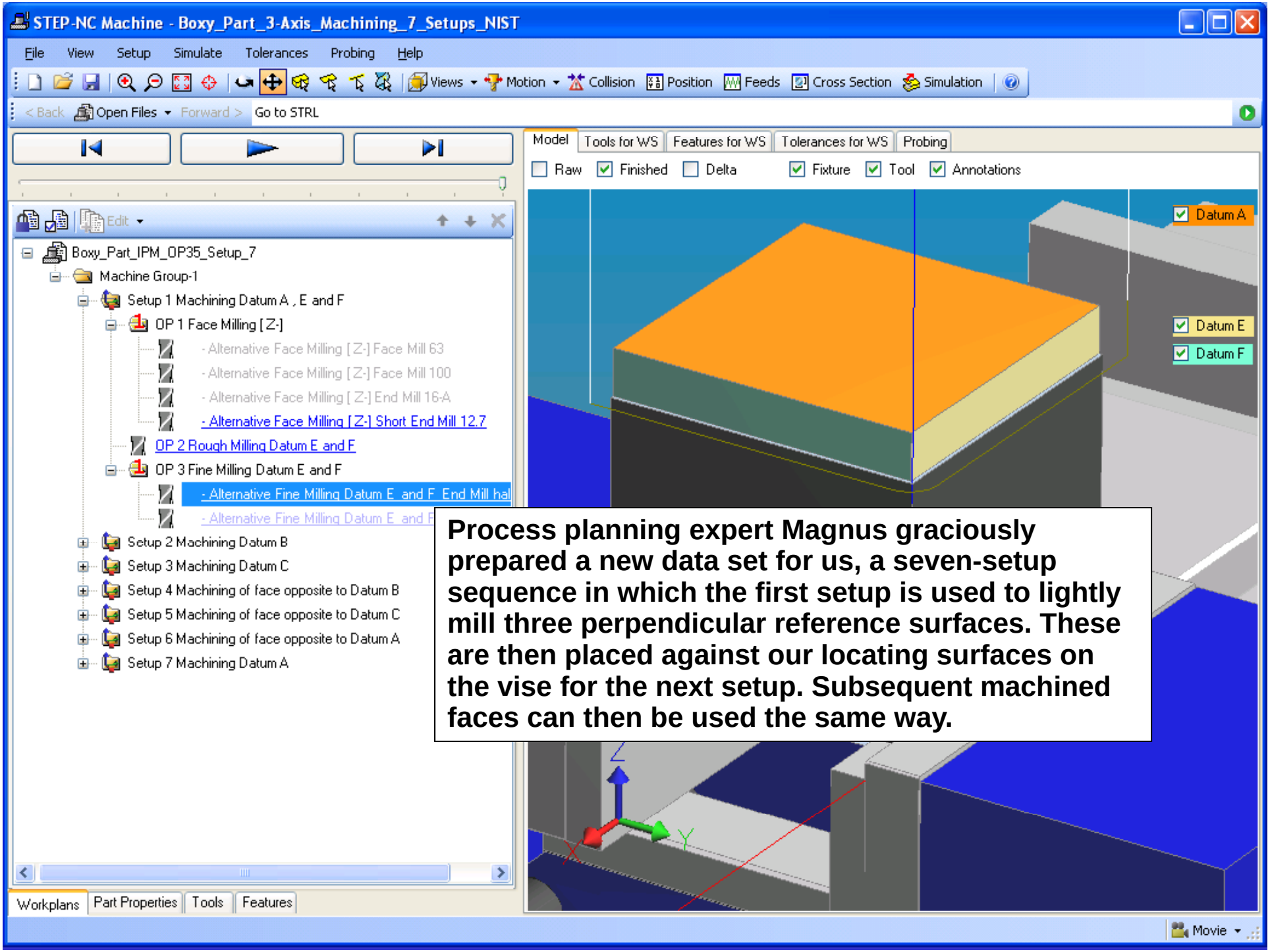
**In our case, we want the right face, back face and bottom face to be (0, 0, 0) in XYZ, respectively.**


STEP-NC Machine - Boxy_Part_3-Axis_Machining_8_Setups_v7
File View Setup Simulate Tolerances Probing Help
Views Motion Collision Position Feeds Cross Section Simulation
< Back Open Files Forward > Go to STRL
Edit
default workpiece
Machine Group-1
Setup 1 Machi
OP 1 Fac
Setup 2 Machi
Setup 3 Machi
Setup 4 Machi
Setup 5 Machi
Setup 6 Machi
Setup 7 Machi
Setup 8 Machi
Enable/Disable Items
Show NC Functions
Import
Optimize Feeds ...
Insert NC Function ...
Insert Workplan
Change to Selective
Edit Name ...
Edit STRL ...
Edit Setup Placement ...
Add Faces to All Workingsteps
Workpiece Properties ...
Machine Parameters ...
Model Tools for WS Features for WS Tolerances for WS Probing
Raw Finished Delta Fixture Tool Annotations
Datum A
Workplan Setup Placement
Location -42.5 -42.5 82.5
Z-axis 0 0 -1
X-axis -1 0 0
Modify
Cancel
Delete
In the new version of ST-Machine, we can do this, and have set up offsets so the locating surfaces at each setup are (0, 0, 0) as we like. This means we only need to set up one work coordinate system, saving lots of time.
Workplans Part Properties Tools Features
Selected Face #526 at (42.5, 11.433, -37.7761)
Movie

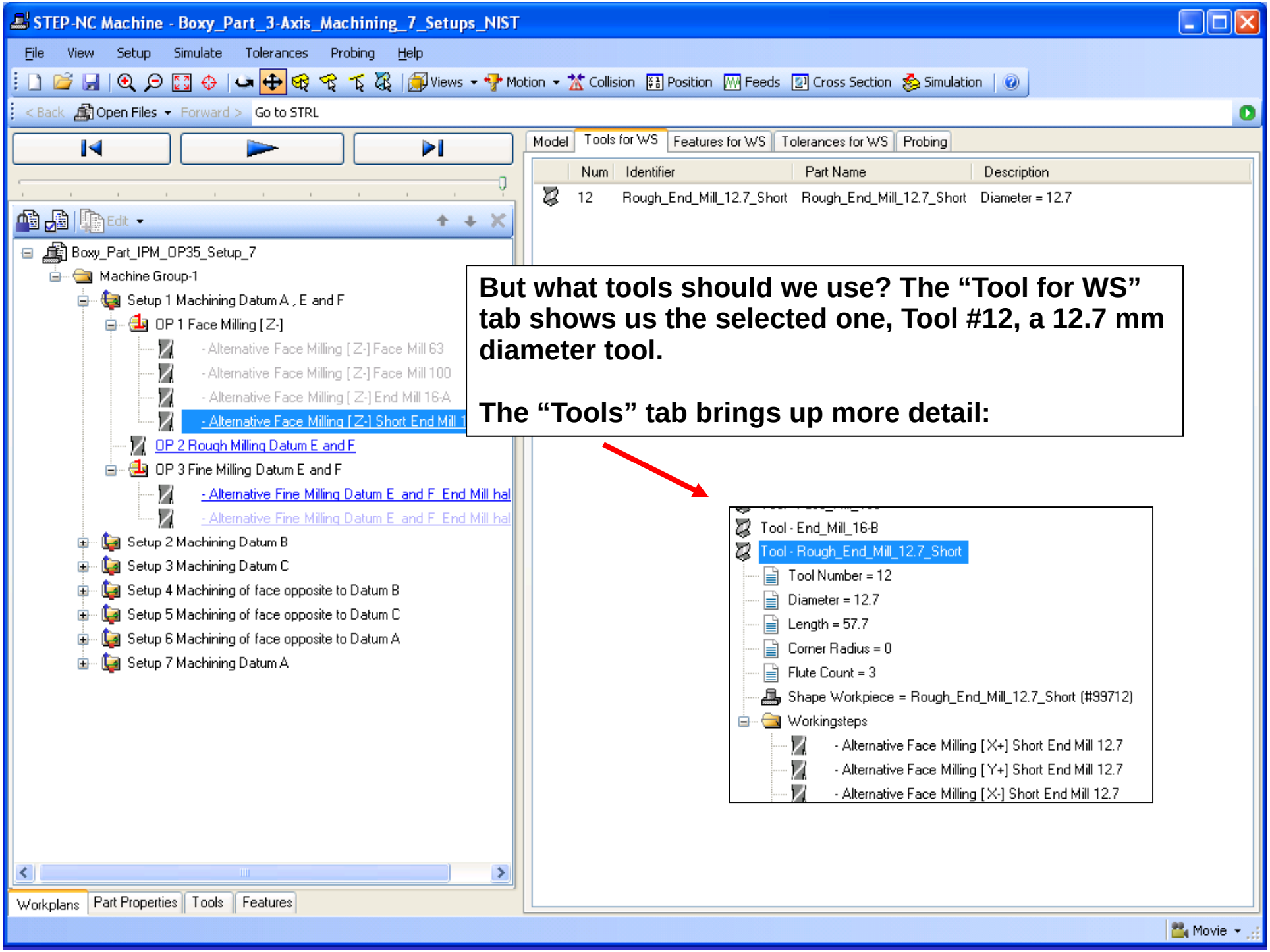



**An outside expert preferred to center the part in the vise during machine to evenly distribute clamping forces, and use the machine's perpendicularity to square up the part rather than using a machinist's square.**

**We wanted to rework the data to include an initial setup in which three orthogonal surfaces were milled.**

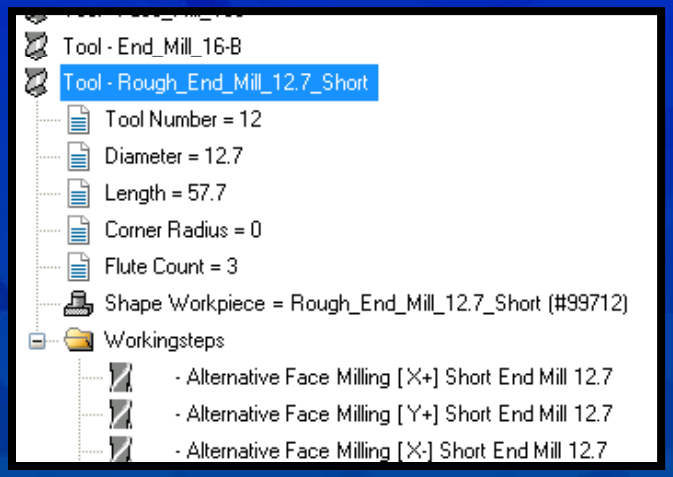



**Process planning expert Magnus graciously prepared a new data set for us, a seven-setup sequence in which the first setup is used to lightly mill three perpendicular reference surfaces. These are then placed against our locating surfaces on the vise for the next setup. Subsequent machined faces can then be used the same way.**


STEP-NC Machine - Boxy_Part_3-Axis_Machining_7_Setups_NIST
File View Setup Simulate Tolerances Probing Help
Views Motion Collision Position Feeds Cross Section Simulation
< Back Open Files Forward > Go to STRL
Edit
Boxy_Part_IPM_OP35_Setup_7
Machine Group-1
Setup 1 Machining Datum A , E and F
OP 1 Face Milling [Z-]
- Alternative Face Milling [Z-] Face Mill 63
- Alternative Face Milling [Z-] Face Mill 100
- Alternative Face Milling [Z-] End Mill 16-A
- Alternative Face Milling [Z-] Short End Mill 1
OP 2 Rough Milling Datum E and F
OP 3 Fine Milling Datum E and F
- Alternative Fine Milling Datum E and F End Mill hal
- Alternative Fine Milling Datum E and F End Mill hal
Setup 2 Machining Datum B
Setup 3 Machining Datum C
Setup 4 Machining of face opposite to Datum B
Setup 5 Machining of face opposite to Datum C
Setup 6 Machining of face opposite to Datum A
Setup 7 Machining Datum A
Workplans Part Properties Tools Features
Model Tools for WS Features for WS Tolerances for WS Probing
Num Identifier Part Name Description
12 Rough_End_Mill_12.7_Short Rough_End_Mill_12.7_Short Diameter = 12.7
But what tools should we use? The “Tool for WS” tab shows us the selected one, Tool #12, a 12.7 mm diameter tool.
The “Tools” tab brings up more detail:
Tool - End_Mill_16-B
Tool - Rough_End_Mill_12.7_Short
Tool Number = 12
Diameter = 12.7
Length = 57.7
Corner Radius = 0
Flute Count = 3
Shape Workpiece = Rough_End_Mill_12.7_Short (#99712)
Workingsteps
- Alternative Face Milling [X+] Short End Mill 12.7
- Alternative Face Milling [Y+] Short End Mill 12.7
- Alternative Face Milling [X-] Short End Mill 12.7
Movie

# Tool Requirements

- But we need to know what flexibility we have in choosing tools.
  - Do we need exactly a 57.7 mm cutting length?
  - Can it be shorter? How much shorter?
  - Can it be a bit narrower? Will 12.5 mm be OK?
- This brought up the subject of tool requirements that we will be discussing later in this meeting.




```
SCHEMA tool_requirements_schema;

TYPE prescription_type = ENUMERATION OF (
     required_to_prevent_collision,
     required_to_remove_material,
     required_by_customer,
     recommended_to_mimimize_deflection,
     recommended_to_minimize_vibration,
     recommended_from_best_practices,
     recommended_from_testing
);
END_TYPE;
```

# What we would do differently

- We would do this part in six setups
  - the first of seven setups was clamped high to allow for the machining of large reference surfaces on the right and back sides
  - since it was clamped high, we chose not to do pocketing operations that might pop the part out
  - we really only need small reference surfaces, so we can clamp more fully in the first setup and finish up all the operations at once
- We would do spiral entry cuts for the pockets, using a face-cutting end mill for clean cutting

# What we want added

- We generate NC code at the end, since our Siemens 840D doesn't support STEP-NC natively
- The machinist looks at the NC code during operation for reminders about what to do
- Reminders, via comments, should include:
  - how the part is to be oriented
  - what reference surfaces should be used and at what coordinate values
  - what tool is being used and the chip loading
  - what operation will be done
  - how long this will take

**Ideally, the STEP-NC process plan should come out of the CAM system with all the information the machinist needs to set up and machine the part.**

**No sticky notes required!**