



DIN 902
Noppenerkennungsschraube (M10)
für blinde Mitarbeiter



DIN 903
Bohrsenkgewindeschneidschraube



DIN 879
Für Löcher die auf
Der falschen Seite
angesenkt wurden



DIN 880
Schrauben in Feld-
stecher Form für
doppelt gebohrte
Löcher



DIN 904
Rohrzangen-
kopfschraube



DIN 905
Zwillingschraube

Schraubenkopf auswechselbar



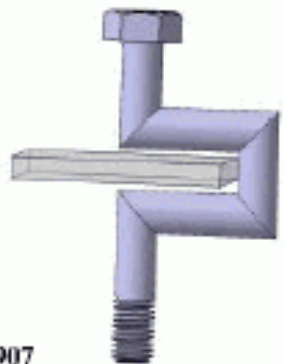
DIN 906
Vario-
mogelschraube
zum Vortäuschen stabiler
mechanischer Verbindungen

CAD-Einführung

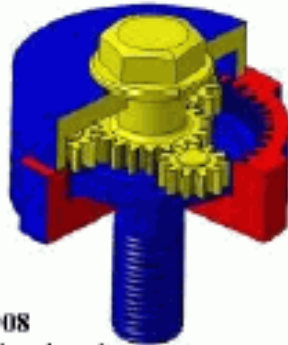
©FabLab Lübeck



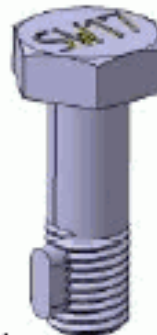
DIN 883
Sonderschraube zur
Verringerung der Montagezeit



DIN 907
Ausweichschraube



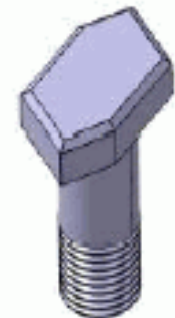
DIN 908
Getriebeschraube
nur in Verwendung mit
Getriebeschraubenschlüssel



DIN 909
Sonderschraube
mit Passfeder als
Ausdrehsicherung



DIN 885
Für wechselnde Winkelfehler



DIN 886
Für Schlüsselweite
13, 17 und 19

Konstruieren aber womit?



AUTODESK®
FUSION 360™



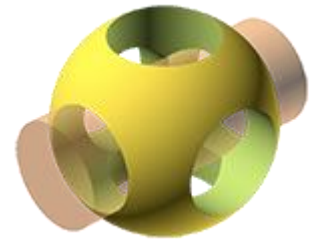
Moment of
Inspiration



FreeCAD



AUTODESK
INVENTOR



 **SOLIDWORKS**

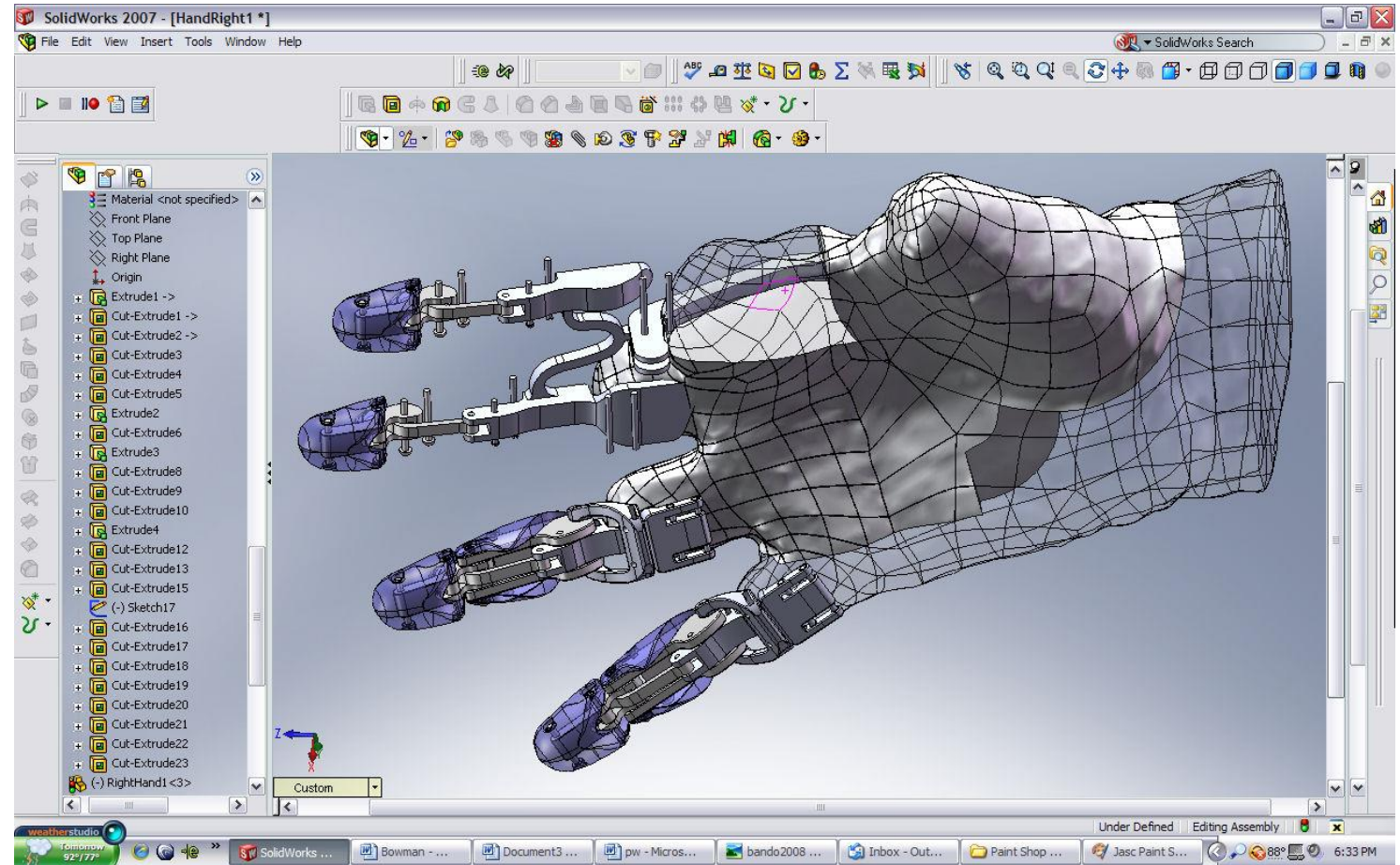


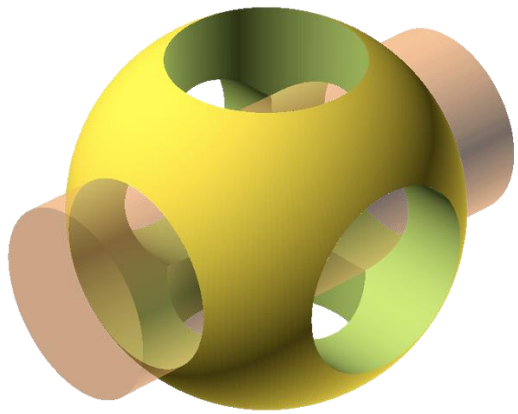
SketchUp

SOLIDWORKS



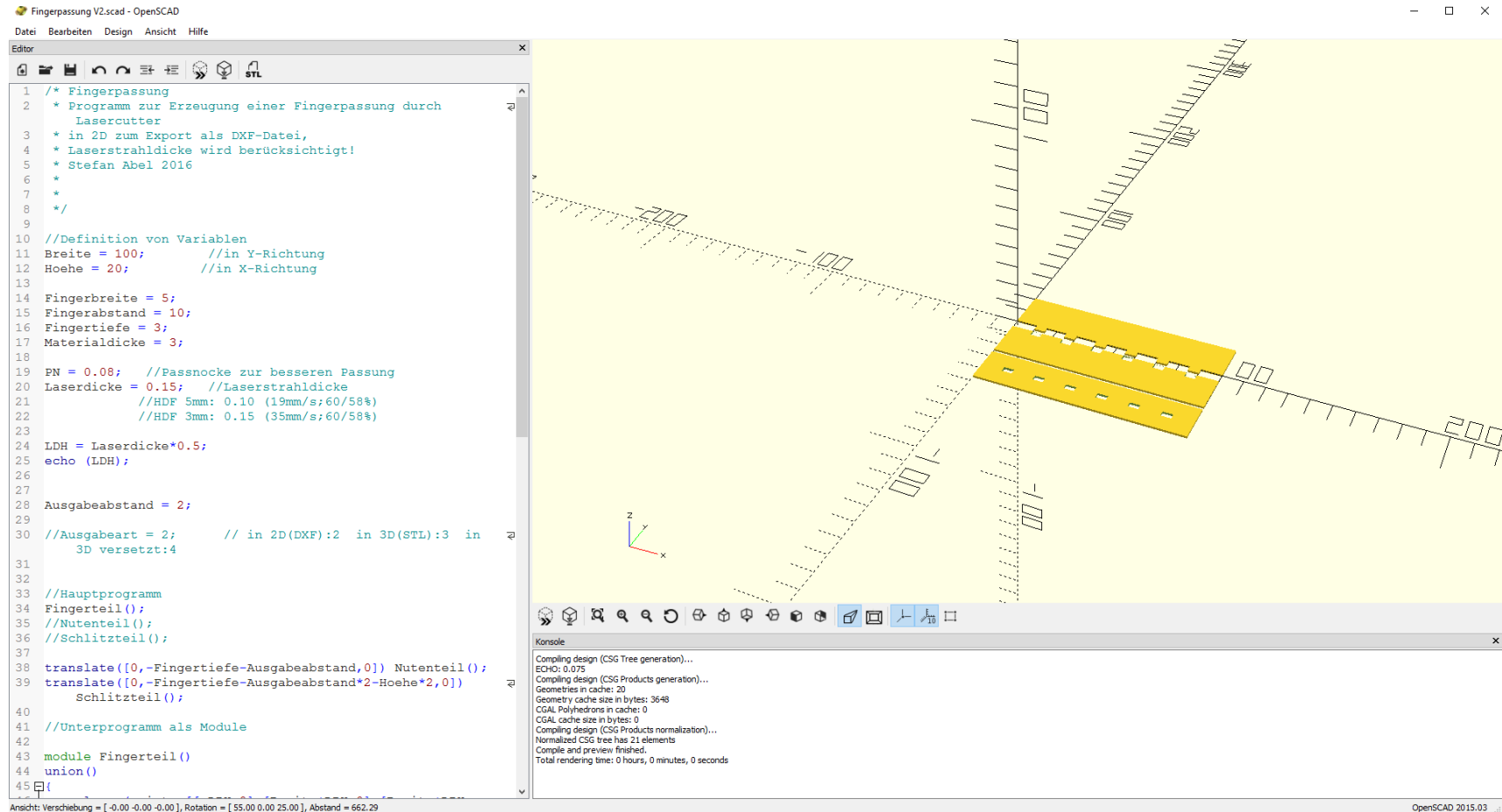
\$150 | €135 | £100





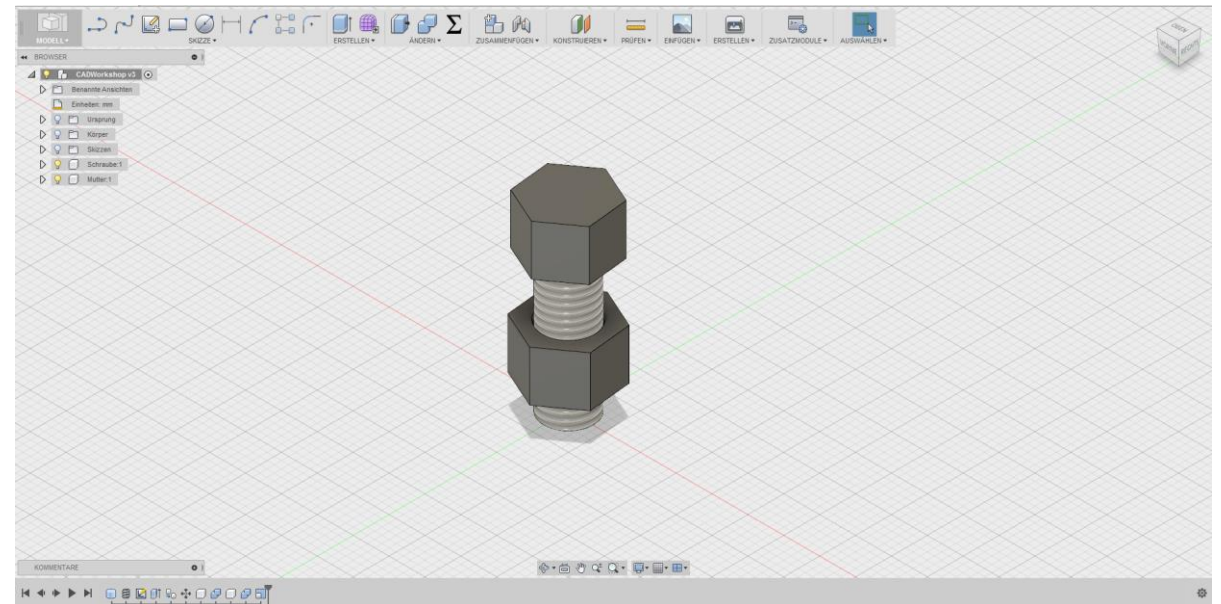
OpenScad

Mehr dazu in
unserem OpenSCAD-
Workshop



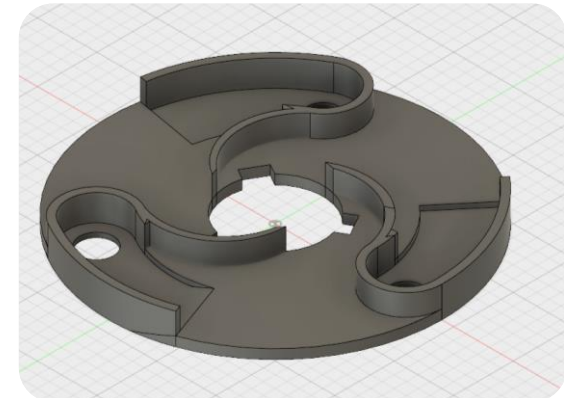
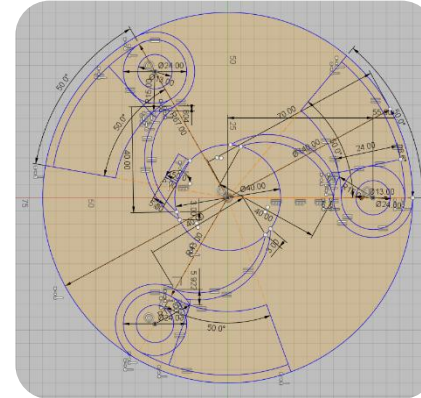
 AUTODESK®
FUSION 360™

- Freie Lizenz für nicht kommerzielle Nutzer und Studenten
- Cloud basierte Lösung
- Keine eingeschränkten Exportfunktionalitäten
- Intuitiver aber professioneller Einstieg möglich
- Viele Internet Tutorials und aktive Community



Konstruieren aber wie?

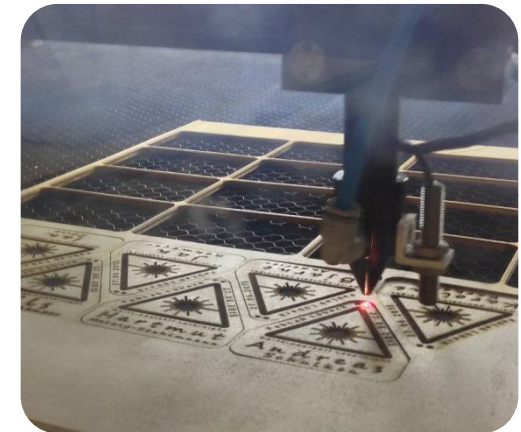
- Von der Zeichnung zum Körper
 - Einfache geometrische Objekte kombinieren
 - Abstände festlegen
 - Beziehungen festlegen
 - Zeichnung einfach ins 3D extrudieren



- Konstruieren mit einfachen Mathematischen Operationen
 - Körper subtrahieren
 - Körper addieren
 - Körper teilen
 - Schnittmengen bestimmen

Fertigungsgerechte Konstruktion

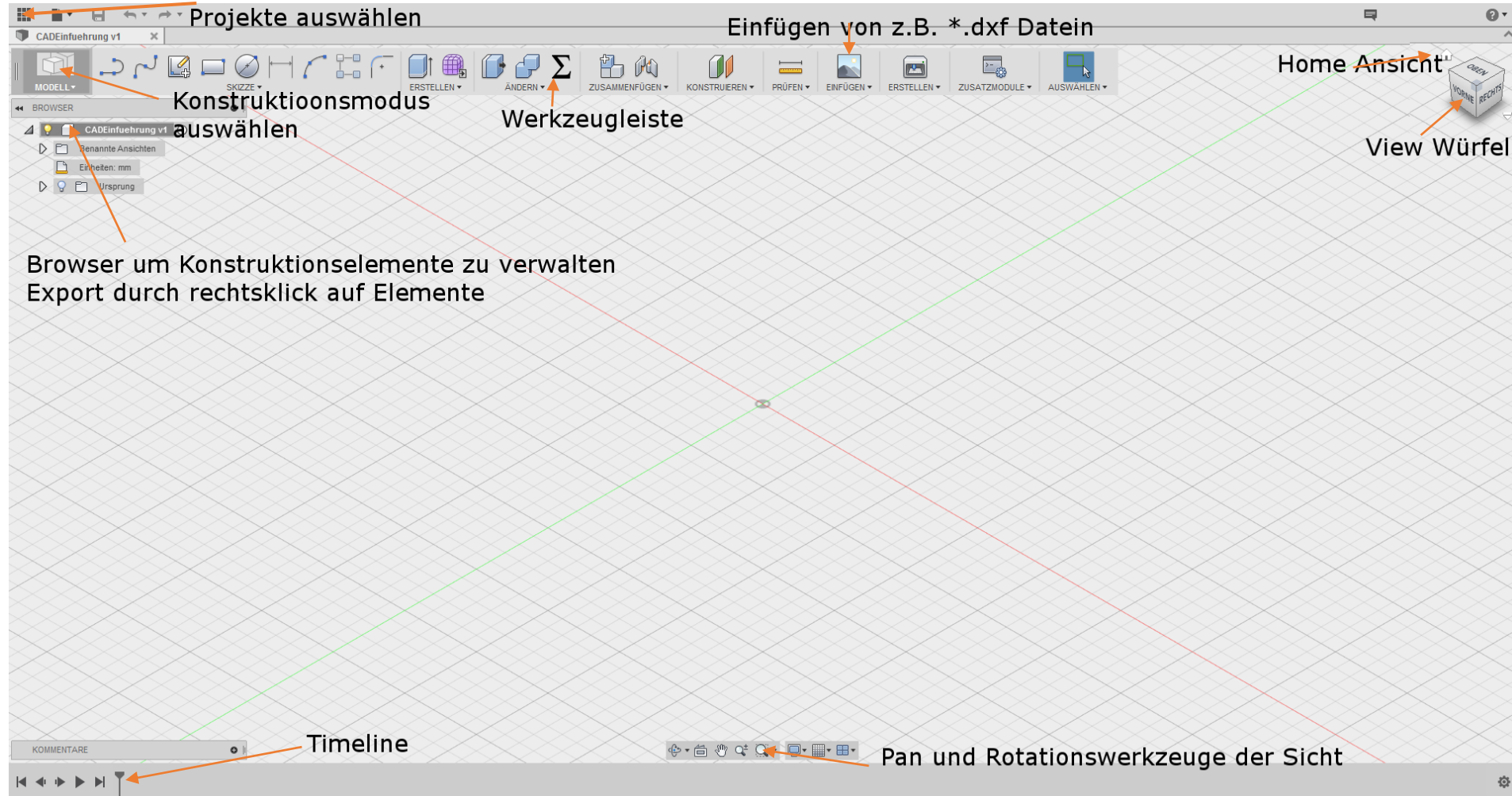
- 3D-Drucker
 - Exportformat: *.stl
 - Überhänge beachten
 - Wandstärke beachten
 - Ausrichtung beachten
 - Solides(Wasserdichtes) Modell erzeugen
- Lasercutter
 - Exportformat: *.dxf
 - Auf volldefinierte Beziehungen achten (Längenangaben etc.)
 - Immer in einer Skizze und nicht in mehreren arbeiten
 - Geschlossene Konturen erzeugen
- CNC-Fräse
 - Exportformat: *.stp
 - Materialdicke beachten
 - Ausrichtung beachten
 - Etc.



Los geht's

1. Fusion360 Übersicht
2. Fusion360 Zeichnung erstellen
 1. Wir konstruieren einen Smiley
3. Fusion360 im Dreidimensionalen
 1. Wir konstruieren eine Schraube

Fusion360 Übersicht



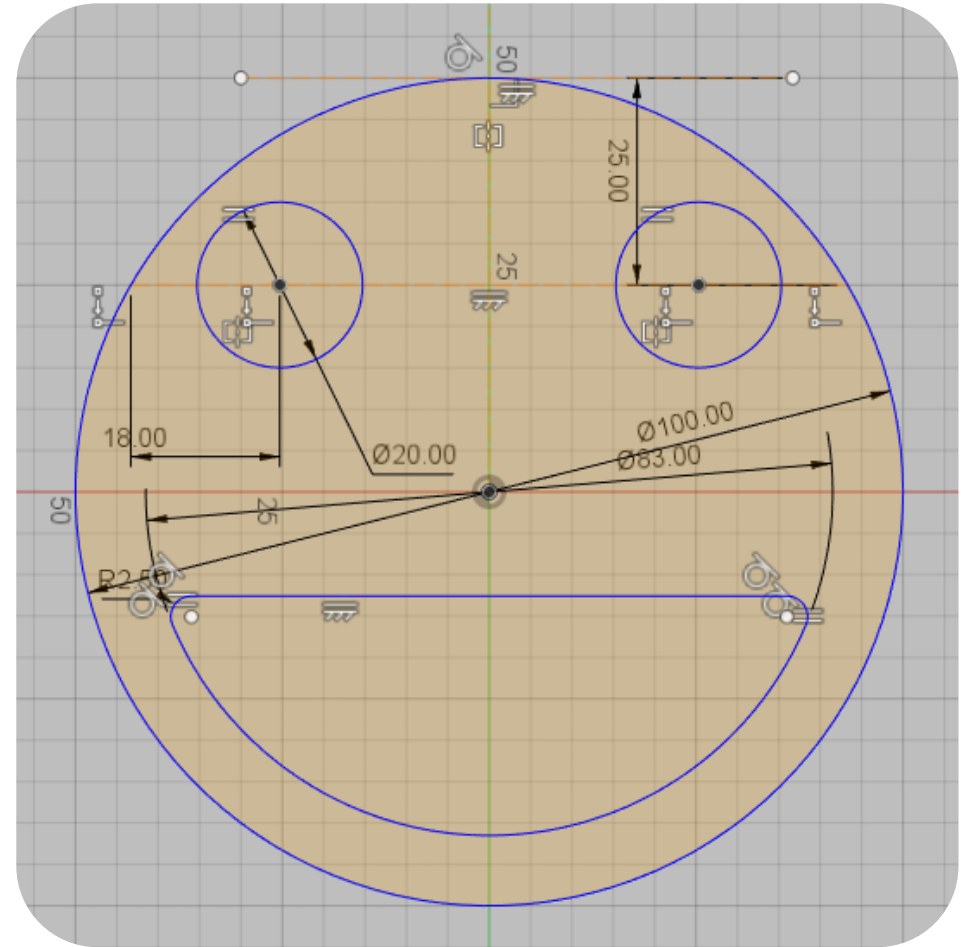
Fusion360 Zeichnung erstellen

Wir konstruieren einen Smiley

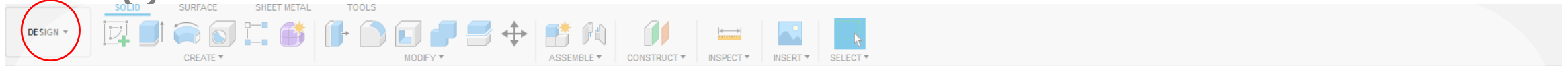
- Zeichenebene Festlegen
- Außenkontur zeichnen
- Augen zeichnen
- Mund zeichnen
- Zeichnung exportieren

Was lernen wir?

- Mit Zeichenwerkzeugen umgehen
- Konstruktionslinien erstellen
- Constrains setzen
- Mit Sketchpalette arbeiten
- *.dxf Datei exportieren



Allgemeines



BROWSER

- Wir arbeiten immer im Design-Modus

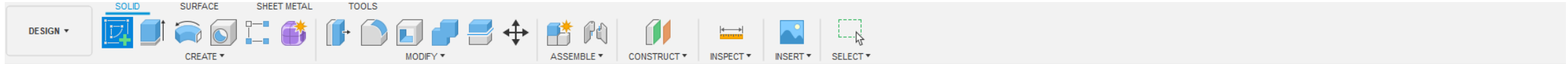
- Für eine Zeichnung immer nur in einer Skizze arbeiten
- Aktive Werkzeuge werden rechts unter dem Mauszeiger angezeigt
- Um Werkzeug zu deaktivieren muss `ESC` gedrückt werden
- Über den Würfel rechts können Ansichten gewechselt werden
- Durch Rechtsklick auf den Würfel, zurück in die Default-Ansicht



COMMENTS



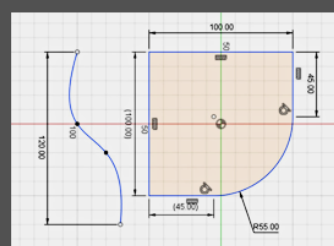
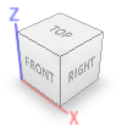
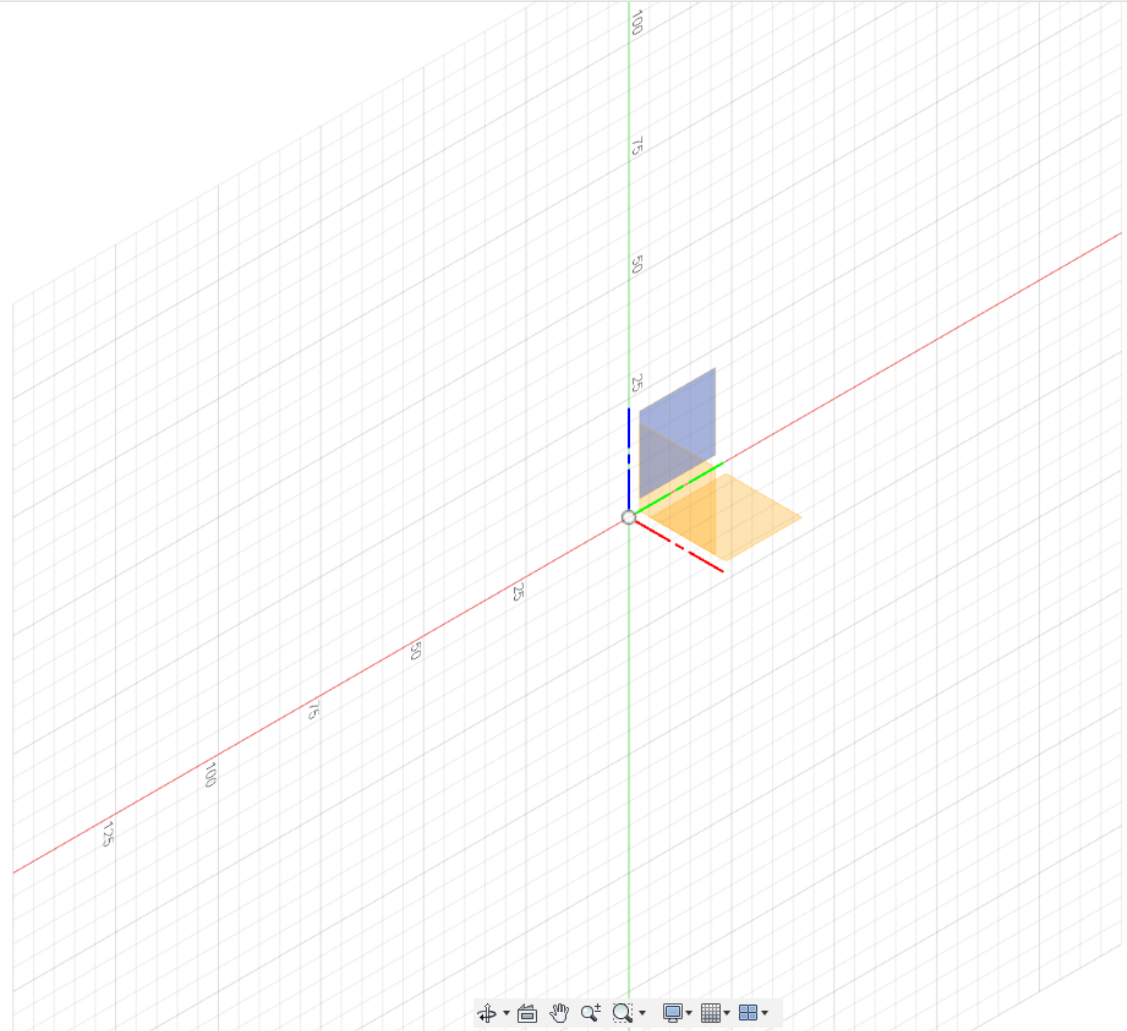
Skizzenebene setzen



Create Sketch

Use Sketch mode to create basic geometry profiles that define the design. First select a construction plane. Then create lines, arcs, or points and use dimensions to constrain the boundaries. Sketches are used with 3D creation commands such as Extrude.

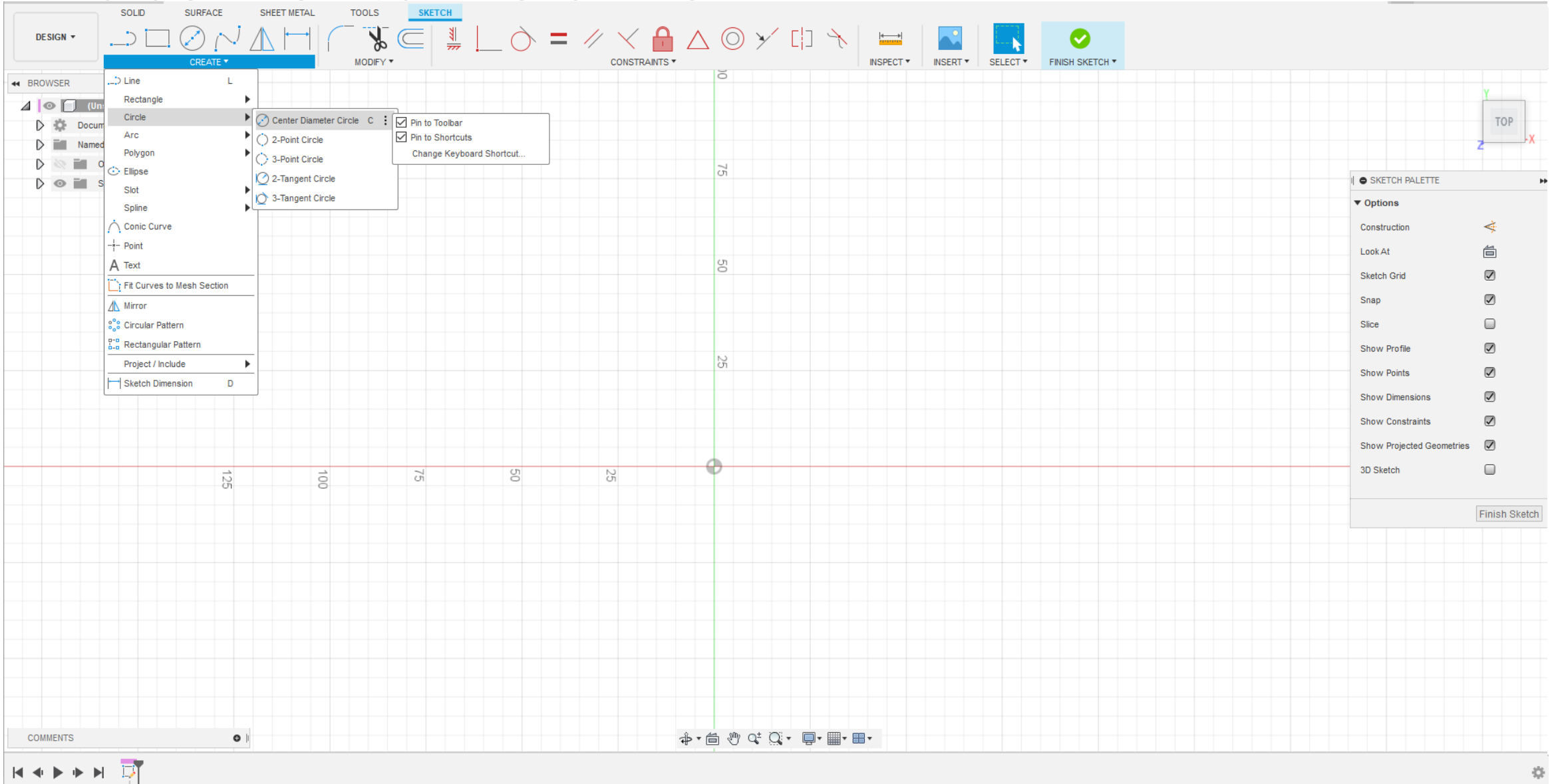
After creating the boundary, select Finish Sketch to exit Sketch mode.

COMMENTS



Außenkontur zeichnen



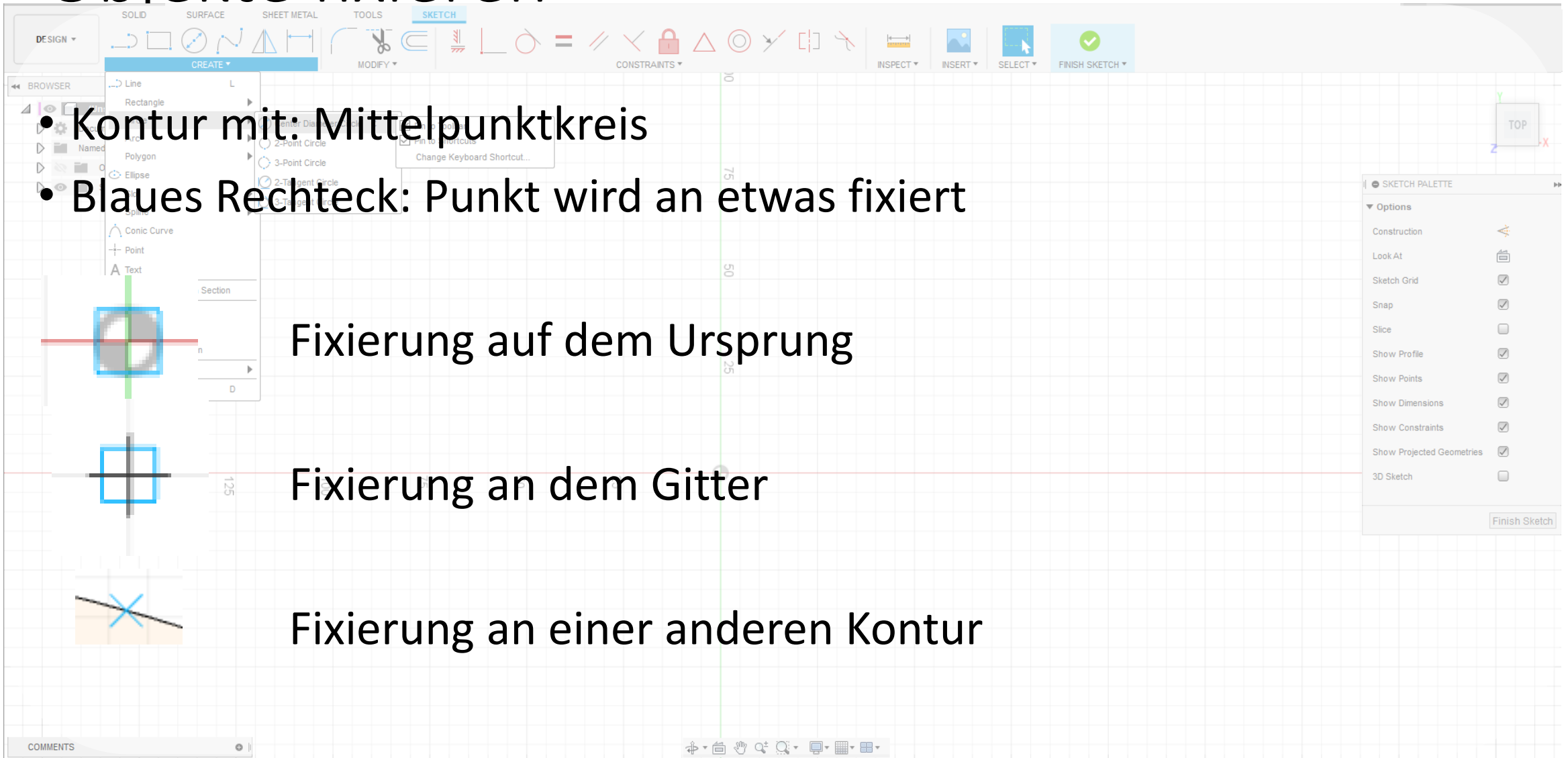
The image shows a CAD software interface with a grid. A vertical green line is drawn at the 75 mark on the horizontal axis. A horizontal red line is drawn at the 50 mark on the vertical axis. A small circle is centered at the intersection of these two lines. The 'CREATE' menu is open, showing options for drawing a circle. A sub-menu for 'Circle' is also open, listing various circle creation methods: Center Diameter Circle (C), 2-Point Circle, 3-Point Circle, 2-Tangent Circle, and 3-Tangent Circle. A tooltip for the 'Center Diameter Circle' option is visible, containing the following text:

- Pin to Toolbar
- Pin to Shortcuts
- Change Keyboard Shortcut...

The 'SKETCH PALETTE' is open on the right side, showing various options for sketching, such as Construction, Look At, Sketch Grid, Snap, Slice, Show Profile, Show Points, Show Dimensions, Show Constraints, Show Projected Geometries, and 3D Sketch. The 'Finish Sketch' button is visible at the bottom of the palette.

Objekte fixieren

- Kontur mit Mittelpunktkreis
- Blaues Rechteck: Punkt wird an etwas fixiert

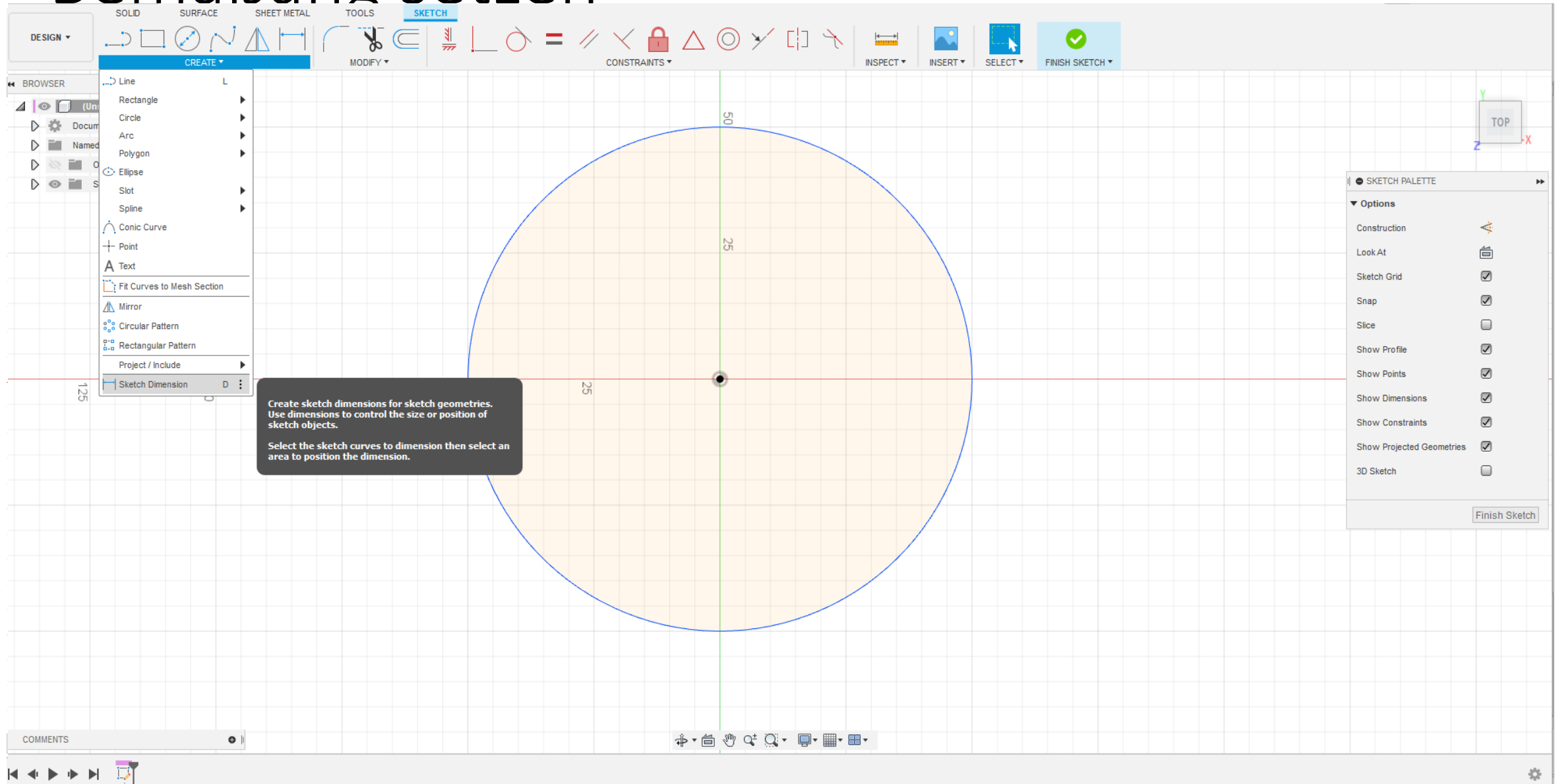


Fixierung auf dem Ursprung

Fixierung an dem Gitter

Fixierung an einer anderen Kontur

Bemaßung setzen



The screenshot shows a CAD software interface with a sketch of a circle on a grid. The circle is centered at the origin of a coordinate system. A vertical dimension line is drawn from the center to the top edge of the circle, labeled '50'. A horizontal dimension line is drawn from the center to the left edge of the circle, labeled '25'. The 'CREATE' menu is open, and the 'Sketch Dimension' tool is highlighted. A tooltip is visible over the 'Sketch Dimension' tool, providing instructions on how to use it.

CREATE

- Line
- Rectangle
- Circle
- Arc
- Polygon
- Ellipse
- Slot
- Spline
- Conic Curve
- Point
- Text
- Fit Curves to Mesh Section
- Mirror
- Circular Pattern
- Rectangular Pattern
- Project / Include
- Sketch Dimension

MODIFY

CONSTRAINTS

INSPECT

INSERT

SELECT

FINISH SKETCH

SKETCH PALETTE

- Options
 - Construction
 - Look At
 - Sketch Grid
 - Snap
 - Slice
 - Show Profile
 - Show Points
 - Show Dimensions
 - Show Constraints
 - Show Projected Geometries
 - 3D Sketch

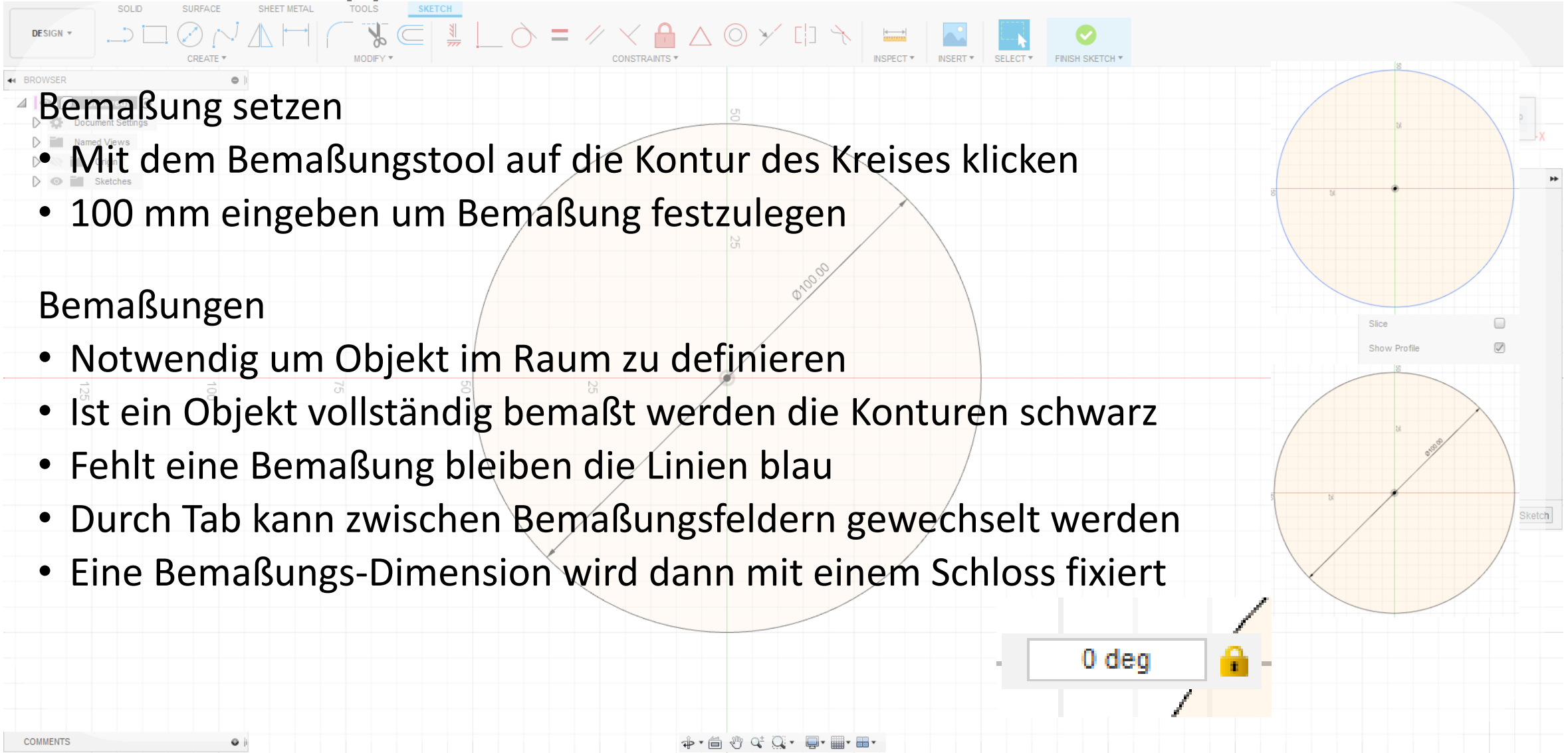
Finish Sketch

TOOLTIP:

Create sketch dimensions for sketch geometries. Use dimensions to control the size or position of sketch objects.

Select the sketch curves to dimension then select an area to position the dimension.

Bemaßung setzen



Bemaßung setzen

- Mit dem Bemaßungstool auf die Kontur des Kreises klicken
- 100 mm eingeben um Bemaßung festzulegen

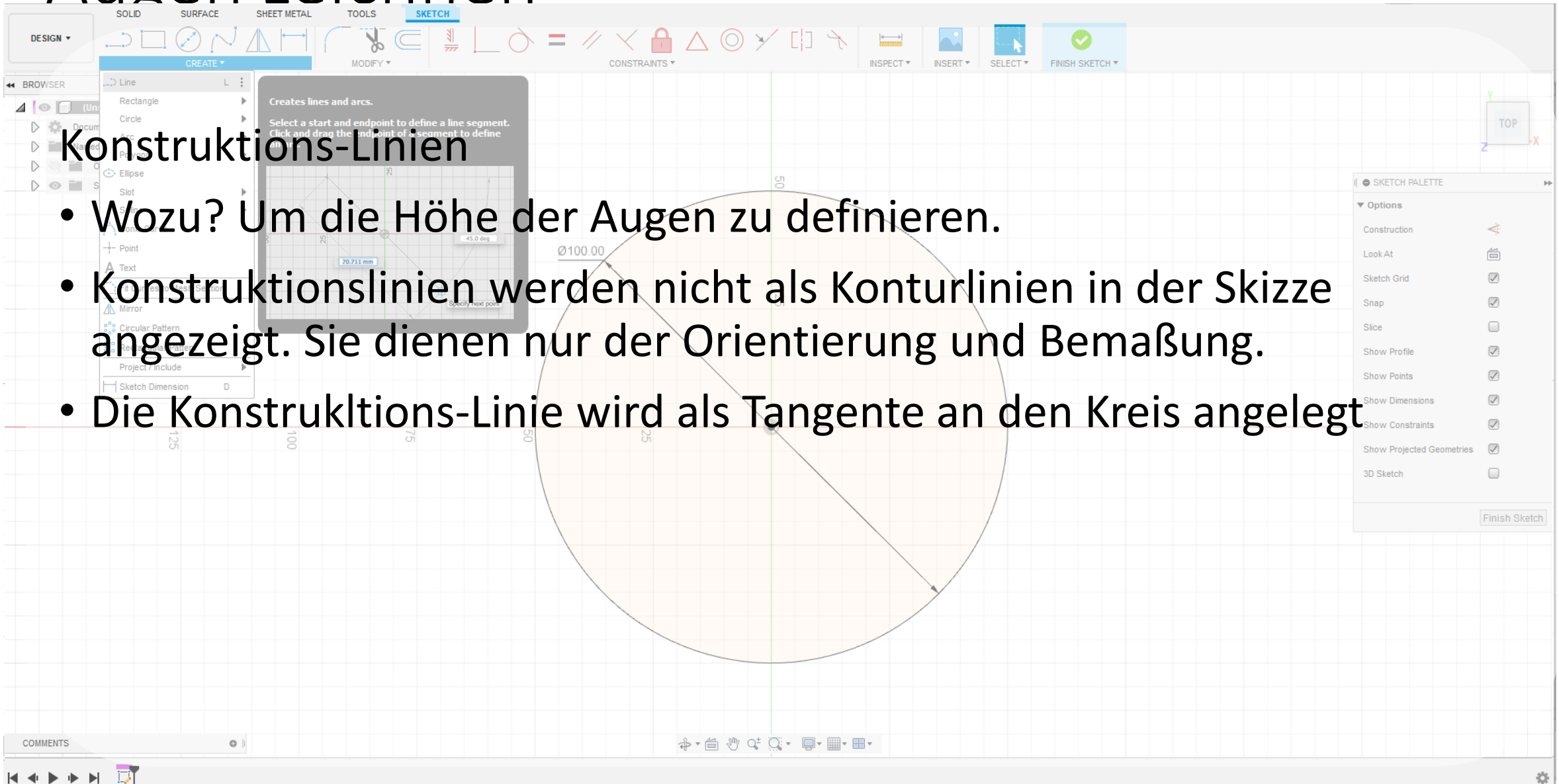
Bemaßungen

- Notwendig um Objekt im Raum zu definieren
- Ist ein Objekt vollständig bemaßt werden die Konturen schwarz
- Fehlt eine Bemaßung bleiben die Linien blau
- Durch Tab kann zwischen Bemaßungsfeldern gewechselt werden
- Eine Bemaßungs-Dimension wird dann mit einem Schloss fixiert

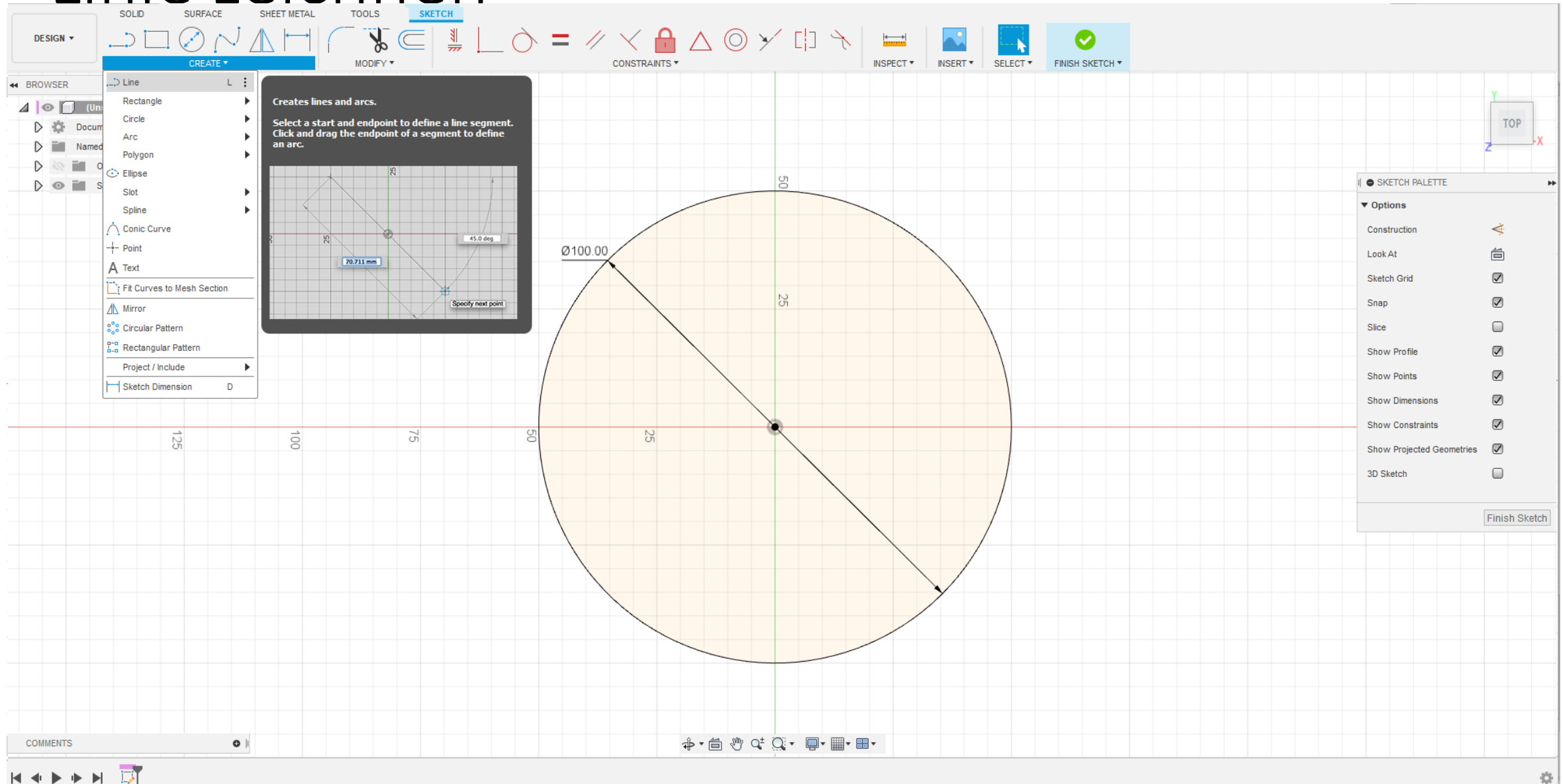
Augen zeichnen

Konstruktions-Linien

- Wozu? Um die Höhe der Augen zu definieren.
- Konstruktionslinien werden nicht als Konturlinien in der Skizze angezeigt. Sie dienen nur der Orientierung und Bemaßung.
- Die Konstruktions-Linie wird als Tangente an den Kreis angelegt



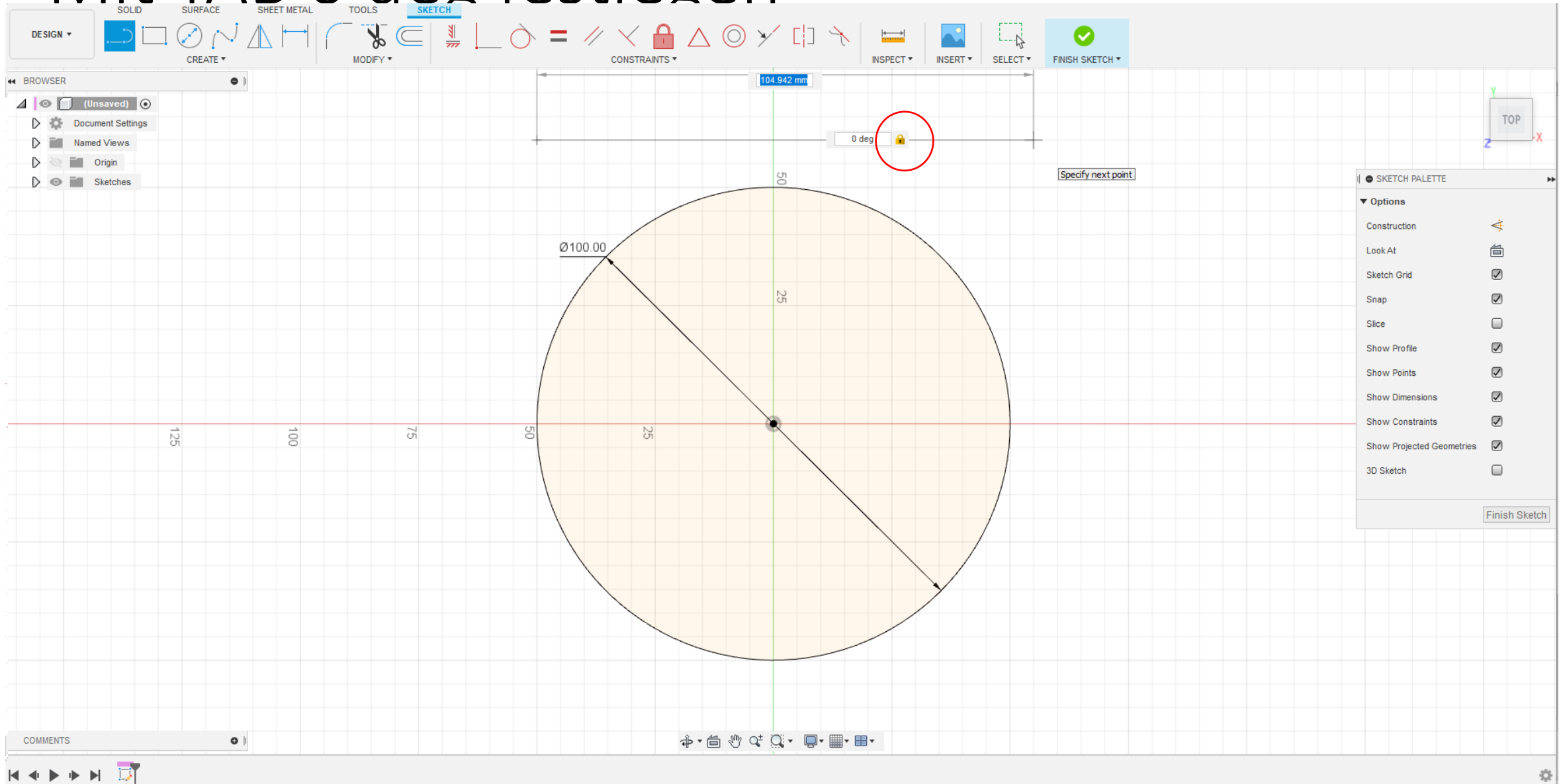
Linie zeichnen



The image shows a CAD software interface with the following components:

- Top Toolbar:** Includes tabs for SOLID, SURFACE, SHEET METAL, TOOLS, and SKETCH. The SKETCH tab is active, showing icons for line creation, constraints, inspection, insertion, selection, and finishing the sketch.
- Left Sidebar:** Contains a BROWSER and a 'CREATE' menu with options like Line, Rectangle, Circle, Arc, Polygon, Ellipse, Slot, Spline, Conic Curve, Point, Text, Fit Curves to Mesh Section, Mirror, Circular Pattern, Rectangular Pattern, Project / Include, and Sketch Dimension.
- Central Workspace:** A grid-based workspace showing a circle with a diameter of $\varnothing 100.00$ and a line drawn from the center at a 45.0 degree angle. Dimensions of 25 and 50 are visible along the axes.
- Right Sidebar:** A 'SKETCH PALETTE' with 'Options' such as Construction, Look At, Sketch Grid, Snap, Slice, Show Profile, Show Points, Show Dimensions, Show Constraints, Show Projected Geometries, and 3D Sketch. A 'Finish Sketch' button is at the bottom.
- Bottom:** A 'COMMENTS' section and a navigation toolbar.

Mit TAB 0 deg festlegen

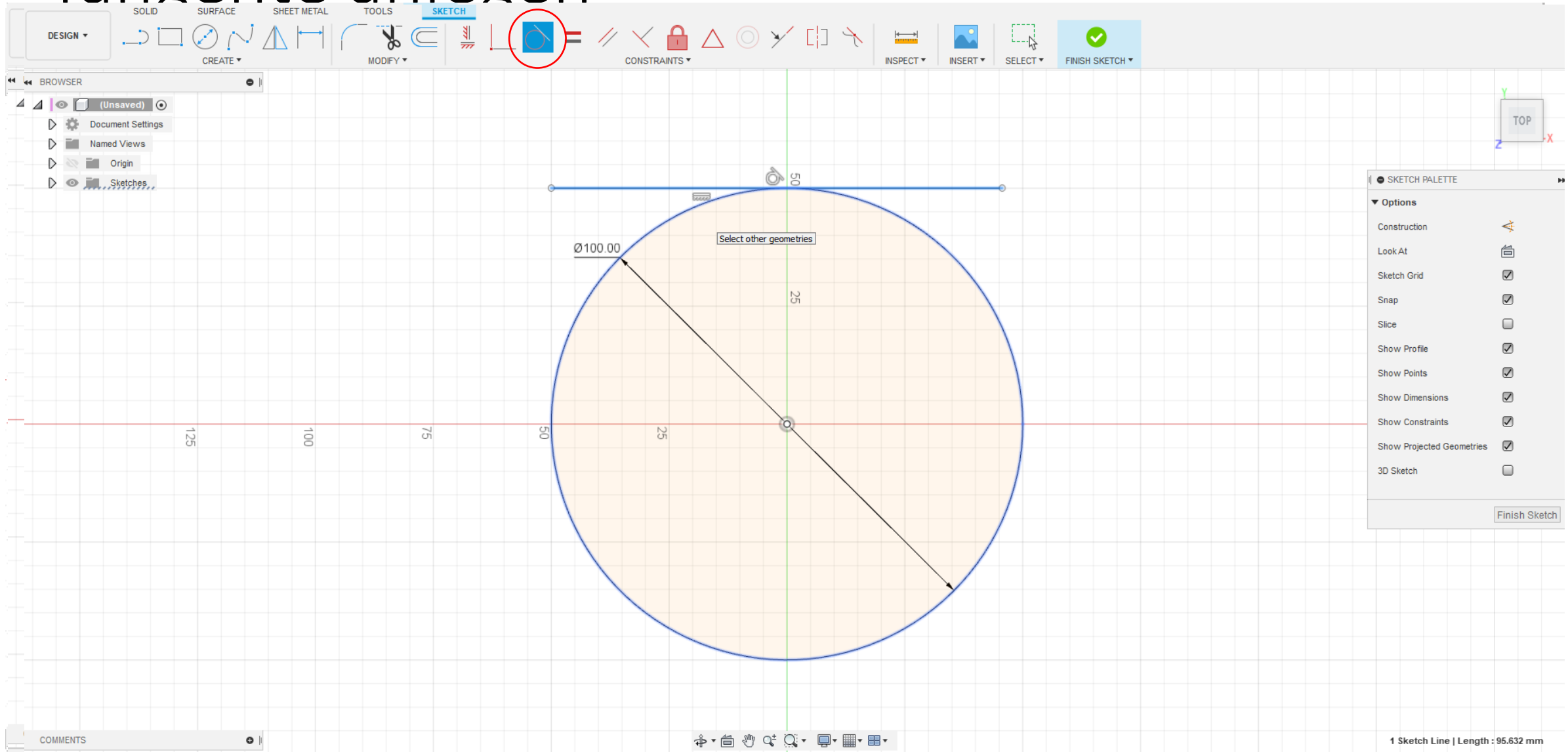


Constraints



- Constraints sind nützlich um Konturen zueinander zu definieren.
- Sie sind mit die wichtigsten Werkzeuge im Sketch Bereich

Tangente anlegen



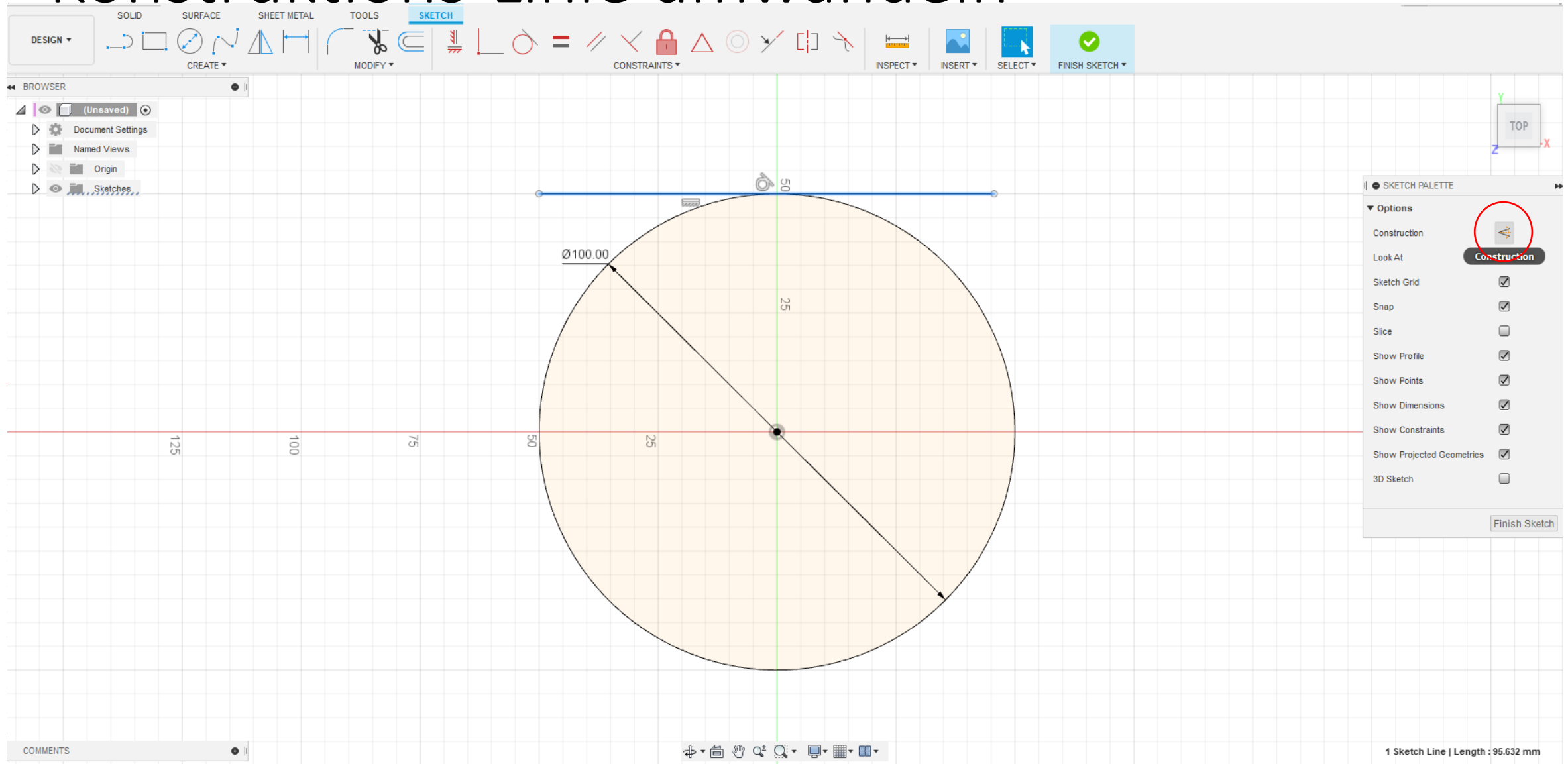
The image shows a CAD software interface with a sketch of a circle and a horizontal line tangent to its top. The circle has a diameter of $\varnothing 100.00$. The horizontal line is positioned 25 units above the top of the circle. The sketch is on a grid with dimensions marked along the axes: 125, 100, 75, 50, and 25.

SKETCH PALETTE Options:

- Construction:
- Look At:
- Sketch Grid:
- Snap:
- Slice:
- Show Profile:
- Show Points:
- Show Dimensions:
- Show Constraints:
- Show Projected Geometries:
- 3D Sketch:

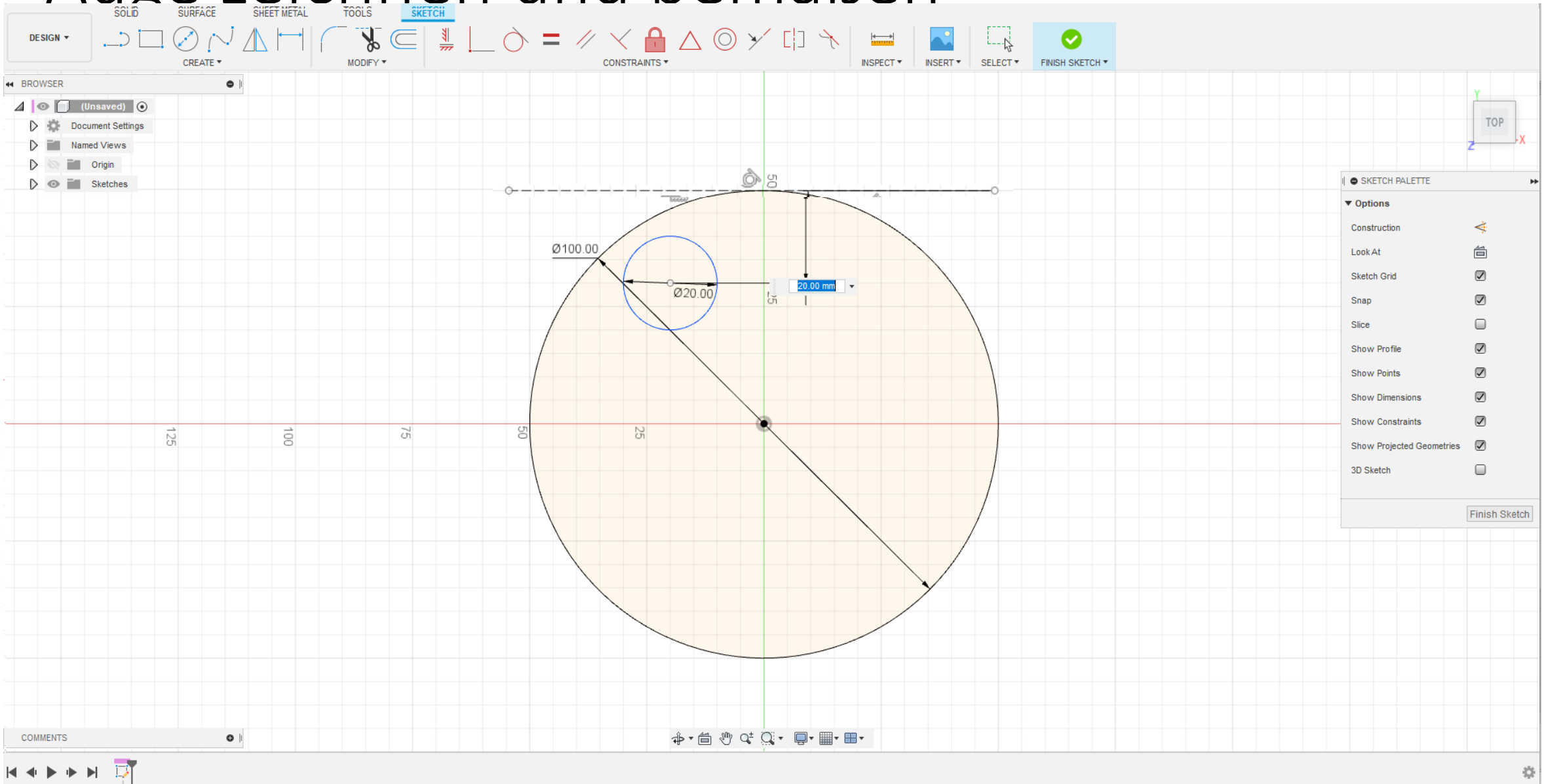
Status Bar: 1 Sketch Line | Length : 95.632 mm

Konstruktions-Linie umwandeln



The image shows a CAD software interface in 'SKETCH' mode. A circle with a diameter of 100.00 is centered on a horizontal red axis. A horizontal blue construction line is drawn above the circle, with a dimension of 50 from the vertical green axis to its center. The sketch palette on the right has the 'Construction' icon circled in red. The status bar at the bottom right indicates '1 Sketch Line | Length : 95.632 mm'.

Auge zeichnen und bemaßen

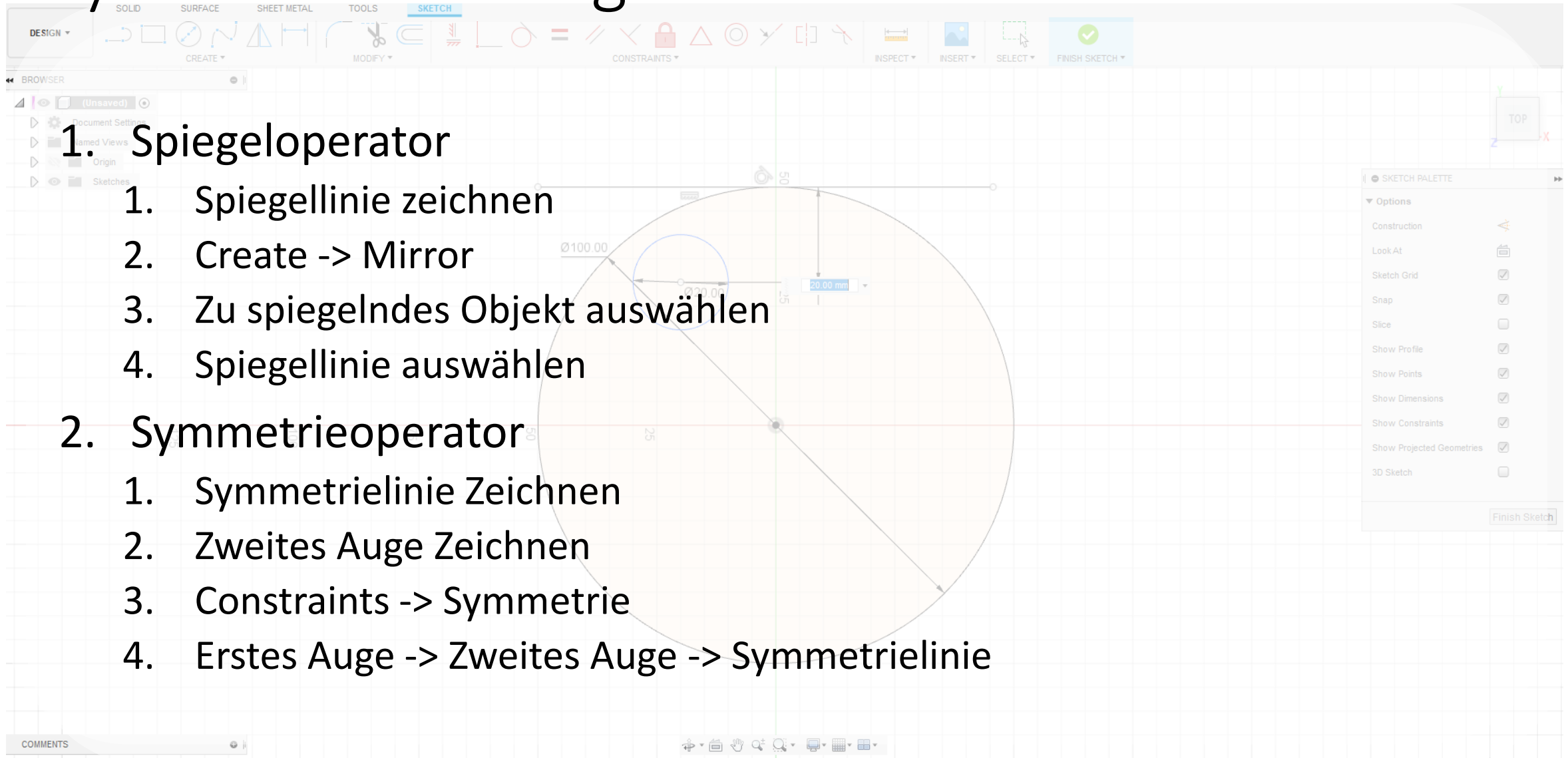


The image shows a CAD software interface with a sketch of an eye on a grid. The sketch consists of a large circle with a diameter of $\varnothing 100.00$ and a smaller circle with a diameter of $\varnothing 20.00$ inside it. The center of the large circle is at the origin (0,0). The center of the small circle is at (25, 20). A dimension line indicates a distance of 20.00 mm from the horizontal centerline to the center of the small circle. The sketch is highlighted in orange. The interface includes a top toolbar with various tools, a left sidebar with a browser, and a right sidebar with a sketch palette. The sketch palette has the following options:

- Options
- Construction
- Look At
- Sketch Grid
- Snap
- Slice
- Show Profile
- Show Points
- Show Dimensions
- Show Constraints
- Show Projected Geometries
- 3D Sketch

At the bottom right of the sketch palette is a "Finish Sketch" button. The bottom of the interface shows a comments bar and a navigation toolbar.

Symmetrische Augen zeichnen



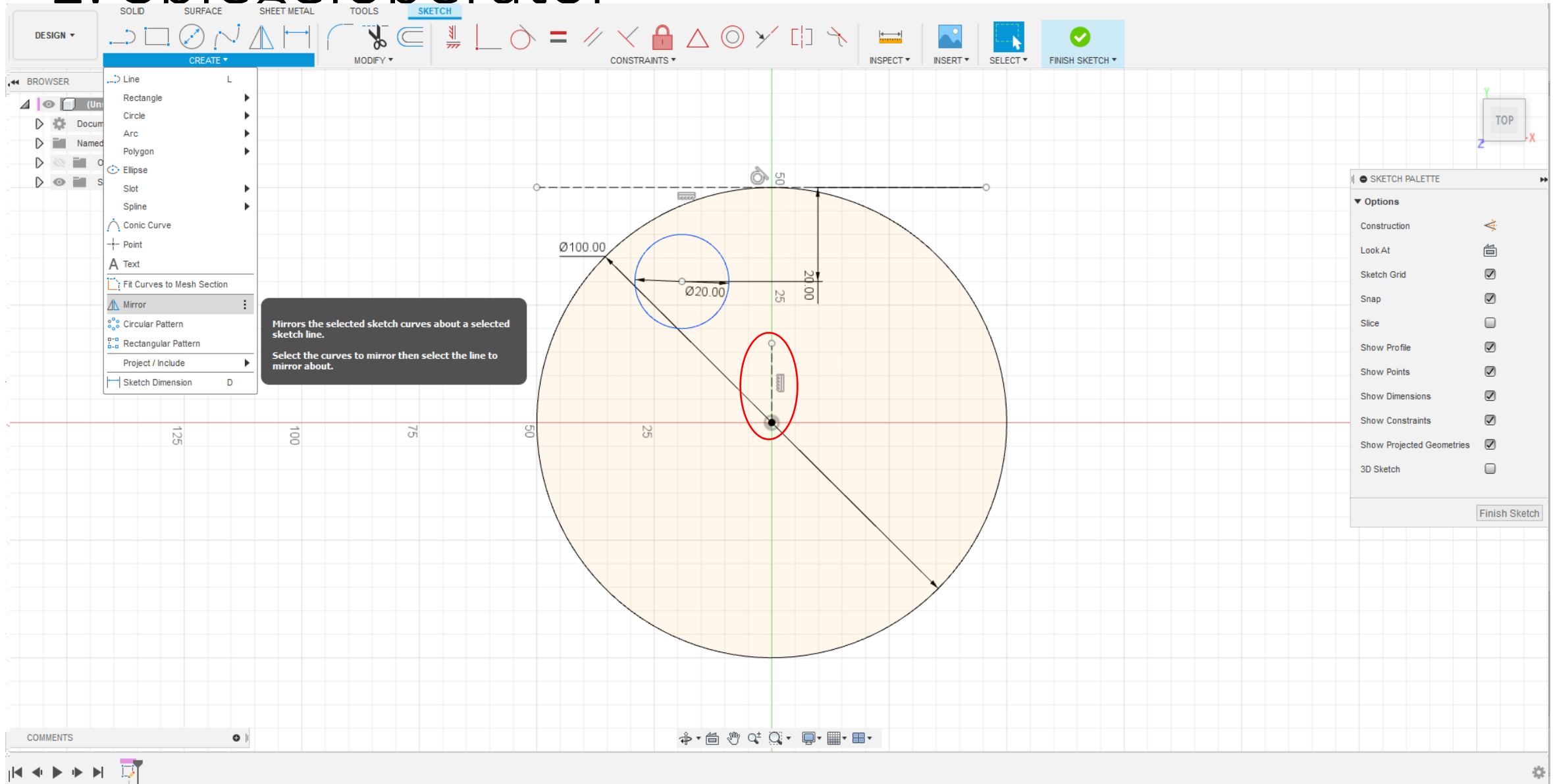
1. Spiegeloperator

1. Spiegellinie zeichnen
2. Create -> Mirror
3. Zu spiegelndes Objekt auswählen
4. Spiegellinie auswählen

2. Symmetrieoperator

1. Symmetrielinie Zeichnen
2. Zweites Auge Zeichnen
3. Constraints -> Symmetrie
4. Erstes Auge -> Zweites Auge -> Symmetrielinie

1. Spiegeloperator

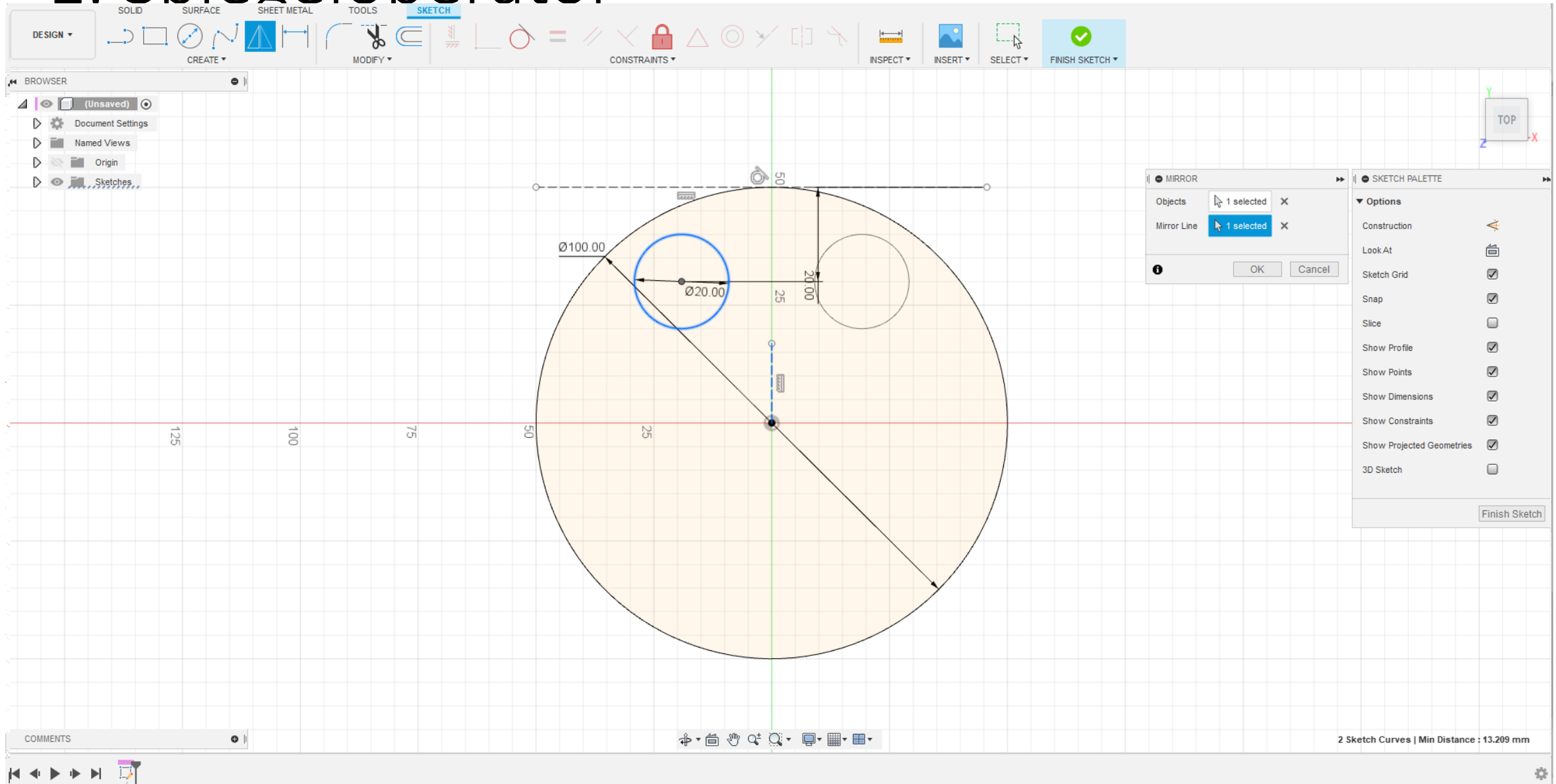


The image shows a CAD software interface with a sketch of a circle on a grid. A vertical green line serves as the mirror axis, and a horizontal red line represents the diameter. A blue circle of diameter $\varnothing 20.00$ is positioned in the upper-left quadrant, and its mirror image is shown in the lower-right quadrant. A red oval highlights the vertical mirror line. A tooltip box contains the following text:

Mirrors the selected sketch curves about a selected sketch line.
Select the curves to mirror then select the line to mirror about.

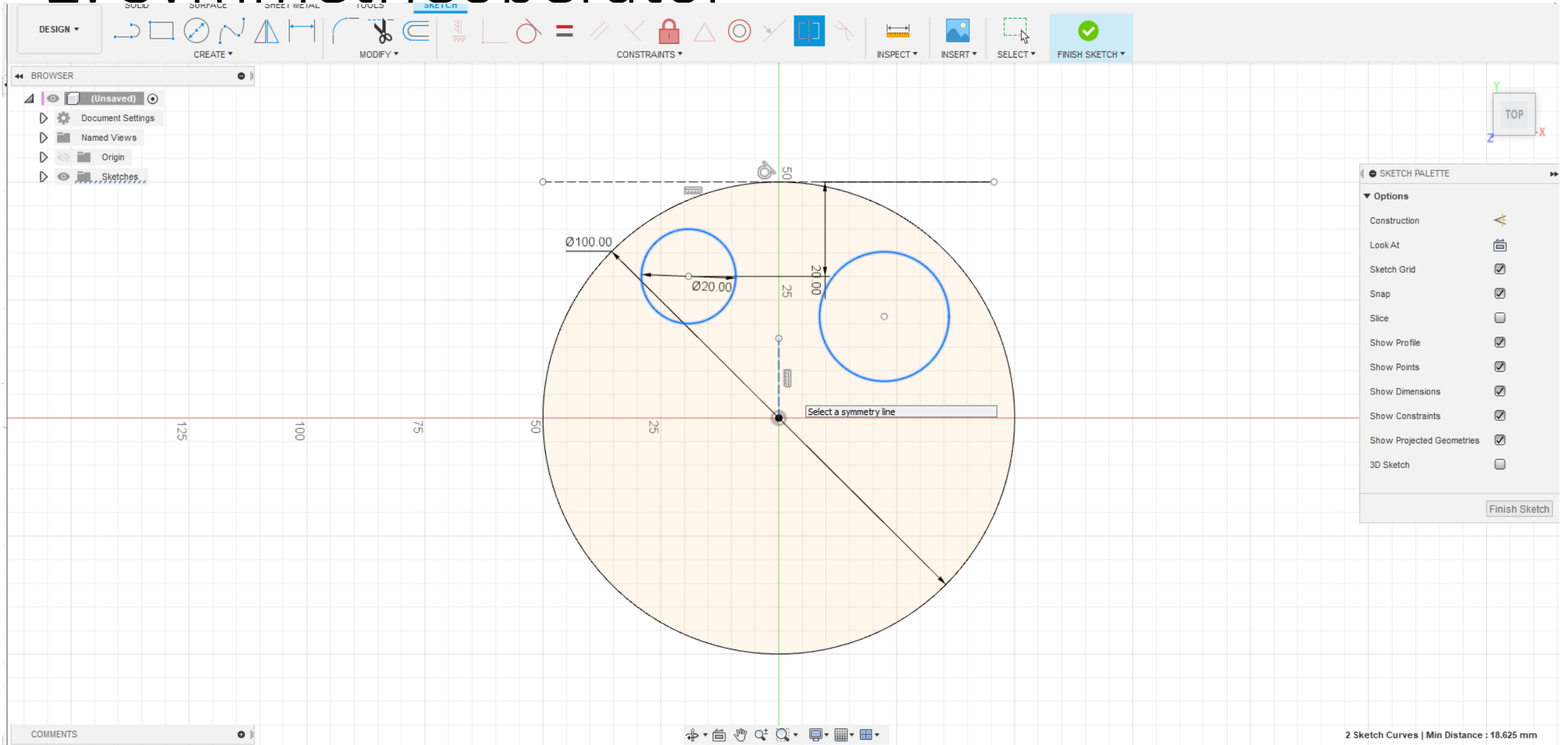
The interface includes a top toolbar with tabs for SOLID, SURFACE, SHEET METAL, TOOLS, and SKETCH. The SKETCH tab is active, showing options for CREATE, MODIFY, CONSTRAINTS, INSPECT, INSERT, SELECT, and FINISH SKETCH. On the left, a BROWSER panel lists sketching tools like Line, Rectangle, Circle, Arc, Polygon, Ellipse, Slot, Spline, Conic Curve, Point, Text, Fit Curves to Mesh Section, Mirror, Circular Pattern, Rectangular Pattern, Project / Include, and Sketch Dimension. On the right, a SKETCH PALETTE panel shows various options such as Construction, Look At, Sketch Grid, Snap, Slice, Show Profile, Show Points, Show Dimensions, Show Constraints, Show Projected Geometries, and 3D Sketch. A 'Finish Sketch' button is located at the bottom right of the palette.

1. Spiegeloperator



The screenshot displays a CAD software interface with a sketch of a circle on a grid. The main circle has a diameter of $\varnothing 100.00$. A smaller circle with a diameter of $\varnothing 20.00$ is positioned inside the larger circle. A vertical green line serves as the mirror axis, located 25 units from the center of the large circle. A horizontal dimension of 20.00 units is shown from the center of the large circle to the center of the small circle. The sketch is highlighted in orange. The software interface includes a top toolbar with tabs for SOLID, SURFACE, SHEET METAL, TOOLS, and SKETCH. A left sidebar shows a BROWSER with folders for Document Settings, Named Views, Origin, and Sketches. A right sidebar shows a MIRROR dialog box with 'Objects' and 'Mirror Line' both set to '1 selected', and a SKETCH PALETTE with various options like Construction, Look At, Sketch Grid, Snap, Slice, Show Profile, Show Points, Show Dimensions, Show Constraints, Show Projected Geometries, and 3D Sketch. A status bar at the bottom right indicates '2 Sketch Curves | Min Distance : 13.209 mm'.

2. Symmetrieoperator



The image shows a CAD software interface with a sketch of a circular part. The part is shaded orange and has two holes. The left hole has a diameter of $\varnothing 20.00$ and is positioned 25 units from the center. The right hole is larger and is also positioned 25 units from the center. The total diameter of the part is $\varnothing 100.00$. A vertical green line represents the axis of symmetry. A tooltip "Select a symmetry line" is visible over the vertical axis. The interface includes a top toolbar with "FINISH SKETCH", a left browser, and a right sketch palette.

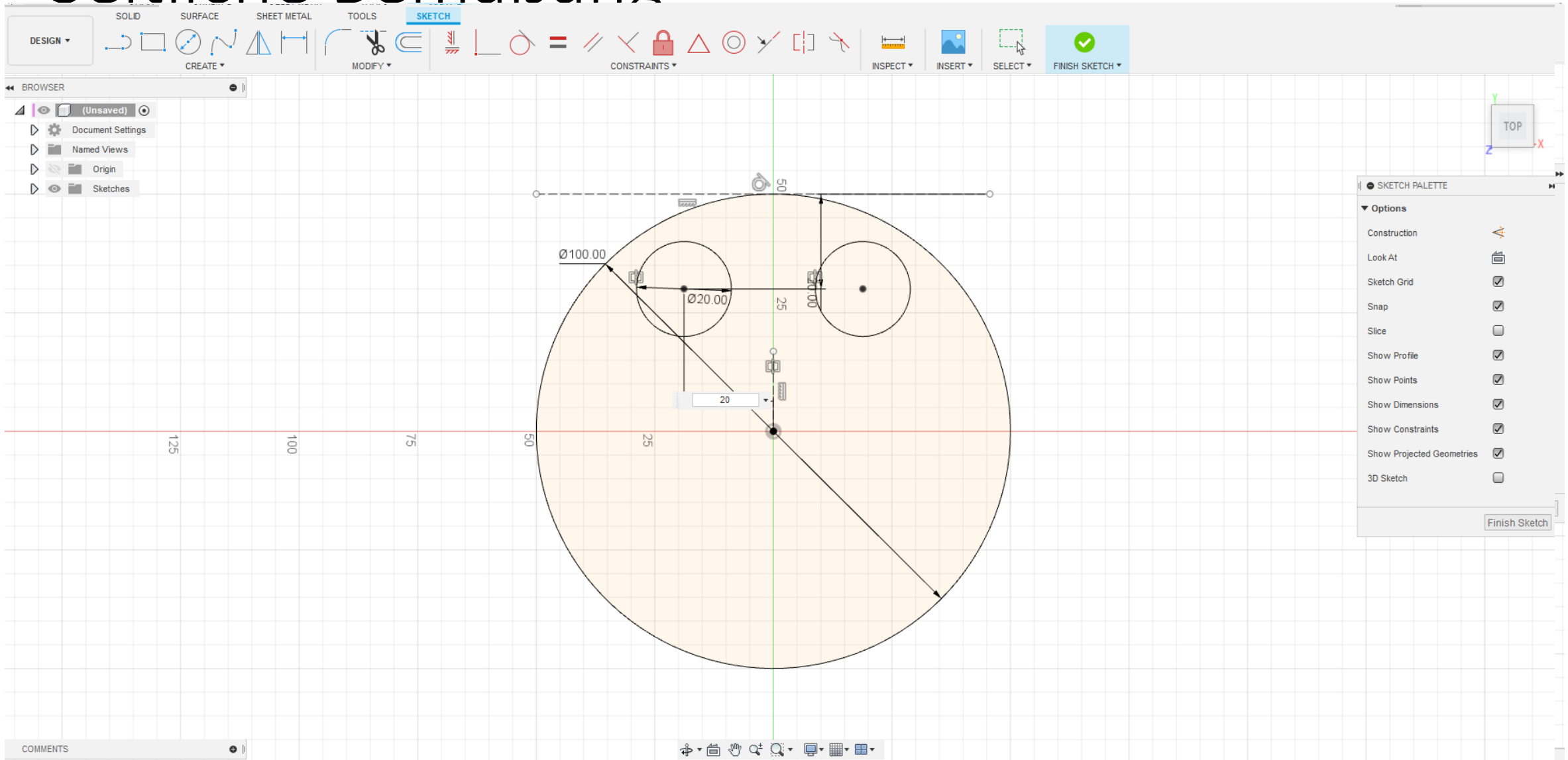
SKETCH PALETTE

- Options
- Construction
- Look At
- Sketch Grid
- Snap
- Slice
- Show Profile
- Show Points
- Show Dimensions
- Show Constraints
- Show Projected Geometries
- 3D Sketch

Finish Sketch

2 Sketch Curves | Min Distance : 18.625 mm

Seitliche Bemaßung

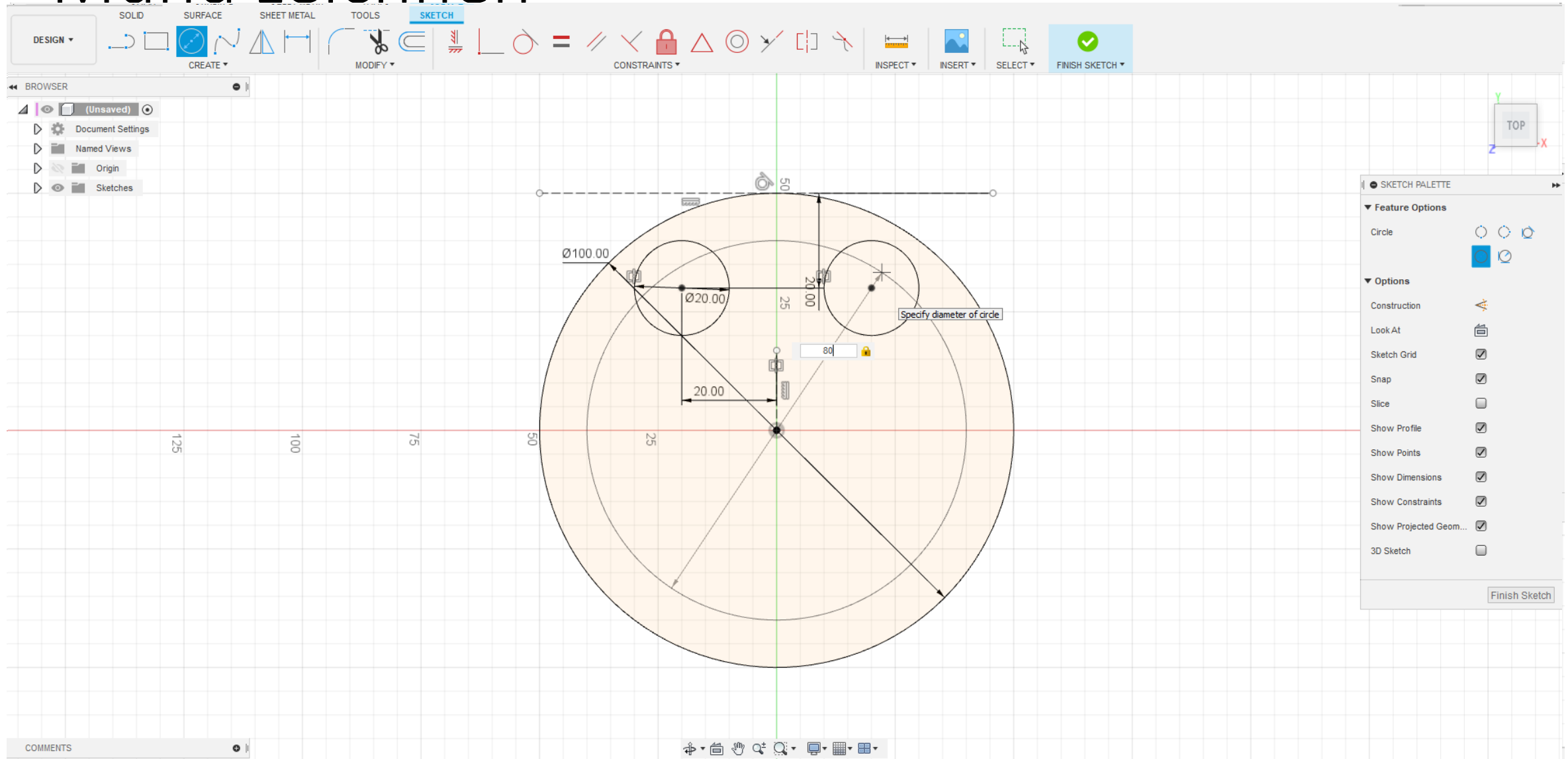


The image shows a CAD software interface with a sketch of a circular part. The sketch is a light orange circle with a diameter of $\varnothing 100.00$. It features two smaller circles, each with a diameter of $\varnothing 20.00$. The sketch is positioned on a grid with a vertical green axis and a horizontal red axis. Dimensions are shown as follows:

- The distance from the left edge of the circle to the center of the left hole is 75.
- The distance from the center of the left hole to the center of the right hole is 25.
- The distance from the center of the right hole to the right edge of the circle is 25.
- The distance from the top edge of the circle to the center of the right hole is 50.
- The distance from the center of the right hole to the top edge of the circle is 25.
- The distance from the center of the right hole to the center of the left hole is 25.
- The distance from the center of the right hole to the center of the left hole is 20.

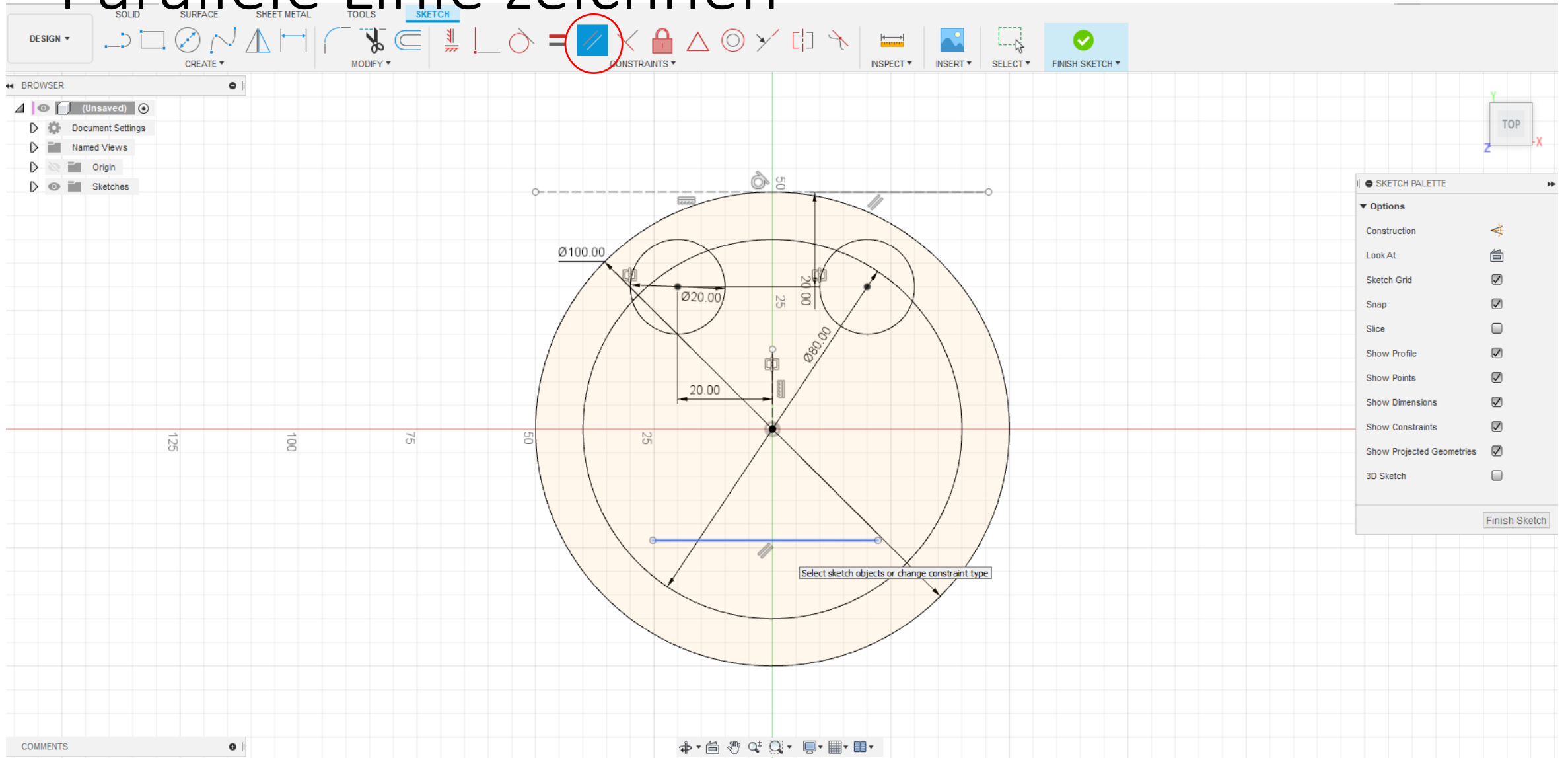
The software interface includes a top toolbar with tabs for SOLID, SURFACE, SHEET METAL, TOOLS, and SKETCH. The SKETCH tab is active, showing various sketching tools like lines, circles, and constraints. A right-hand panel titled "SKETCH PALETTE" contains options for Construction, Look At, Sketch Grid, Snap, Slice, Show Profile, Show Points, Show Dimensions, Show Constraints, Show Projected Geometries, and 3D Sketch. A "Finish Sketch" button is located at the bottom of this panel. The bottom of the interface has a "COMMENTS" section and a navigation bar.

Mund zeichnen



The image shows a CAD software interface with a sketch of a circular object. The sketch is centered on a grid with a vertical green axis and a horizontal red axis. The main circle has a diameter of $\varnothing 100.00$. Two smaller circles are positioned inside the main circle, each with a diameter of $\varnothing 20.00$. Dimensions are provided for the positions of these circles: the left one is 20.00 units from the center, and the right one is 20.00 units from the center and 25.00 units from the top edge. A dimension of 80.00 is also shown, likely representing the distance between the centers of the two small circles. The interface includes a top toolbar with tabs for SOLID, SURFACE, SHEET METAL, TOOLS, and SKETCH. The SKETCH tab is active, showing various drawing tools like lines, circles, and constraints. A right-hand panel titled 'SKETCH PALETTE' lists options for Feature Options, Options, and various display settings. A 'Finish Sketch' button is visible at the bottom right of the sketch area.

Parallele Linie zeichnen



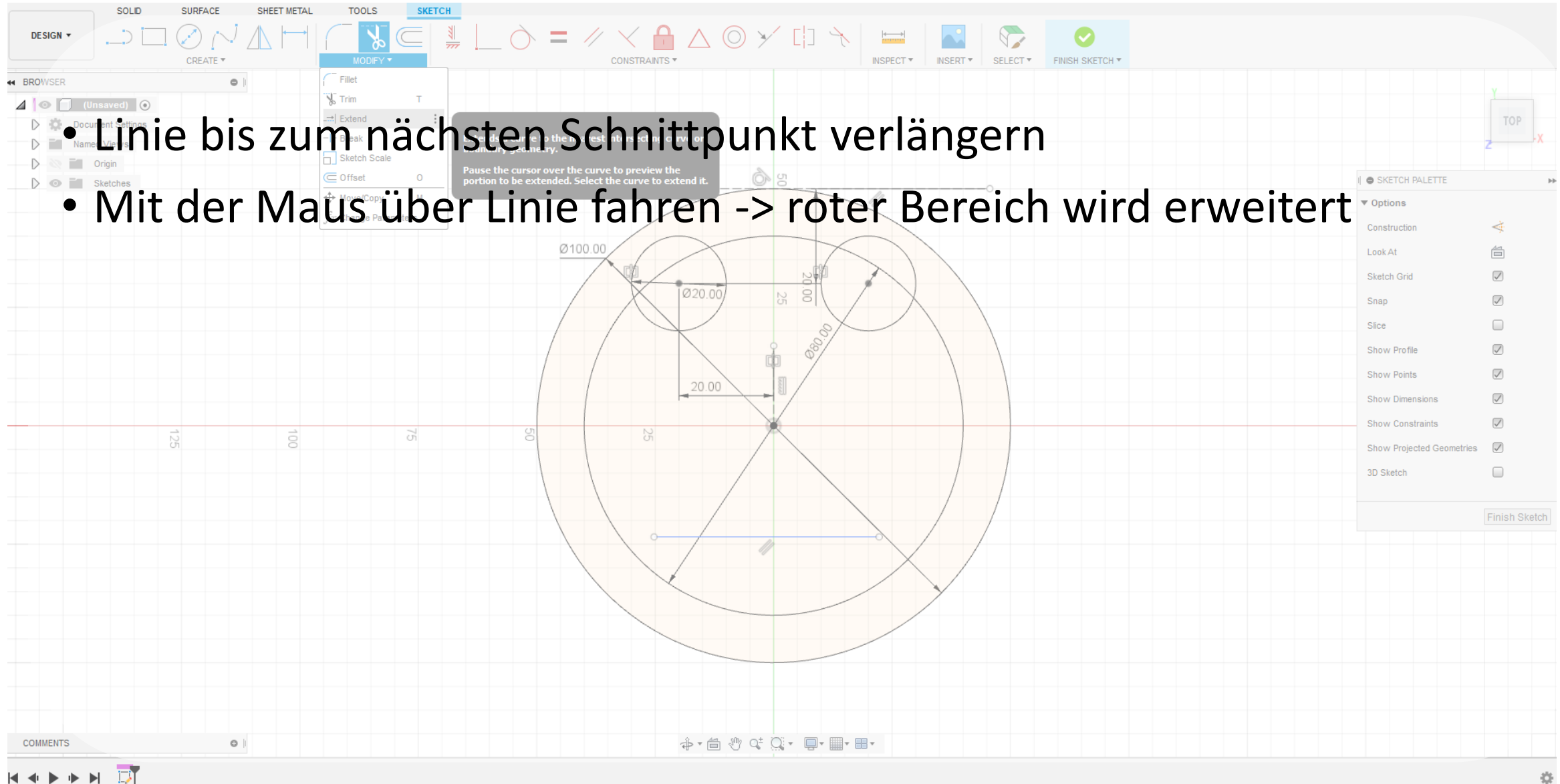
The image shows a CAD software interface with a sketch of a circular part. The top toolbar is visible, with the 'Parallel' constraint icon (two parallel lines) highlighted in a red circle. The sketch includes a large circle with a diameter of $\varnothing 100.00$ and a smaller circle with a diameter of $\varnothing 20.00$. A horizontal line is drawn across the bottom of the sketch, and a blue line is being drawn parallel to it. The 'Parallel' constraint icon is highlighted in the top toolbar. The 'Sketch' tab is active, and the 'Parallel' constraint icon is highlighted in a red circle. The 'Sketch Palette' on the right shows various options, including 'Parallel'.

SKETCH PALETTE

- Options
- Construction
- Look At
- Sketch Grid
- Snap
- Slice
- Show Profile
- Show Points
- Show Dimensions
- Show Constraints
- Show Projected Geometries
- 3D Sketch

Finish Sketch

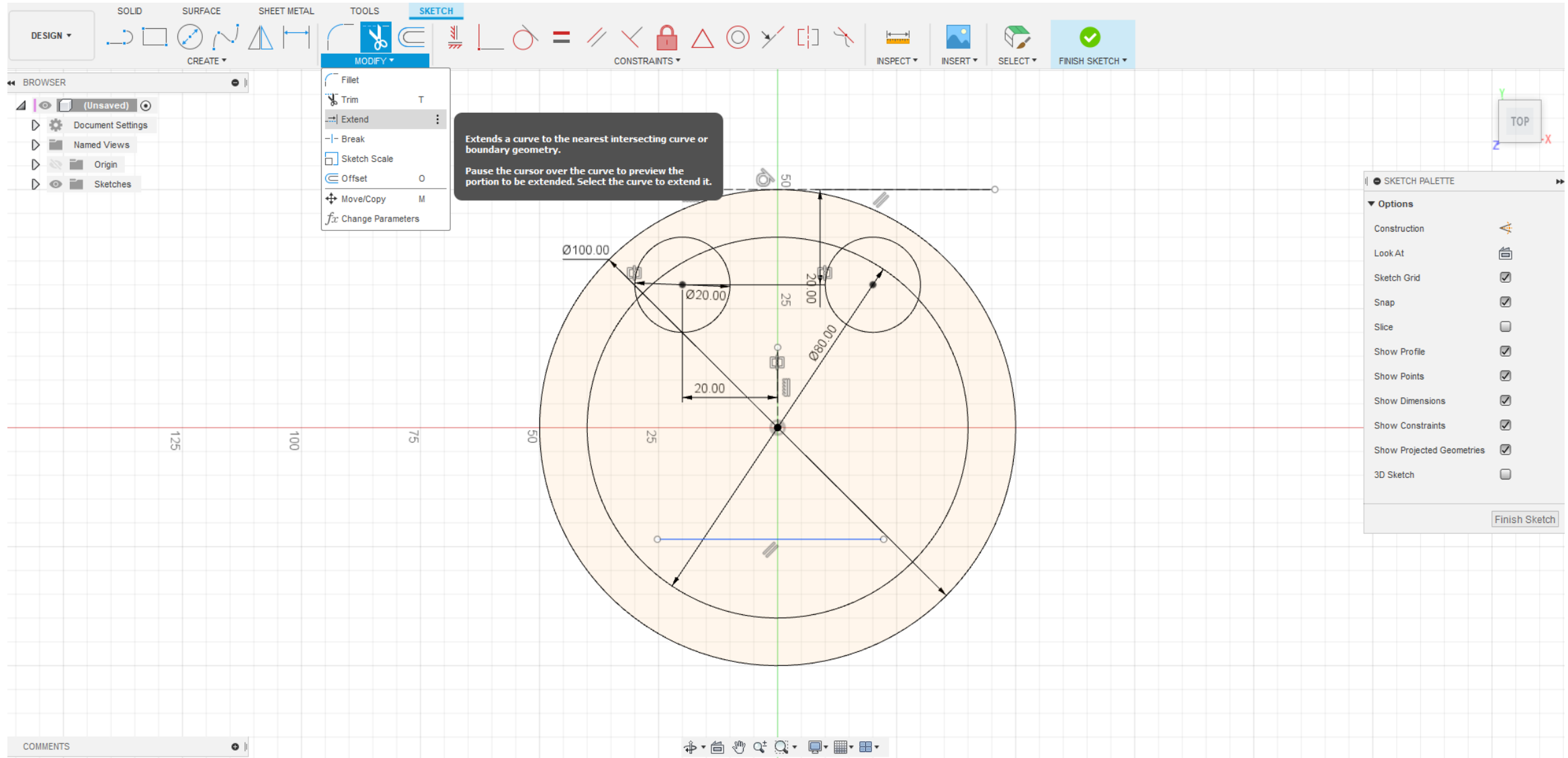
Linien erweitern



The screenshot shows a CAD software interface with a sketch of a circular part. The sketch includes a large outer circle with a diameter of $\varnothing 100.00$ and a smaller inner circle with a diameter of $\varnothing 80.00$. A horizontal line segment is drawn across the bottom of the circles, with a red extension area highlighted. The 'Extend' tool is active, and a tooltip indicates: "Pause the cursor over the curve to preview the portion to be extended. Select the curve to extend it." The interface includes a top toolbar with 'DESIGN', 'SOLID', 'SURFACE', 'SHEET METAL', 'TOOLS', and 'SKETCH' tabs. A 'BROWSER' panel on the left shows the document structure, and a 'SKETCH PALETTE' on the right lists various options like 'Construction', 'Look At', 'Sketch Grid', 'Snap', 'Slice', 'Show Profile', 'Show Points', 'Show Dimensions', 'Show Constraints', 'Show Projected Geometries', and '3D Sketch'. A 'Finish Sketch' button is located at the bottom right of the sketch palette.

- Linie bis zum nächsten Schnittpunkt verlängern
- Mit der Maus über Linie fahren -> roter Bereich wird erweitert

Linien erweitern



DESIGN | SOLID | SURFACE | SHEET METAL | TOOLS | **SKETCH**

CREATE | MODIFY | CONSTRAINTS | INSPECT | INSERT | SELECT | FINISH SKETCH

BROWSER

- (Unsaved)
- Document Settings
- Named Views
- Origin
- Sketches

MODIFY

- Fillet
- Trim T
- Extend**
- Break
- Sketch Scale
- Offset O
- Move/Copy M
- Change Parameters

SKETCH PALETTE

Options

- Construction
- Look At
- Sketch Grid
- Snap
- Slice
- Show Profile
- Show Points
- Show Dimensions
- Show Constraints
- Show Projected Geometries
- 3D Sketch

Finish Sketch

SKETCH

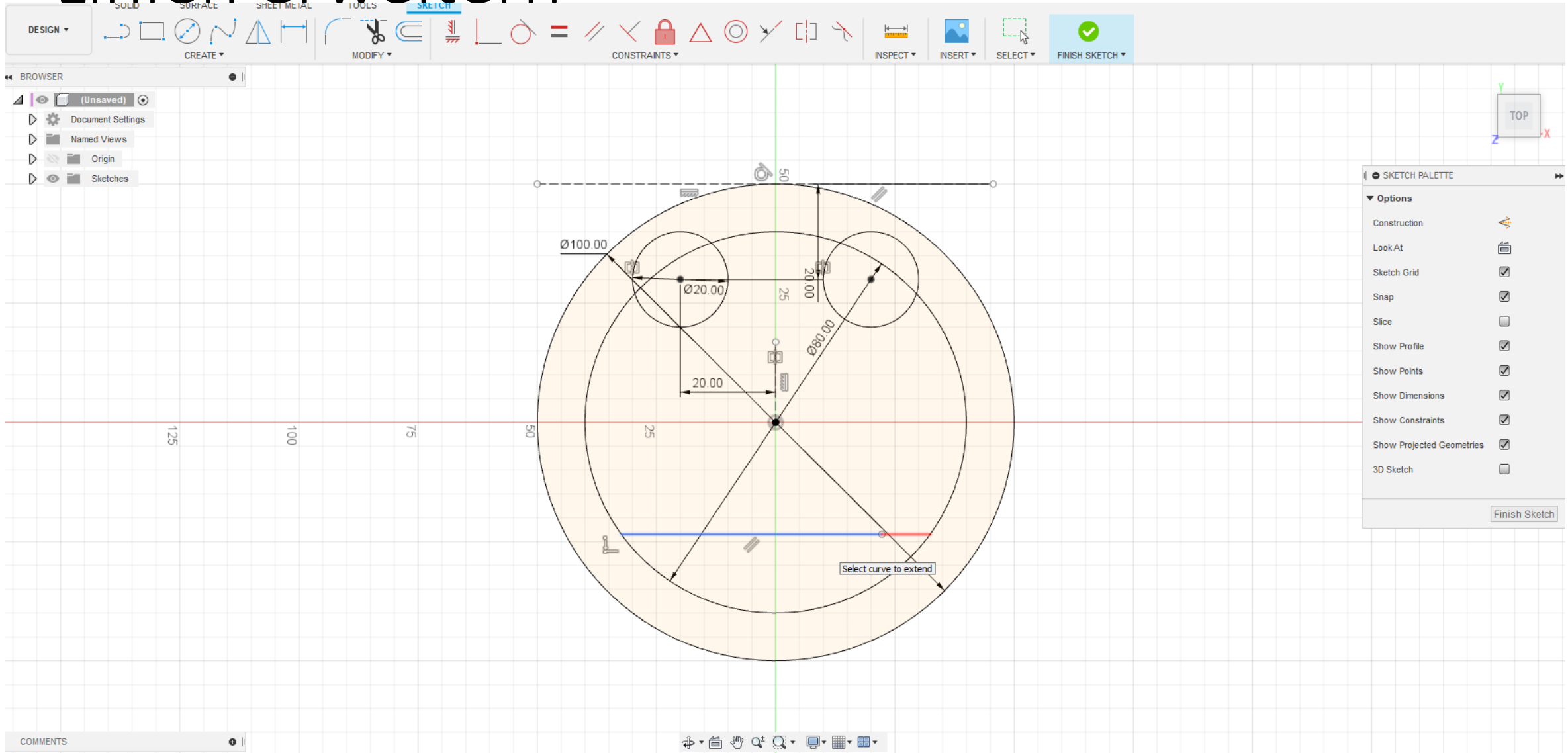
Extends a curve to the nearest intersecting curve or boundary geometry.

Pause the cursor over the curve to preview the portion to be extended. Select the curve to extend it.

Dimensions: $\varnothing 100.00$, $\varnothing 20.00$, $\varnothing 80.00$, 25, 20.00, 20.00, 50, 125, 100, 75, 50

COMMENTS

Linien erweitern



The screenshot shows a CAD software interface with a sketch of a circular part. The sketch is centered on a grid with a red horizontal axis and a green vertical axis. The part has a large outer circle with a diameter of $\varnothing 100.00$ and a smaller inner circle with a diameter of $\varnothing 20.00$. A horizontal line segment is drawn across the bottom of the part, with a tooltip that says "Select curve to extend". The sketch is highlighted in orange. The interface includes a top toolbar with various tools, a left sidebar with a browser, and a right sidebar with a sketch palette.

Top Toolbar: DESIGN, SOLID, SURFACE, SHEET METAL, TOOLS, SKETCH, CREATE, MODIFY, CONSTRAINTS, INSPECT, INSERT, SELECT, FINISH SKETCH.

Left Sidebar (BROWSER): (Unsaved), Document Settings, Named Views, Origin, Sketches.

Right Sidebar (SKETCH PALETTE): Options, Construction, Look At, Sketch Grid, Snap, Slice, Show Profile, Show Points, Show Dimensions, Show Constraints, Show Projected Geometries, 3D Sketch, Finish Sketch.

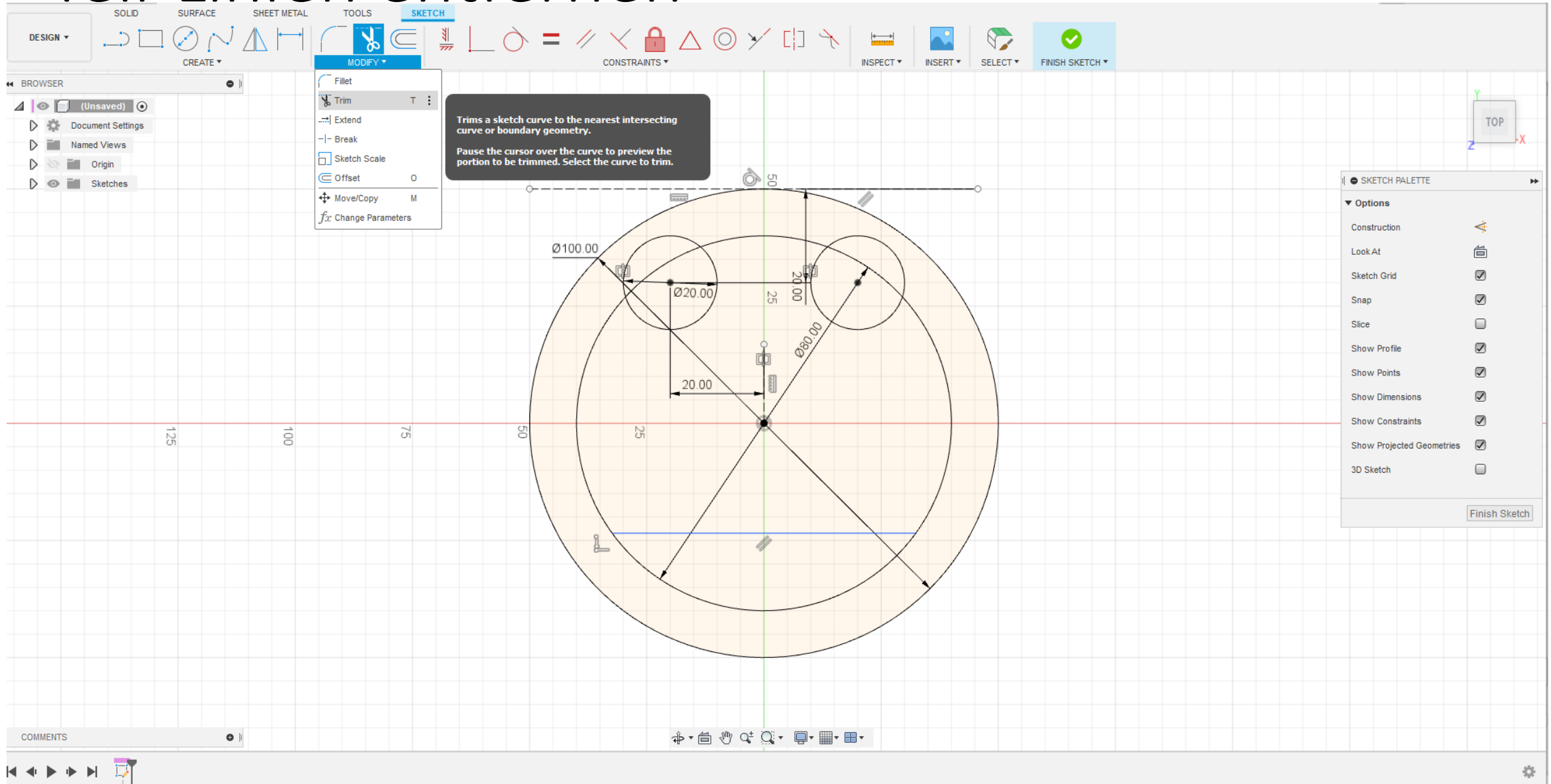
Dimensions: $\varnothing 100.00$, $\varnothing 20.00$, 20.00, 25, 50, 75, 100, 125.

Tooltip: Select curve to extend

COMMENTS



Teil-Linien entfernen



The image shows a CAD software interface with a sketch of a circular part. The 'Trim' tool is selected, and a tooltip explains its function: "Trims a sketch curve to the nearest intersecting curve or boundary geometry. Pause the cursor over the curve to preview the portion to be trimmed. Select the curve to trim." The sketch includes a large circle with a diameter of 100.00, a smaller circle with a diameter of 20.00, and a larger circle with a diameter of 80.00. Dimensions of 25, 20.00, and 50 are shown. The interface includes a top toolbar with 'DESIGN', 'SOLID', 'SURFACE', 'SHEET METAL', 'TOOLS', and 'SKETCH' tabs. A 'BROWSER' panel on the left shows '(Unsaved)', 'Document Settings', 'Named Views', 'Origin', and 'Sketches'. A 'SKETCH PALETTE' on the right lists options like 'Construction', 'Look At', 'Sketch Grid', 'Snap', 'Slice', 'Show Profile', 'Show Points', 'Show Dimensions', 'Show Constraints', 'Show Projected Geometries', and '3D Sketch'. A 'Finish Sketch' button is at the bottom right of the palette. The bottom of the screen shows a 'COMMENTS' panel and navigation icons.

Teil-Linien entfernen

DESIGN ▾ SOLID SURFACE SHEET METAL TOOLS SKETCH

CREATE ▾ MODIFY ▾ CONSTRAINTS ▾ INSPECT ▾ INSERT ▾ SELECT ▾ FINISH SKETCH ▾

← BROWSER (Unsaved) Document Settings Named Views Origin Sketches

TOP

SKETCH PALETTE

- Options
- Construction
- Look At
- Sketch Grid
- Snap
- Slice
- Show Profile
- Show Points
- Show Dimensions
- Show Constraints
- Show Projected Geometries
- 3D Sketch

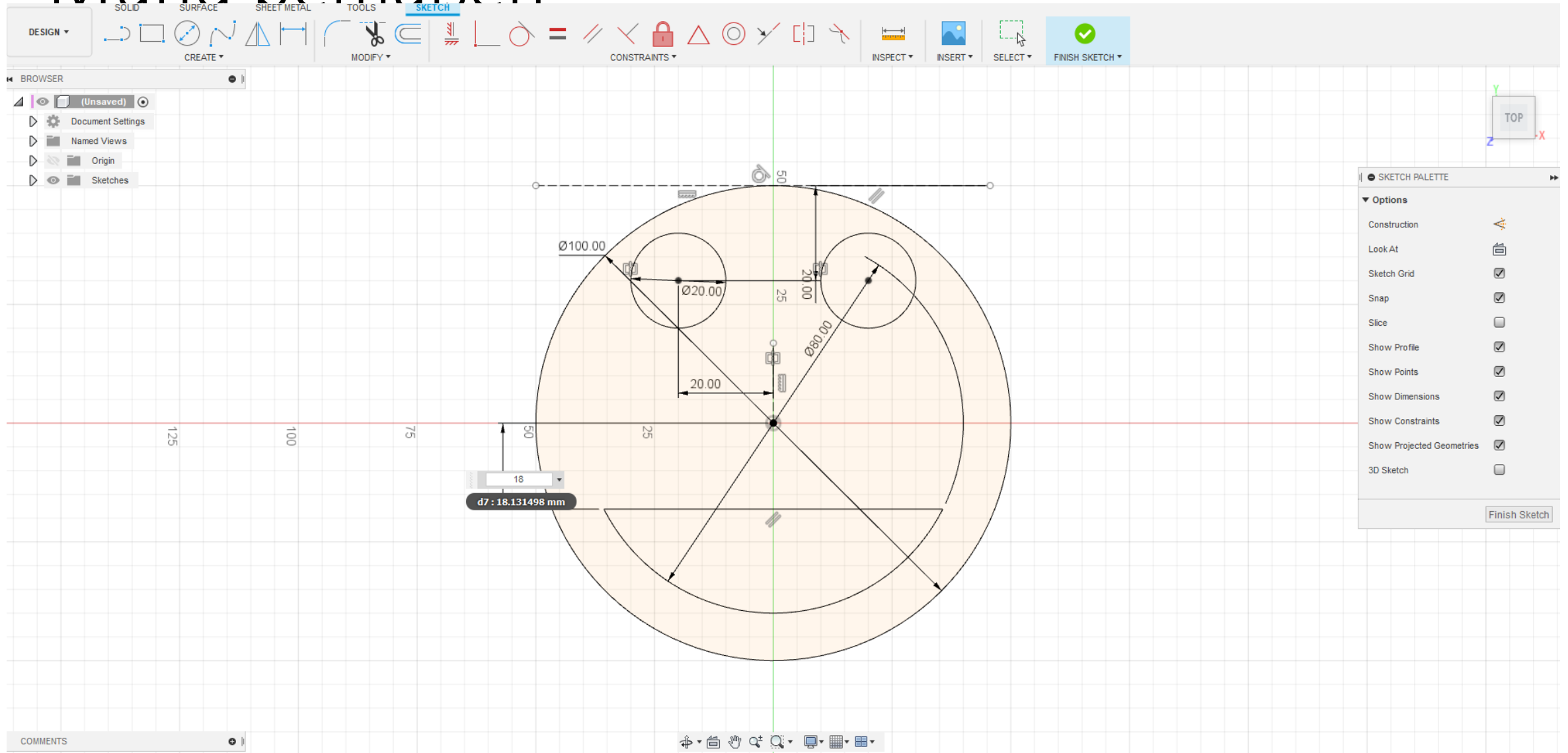
Finish Sketch

1 warning(s)
Constraints and/or dimensions were removed during operation.
[More Info](#)

COMMENTS

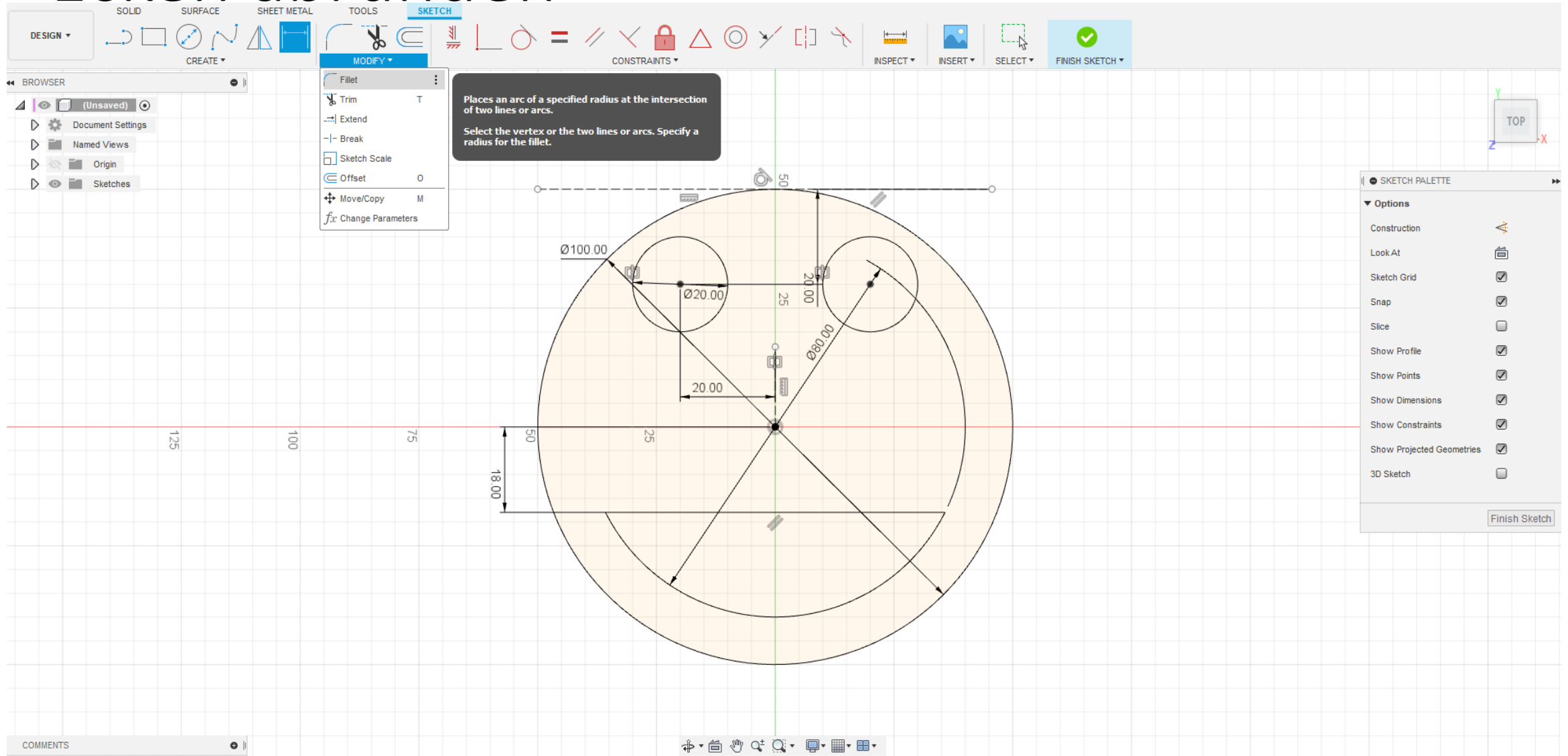
The image shows a CAD software interface with a circular sketch on a grid. The sketch features a large circle with a diameter of 100.00. Inside this circle, there are two smaller circles, each with a diameter of 20.00. A vertical dimension line indicates a distance of 25 from the center to the top of the inner circles. A horizontal dimension line indicates a distance of 20.00 from the center to the right edge of the inner circles. A red arc is highlighted on the right side of the large circle, with a tooltip that says "Select curve section to trim". The interface includes a top toolbar with various tools like SOLID, SURFACE, SHEET METAL, TOOLS, and SKETCH. A left sidebar shows a BROWSER with folders for Document Settings, Named Views, Origin, and Sketches. A right sidebar shows a SKETCH PALETTE with various options like Construction, Look At, Sketch Grid, Snap, Slice, Show Profile, Show Points, Show Dimensions, Show Constraints, Show Projected Geometries, and 3D Sketch. A warning message at the bottom right states "1 warning(s) Constraints and/or dimensions were removed during operation." and provides a "More Info" link. The bottom of the interface has a COMMENTS field and a navigation bar.

Mund bemaßen



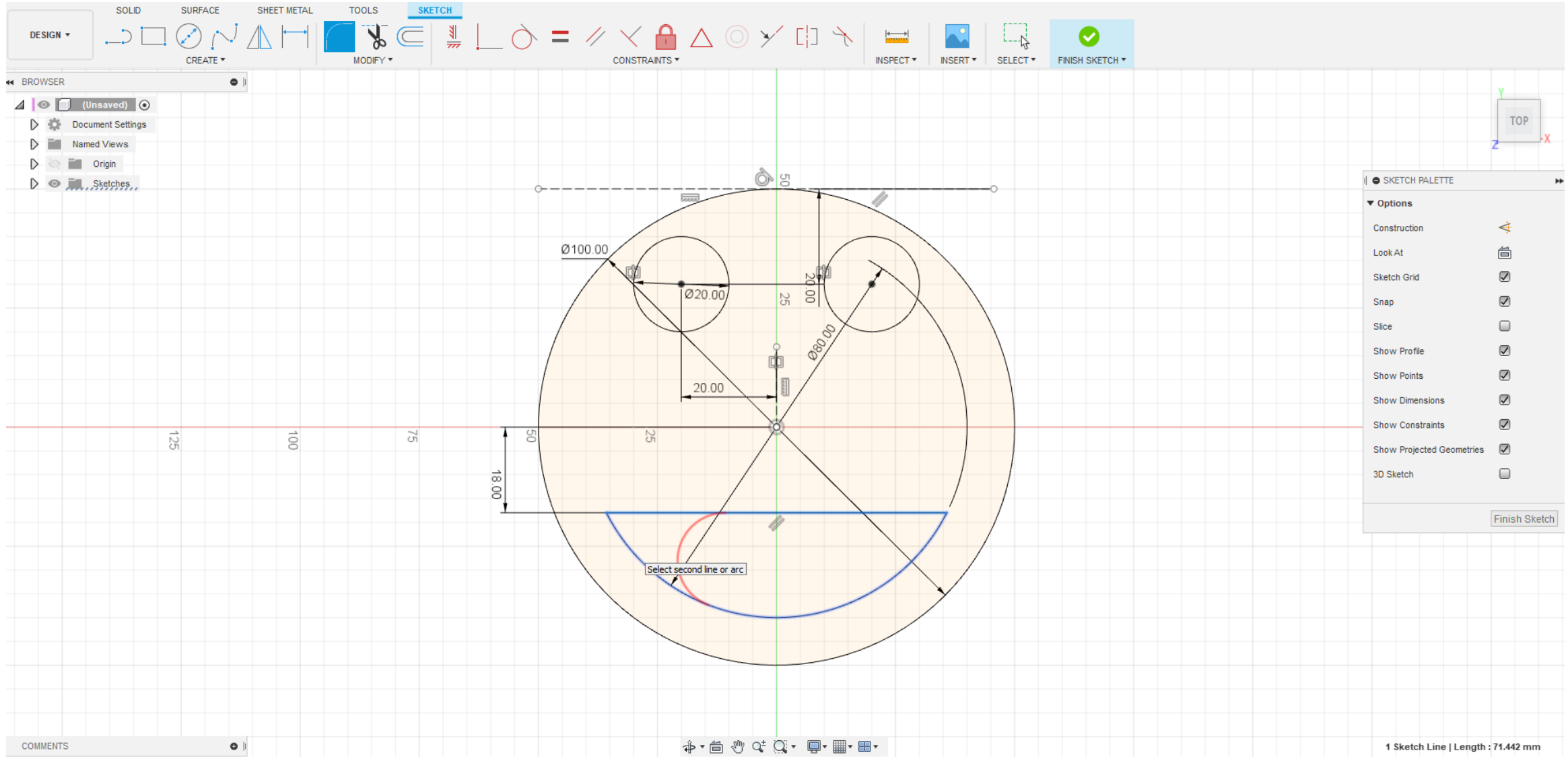
The image shows a CAD software interface with a sketch of a circular part. The sketch is centered on a grid with a red horizontal axis and a green vertical axis. The main circle has a diameter of $\text{Ø}100.00$. Two smaller circles are positioned horizontally from the center, each with a diameter of $\text{Ø}20.00$. A dimension of 20.00 is shown between the center of the main circle and the center of the left small circle. Another dimension of 20.00 is shown between the center of the main circle and the center of the right small circle. A vertical dimension of 25 is shown from the center of the main circle to the top edge of the right small circle. A dimension of 50 is shown from the top edge of the main circle to the top edge of the right small circle. A dimension of 125 is shown from the left edge of the main circle to the left edge of the right small circle. A dimension of 100 is shown from the left edge of the main circle to the center of the main circle. A dimension of 75 is shown from the left edge of the main circle to the center of the right small circle. A dimension of 50 is shown from the center of the main circle to the center of the right small circle. A dimension of 25 is shown from the center of the main circle to the center of the right small circle. A dimension of 18 is shown from the center of the main circle to the center of the right small circle. A dimension of $d7 : 18.131498 \text{ mm}$ is shown from the center of the main circle to the center of the right small circle. The interface includes a top toolbar with tabs for SOLID, SURFACE, SHEET METAL, TOOLS, and SKETCH. The SKETCH tab is active. The toolbar contains icons for CREATE, MODIFY, CONSTRAINTS, INSPECT, INSERT, SELECT, and FINISH SKETCH. A left sidebar shows a BROWSER with folders for Document Settings, Named Views, Origin, and Sketches. A right sidebar shows a SKETCH PALETTE with options for Construction, Look At, Sketch Grid, Snap, Slice, Show Profile, Show Points, Show Dimensions, Show Constraints, Show Projected Geometries, and 3D Sketch. A bottom toolbar contains icons for navigation and editing. A bottom status bar shows navigation arrows and a settings icon.

Ecken abrunden



The image shows a CAD software interface with a sketch of a circular part. The sketch is on a grid with dimensions: a large circle with diameter $\varnothing 100.00$, a smaller circle with diameter $\varnothing 20.00$, and a fillet with radius $R 20.00$. The sketch is centered on a vertical green axis and a horizontal red axis. Dimensions include 125, 100, 75, 50, 25, 18.00, 20.00, 21.00, and 50.00. The software interface includes a top toolbar with 'DESIGN', 'CREATE', 'MODIFY', 'CONSTRAINTS', 'INSPECT', 'INSERT', 'SELECT', and 'FINISH SKETCH' buttons. A 'BROWSER' panel on the left shows '(Unsaved)', 'Document Settings', 'Named Views', 'Origin', and 'Sketches'. A 'SKETCH PALETTE' on the right shows 'Options' with checkboxes for 'Sketch Grid', 'Snap', 'Show Profile', 'Show Points', 'Show Dimensions', 'Show Constraints', 'Show Projected Geometries', and '3D Sketch'. A 'Finish Sketch' button is at the bottom right of the palette. A tooltip for the 'Fillet' tool is visible, stating: 'Places an arc of a specified radius at the intersection of two lines or arcs. Select the vertex or the two lines or arcs. Specify a radius for the fillet.'

Ecken abrunden



DESIGN ▾ SOLID SURFACE SHEET METAL TOOLS SKETCH

CREATE ▾ MODIFY ▾ CONSTRAINTS ▾ INSPECT ▾ INSERT ▾ SELECT ▾ FINISH SKETCH ▾

BROWSER

- (Unsaved)
- Document Settings
- Named Views
- Origin
- Sketches

SKETCH PALETTE

Options

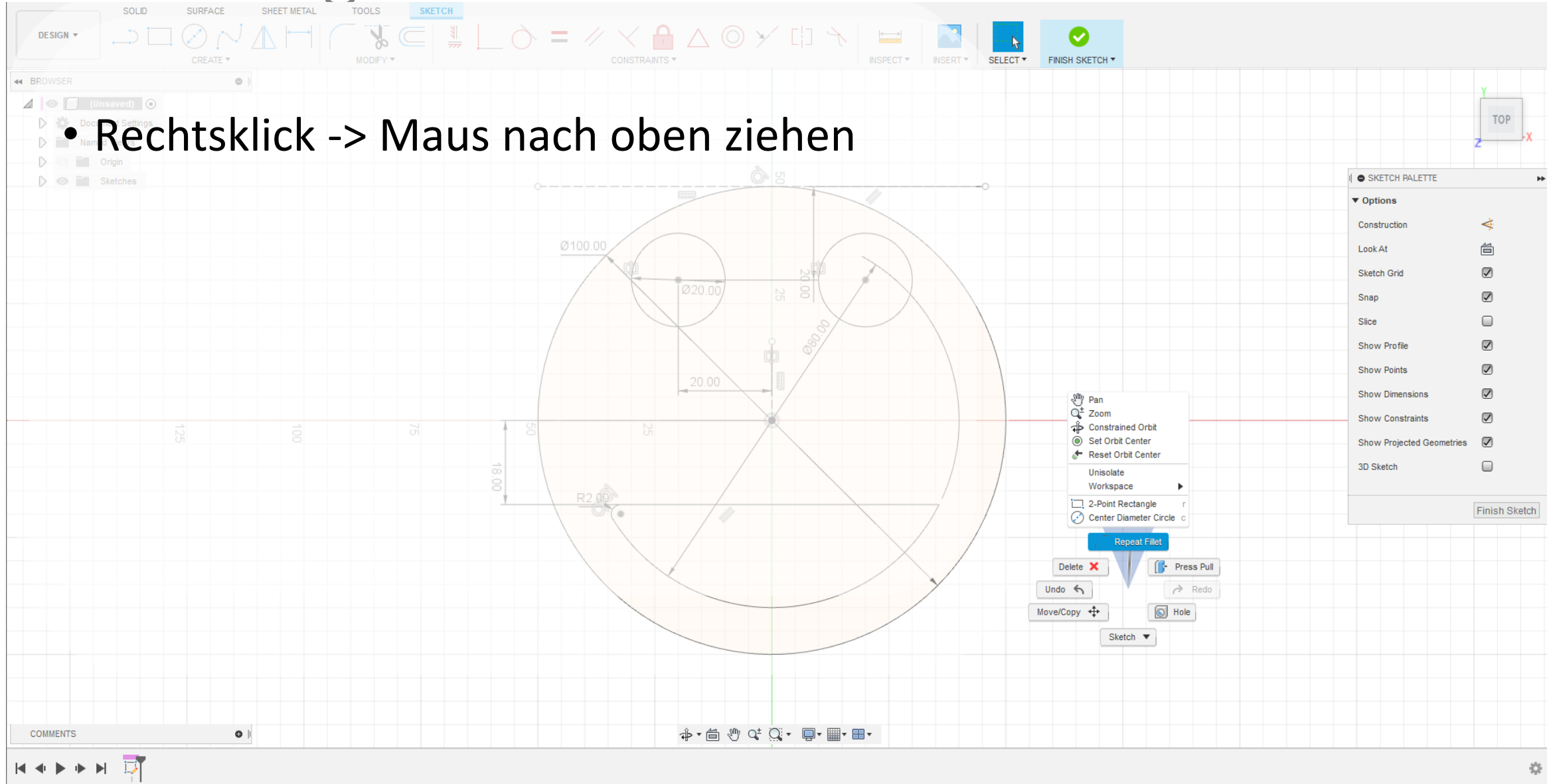
- Construction
- Look At
- Sketch Grid
- Snap
- Slice
- Show Profile
- Show Points
- Show Dimensions
- Show Constraints
- Show Projected Geometries
- 3D Sketch

Finish Sketch

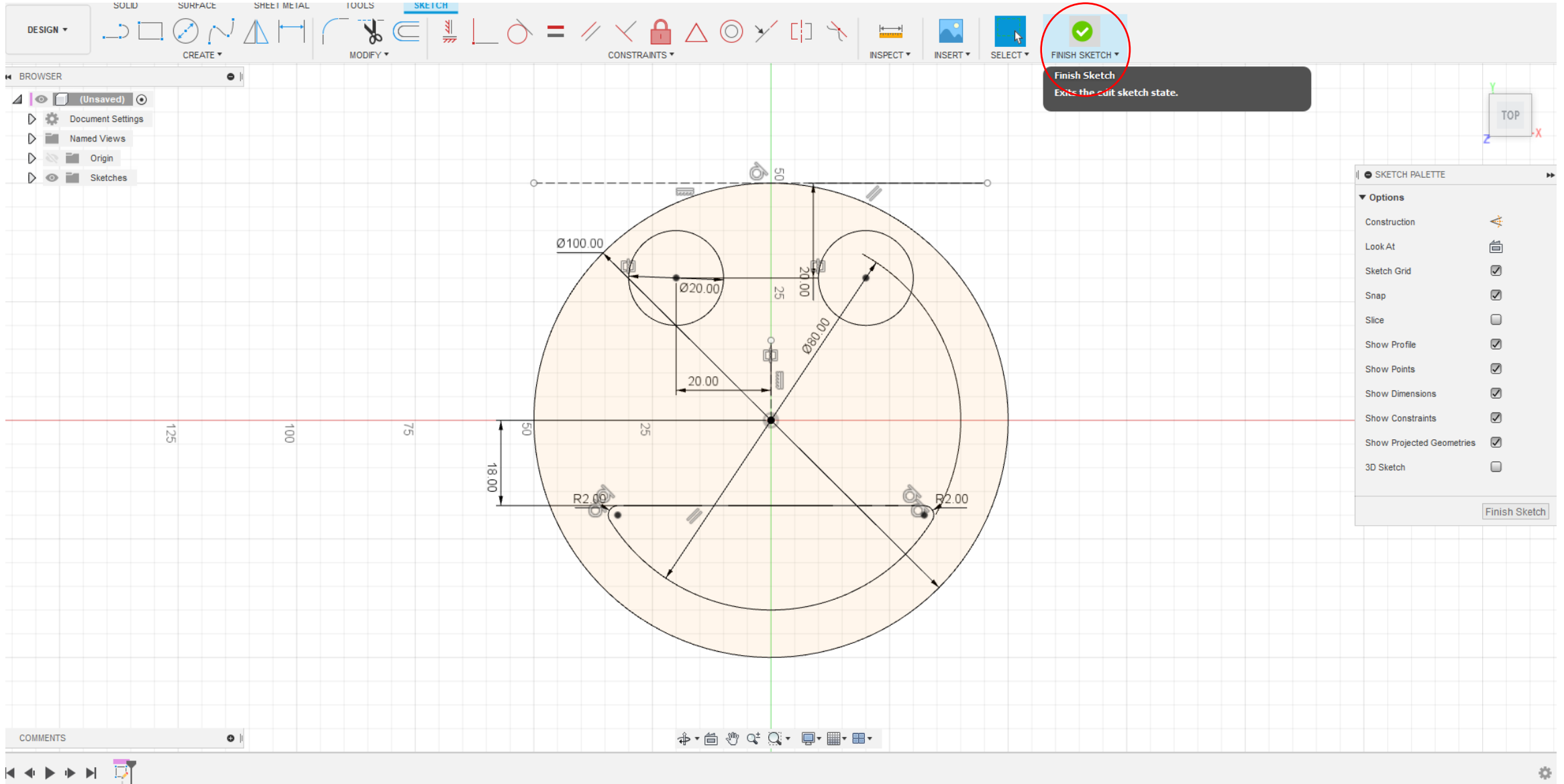
1 Sketch Line | Length : 71.442 mm

Werkzeug wiederholen

- Rechtsklick -> Maus nach oben ziehen

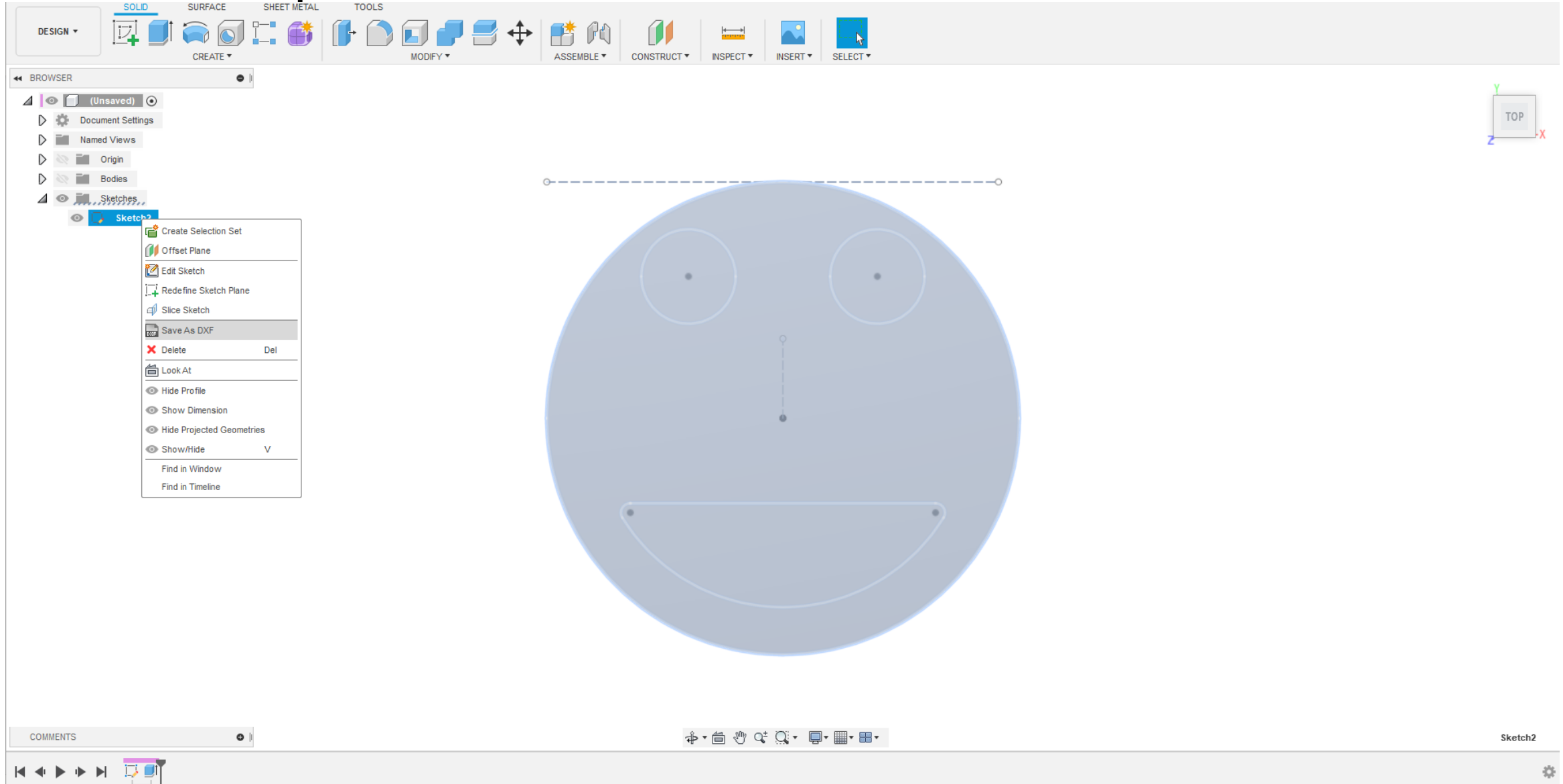


Skizze beenden



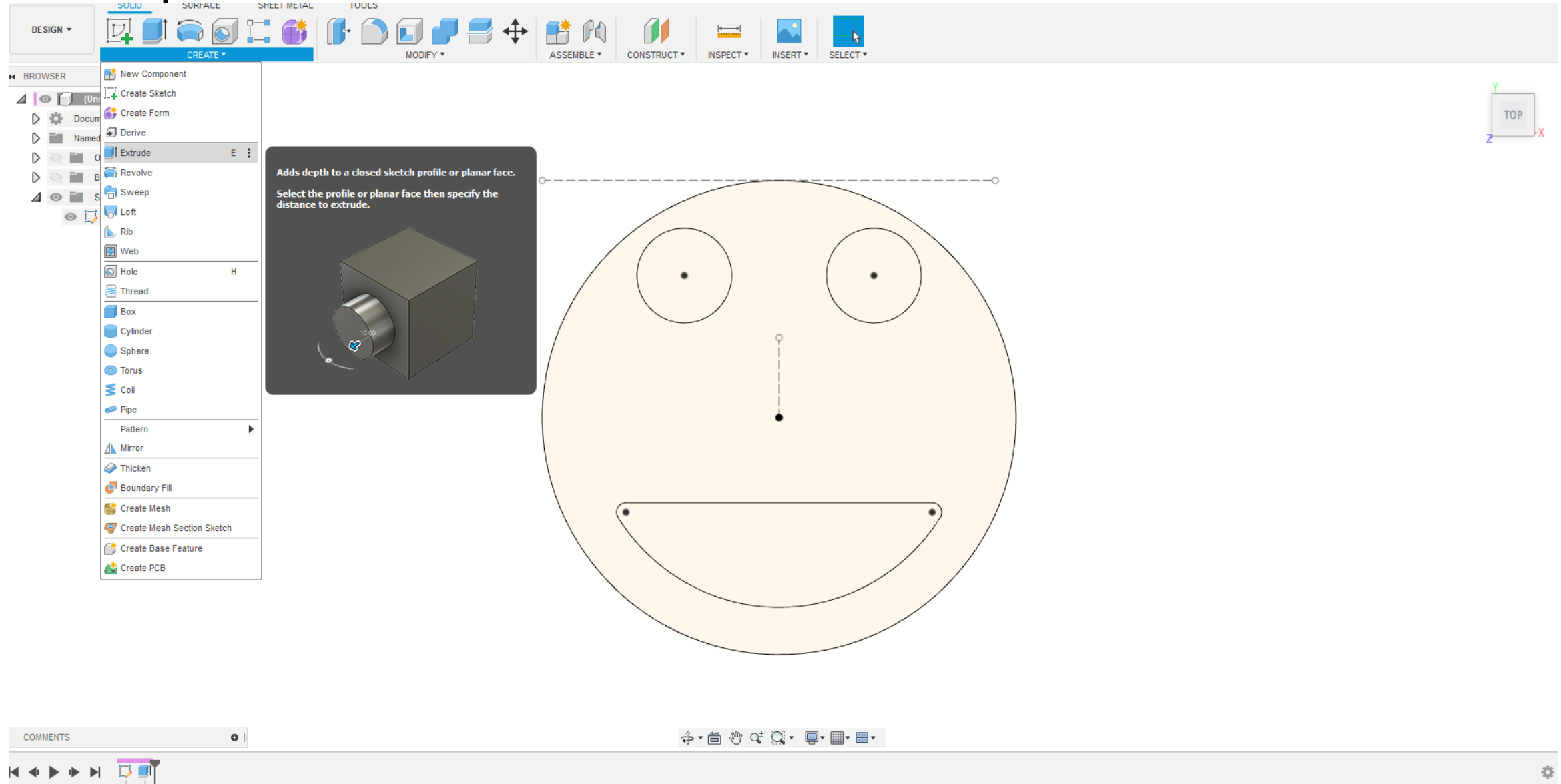
The image shows a CAD software interface with a technical sketch of a circular part. The sketch is centered on a grid with a red horizontal axis and a green vertical axis. The part has a diameter of $\varnothing 100.00$ and a central hole with a diameter of $\varnothing 20.00$. There are two smaller circles with a diameter of $\varnothing 80.00$ and two fillets with a radius of $R2.00$. Dimensions include a distance of 50 from the center to the top edge, 25 from the center to the hole edge, and 20 from the hole edge to the smaller circles. A distance of 18.00 is shown from the bottom edge to the fillets. The interface includes a top toolbar with a 'FINISH SKETCH' button circled in red, a 'SKETCH PALETTE' on the right, and a 'BROWSER' on the left. A tooltip for the 'FINISH SKETCH' button reads: 'Finish Sketch. Exits the edit sketch state.'

*.dxf exportieren



The screenshot displays a CAD software interface with a central workspace containing a blue smiley face sketch. The sketch consists of a large circle, two smaller circles for eyes, and a curved line for a mouth. A dashed horizontal line is positioned above the top of the face, and a vertical dashed line passes through the center of the face. A context menu is open over the sketch, listing various actions such as 'Create Selection Set', 'Offset Plane', 'Edit Sketch', 'Redefine Sketch Plane', 'Slice Sketch', 'Save As DXF' (highlighted), 'Delete', 'Look At', 'Hide Profile', 'Show Dimension', 'Hide Projected Geometries', and 'Show/Hide'. The interface includes a top toolbar with tabs for SOLID, SURFACE, SHEET METAL, and TOOLS, and sub-tabs for DESIGN, CREATE, MODIFY, ASSEMBLE, CONSTRUCT, INSPECT, INSERT, and SELECT. A left sidebar shows a BROWSER with folders for Document Settings, Named Views, Origin, Bodies, and Sketches. A right sidebar shows a 'TOP' view indicator. The bottom of the interface features a COMMENTS section, a navigation toolbar, and a status bar with the text 'Sketch2' and a gear icon.

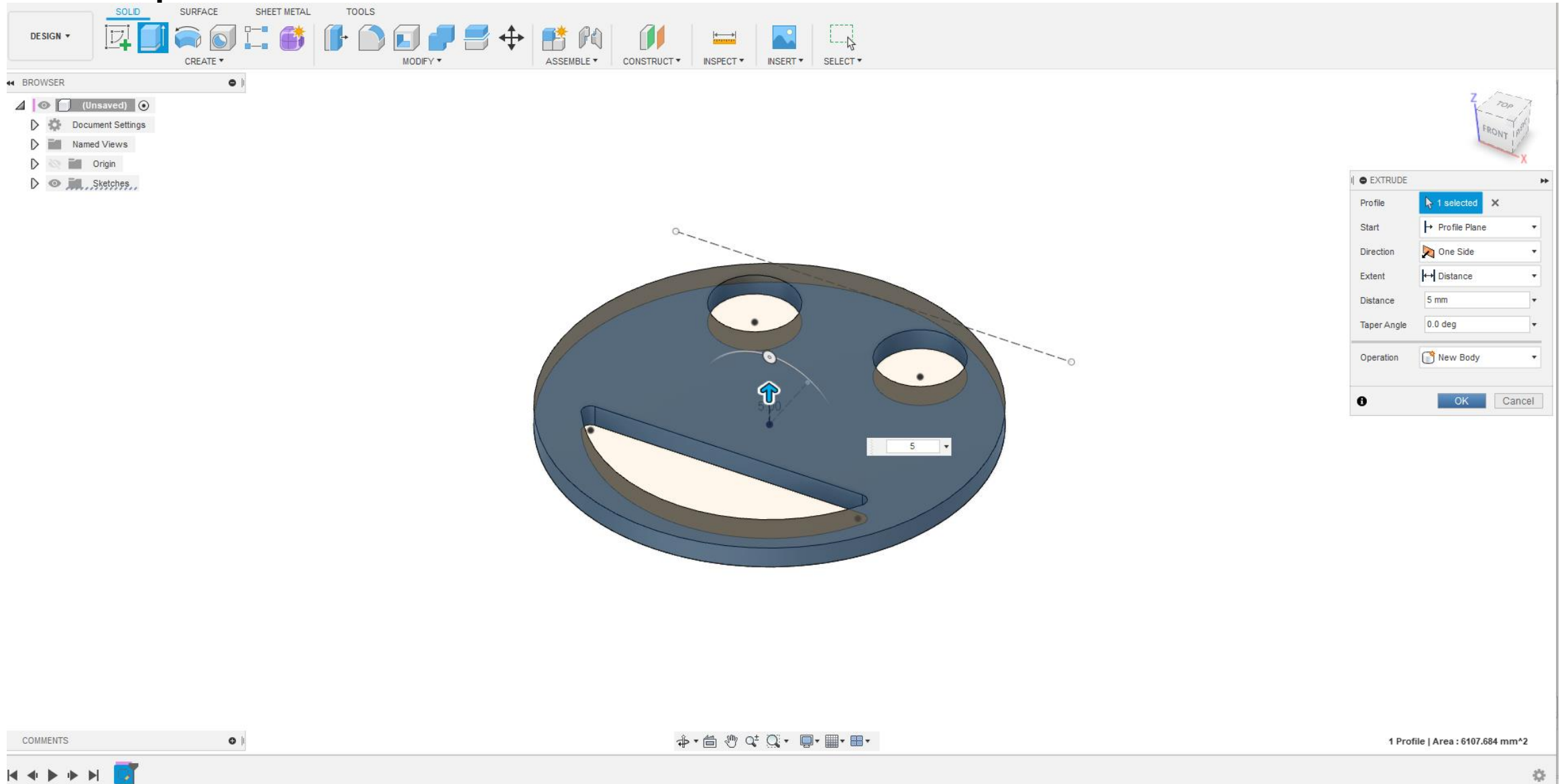
Körper aus Skizze erstellen



The image shows a CAD software interface with the following elements:

- Top Ribbon:** Categorized into SOLID, SURFACE, SHEET METAL, and TOOLS. Sub-sections include CREATE, MODIFY, ASSEMBLE, CONSTRUCT, INSPECT, INSERT, and SELECT.
- Browser Panel (Left):** Lists various CAD features such as New Component, Create Sketch, Create Form, Derive, Extrude, Revolve, Sweep, Loft, Rib, Web, Hole, Thread, Box, Cylinder, Sphere, Torus, Coil, Pipe, Pattern, Mirror, Thicken, Boundary Fill, Create Mesh, Create Mesh Section Sketch, Create Base Feature, and Create PCB.
- Extrude Tool Tip (Center-Left):** A dark grey box with white text that reads: "Adds depth to a closed sketch profile or planar face. Select the profile or planar face then specify the distance to extrude." Below the text is a 3D illustration of a rectangular block with a cylindrical hole extruded from its side.
- Sketch (Center):** A 2D sketch of a smiley face on a light orange background. The face consists of a large outer circle, two smaller circles for eyes, a vertical dashed line for a nose, and a curved line for a mouth. A dashed horizontal line with a double-headed arrow indicates the extrusion distance from the top edge of the face.
- Coordinate System (Top-Right):** A small 3D coordinate system with X, Y, and Z axes. The Z-axis is labeled "TOP".
- Bottom Panel:** Includes a "COMMENTS" field on the left and a set of navigation icons (pan, rotate, zoom, etc.) in the center.

Körper aus Skizze erstellen



The screenshot displays a CAD software interface with the following components:

- Top Ribbon:** Includes tabs for SOLID, SURFACE, SHEET METAL, and TOOLS. The SOLID tab is active, showing icons for CREATE, MODIFY, ASSEMBLE, CONSTRUCT, INSPECT, INSERT, and SELECT.
- Browser:** Located on the left, it shows a tree view with folders for Document Settings, Named Views, Origin, and Sketches.
- 3D Model:** A blue circular plate with two circular holes and a slot. A blue arrow points upwards from the center of the plate, indicating the extrusion direction. A dimension of 5 is shown near the arrow.
- EXTRUDE Dialog Box:** Open on the right side, it contains the following settings:
 - Profile: 1 selected
 - Start: Profile Plane
 - Direction: One Side
 - Extent: Distance
 - Distance: 5 mm
 - Taper Angle: 0.0 deg
 - Operation: New Body
- Bottom Panel:** Includes a COMMENTS section on the left, a navigation toolbar in the center, and a status bar on the right showing "1 Profile | Area : 6107.684 mm^2".

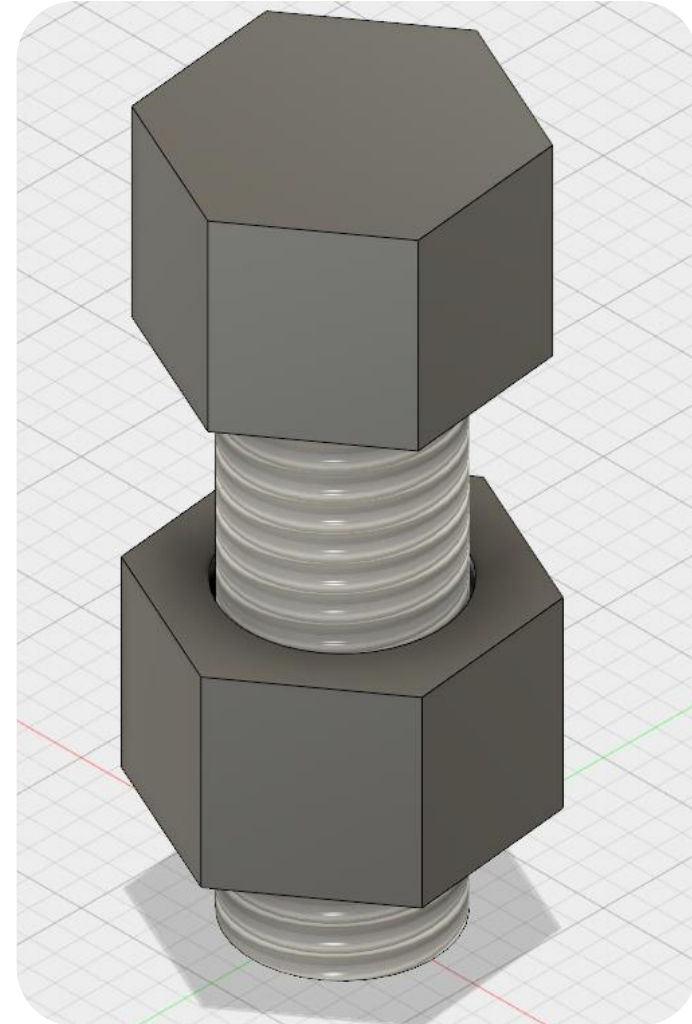
Ich habe es
vergessen!



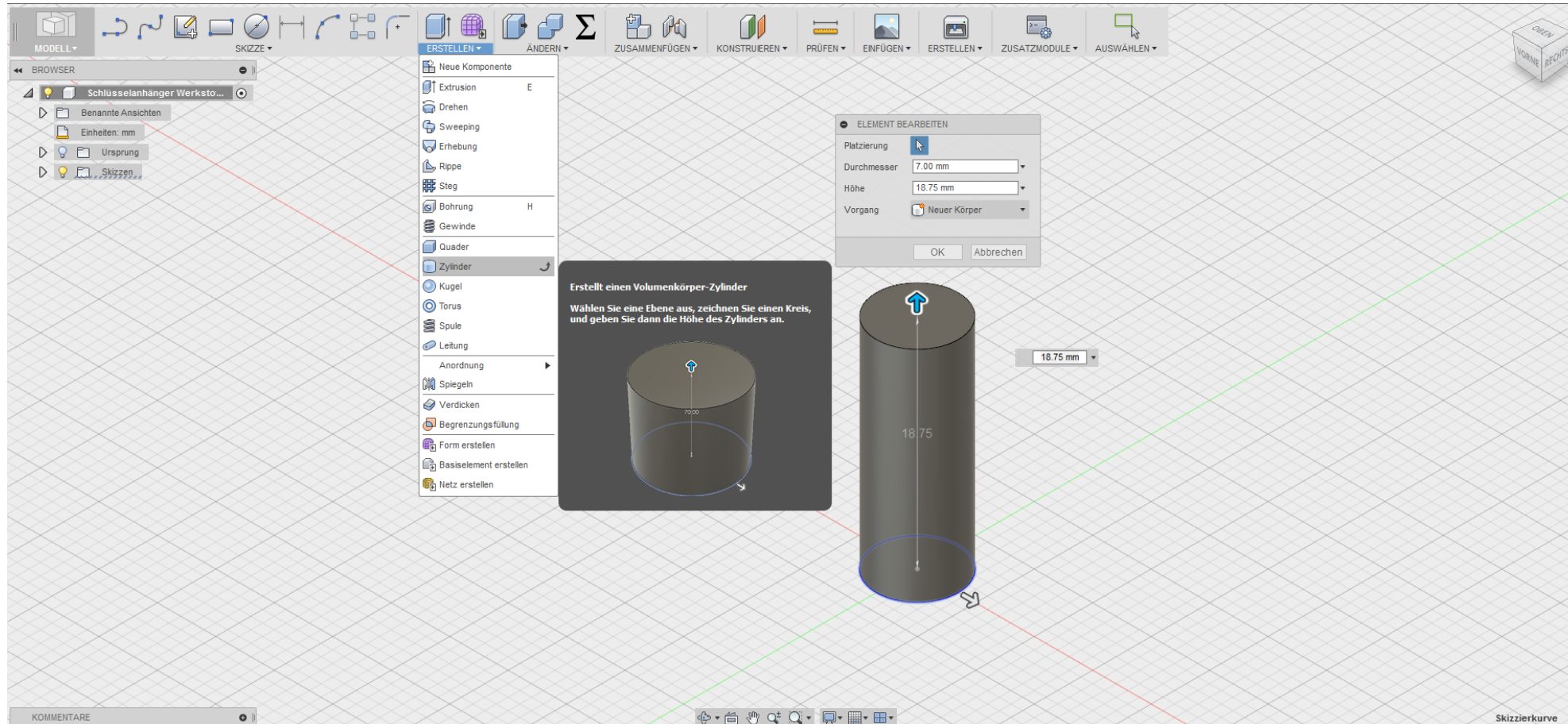
Fusion360 im Dreidimensionalen

Wir konstruieren eine Schraube

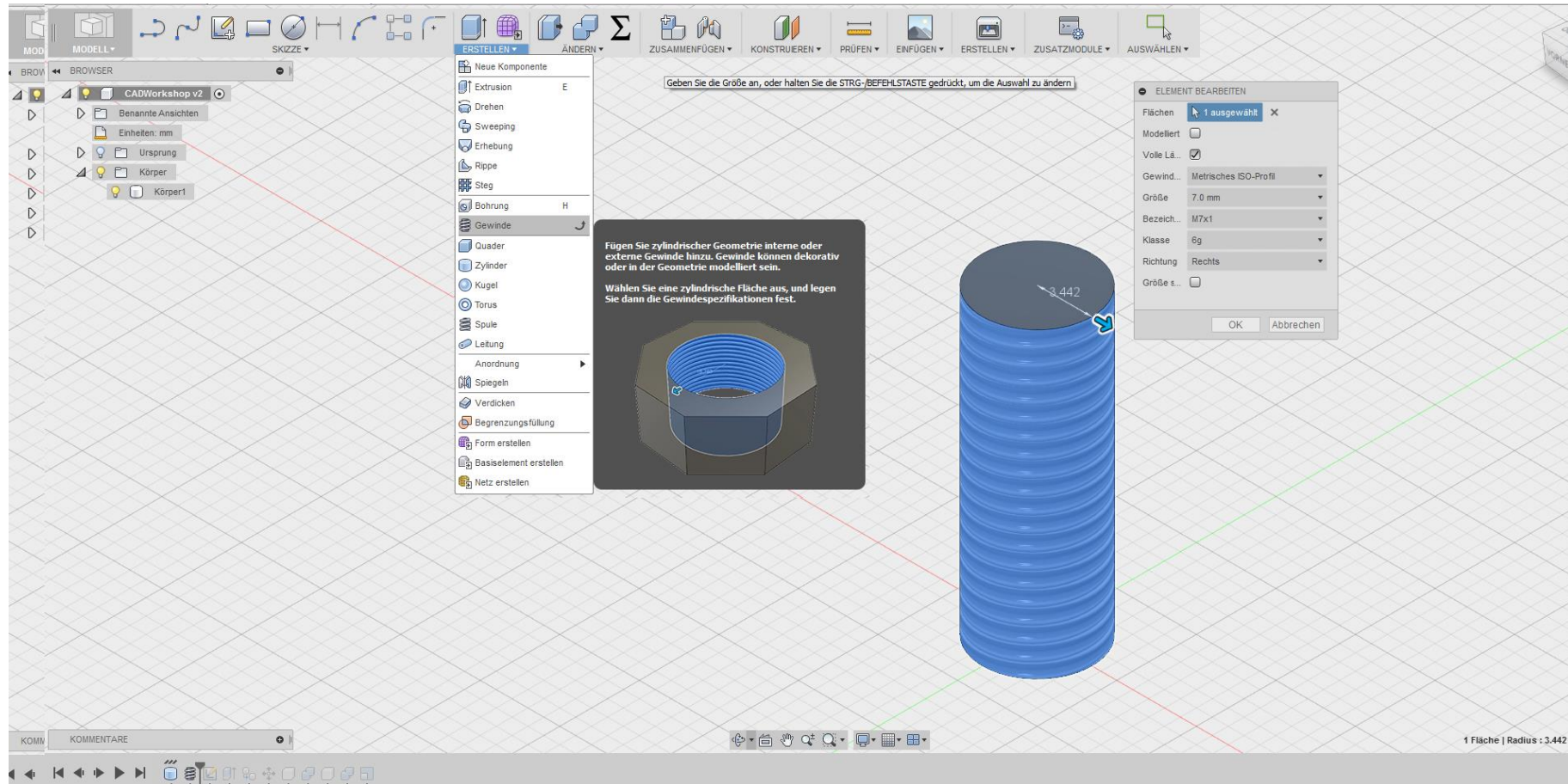
1. Schrauben Schaft erstellen (Zylinder)
2. Gewinde hinzufügen
3. Schraubenkopf erstellen (Polygon)
4. Schraubenkopf duplizieren
5. Schraubenelemente Joinen
6. Mutter erstellen
7. Mutter skalieren



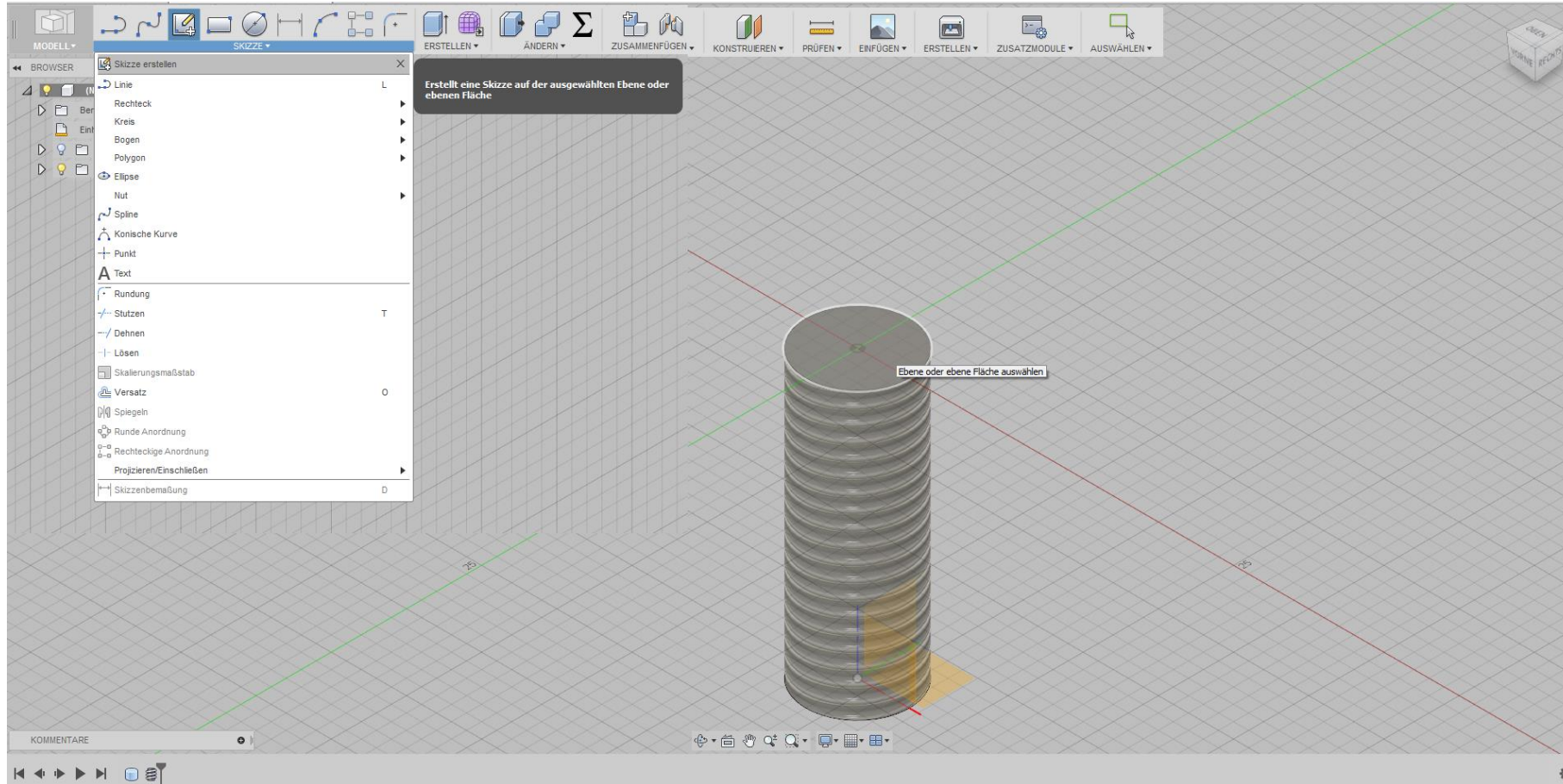
1. Schrauben Schaft erstellen



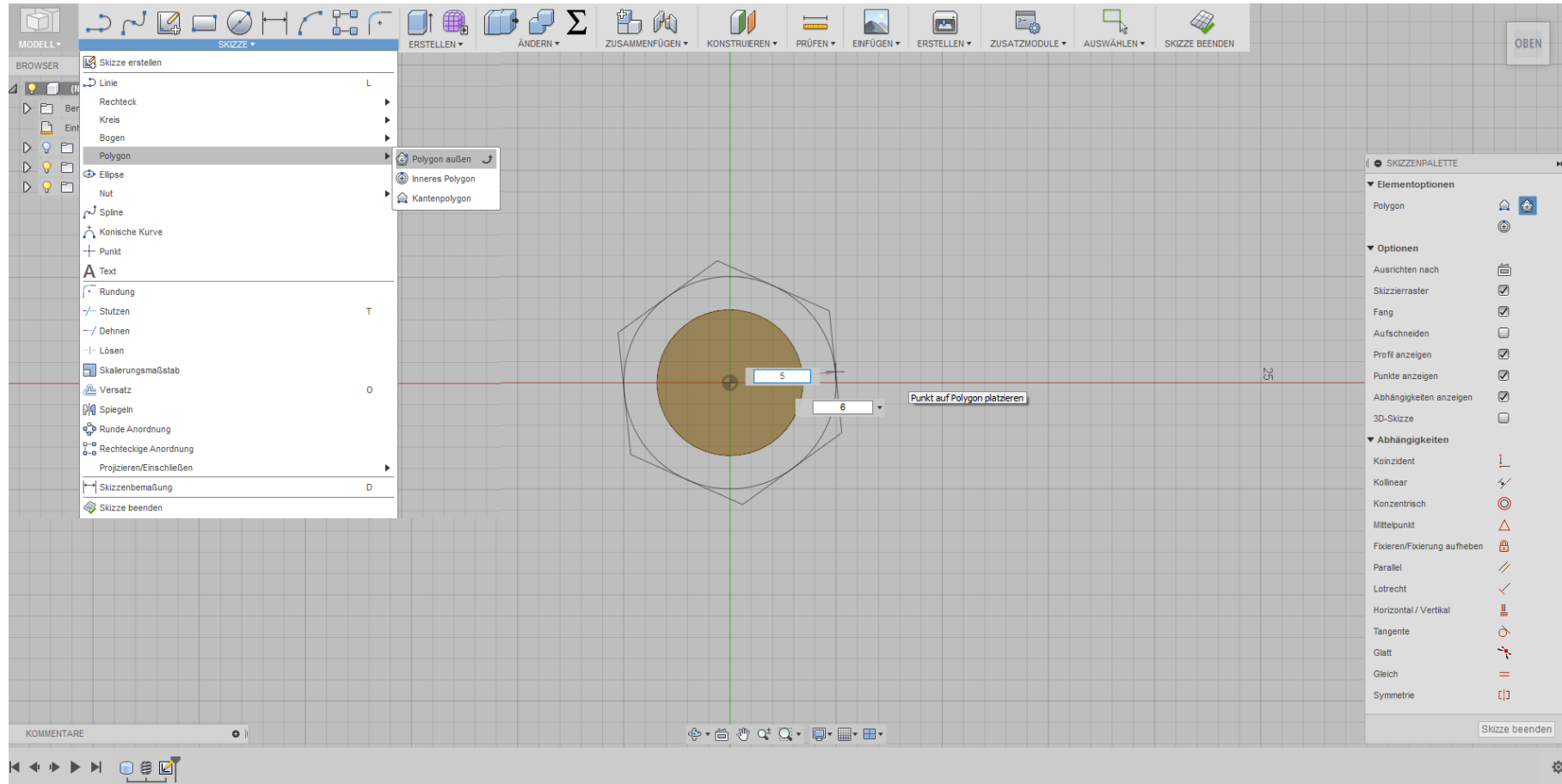
2. Gewinde hinzufügen



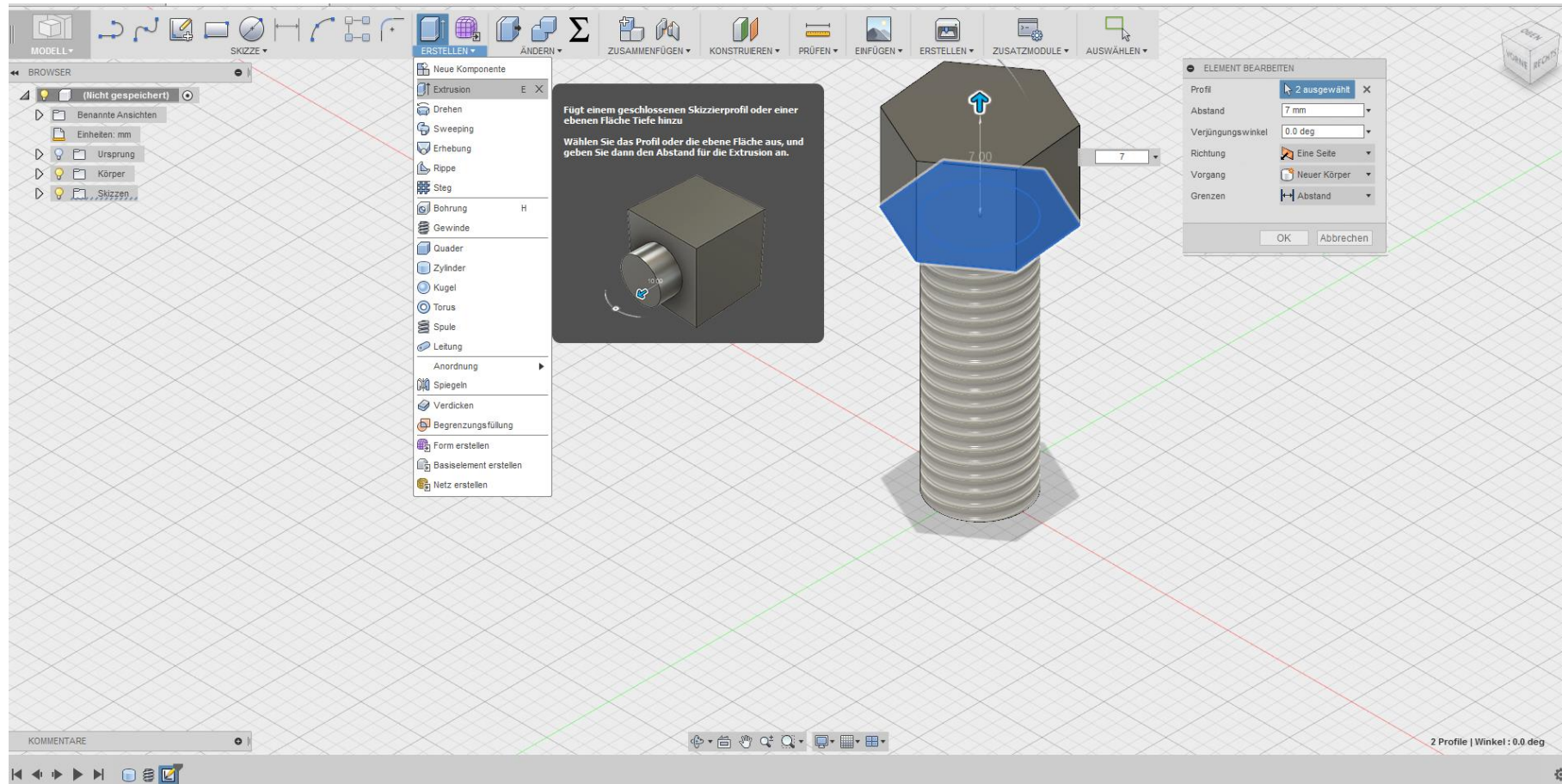
3. Schraubenkopf erstellen (Polygon)



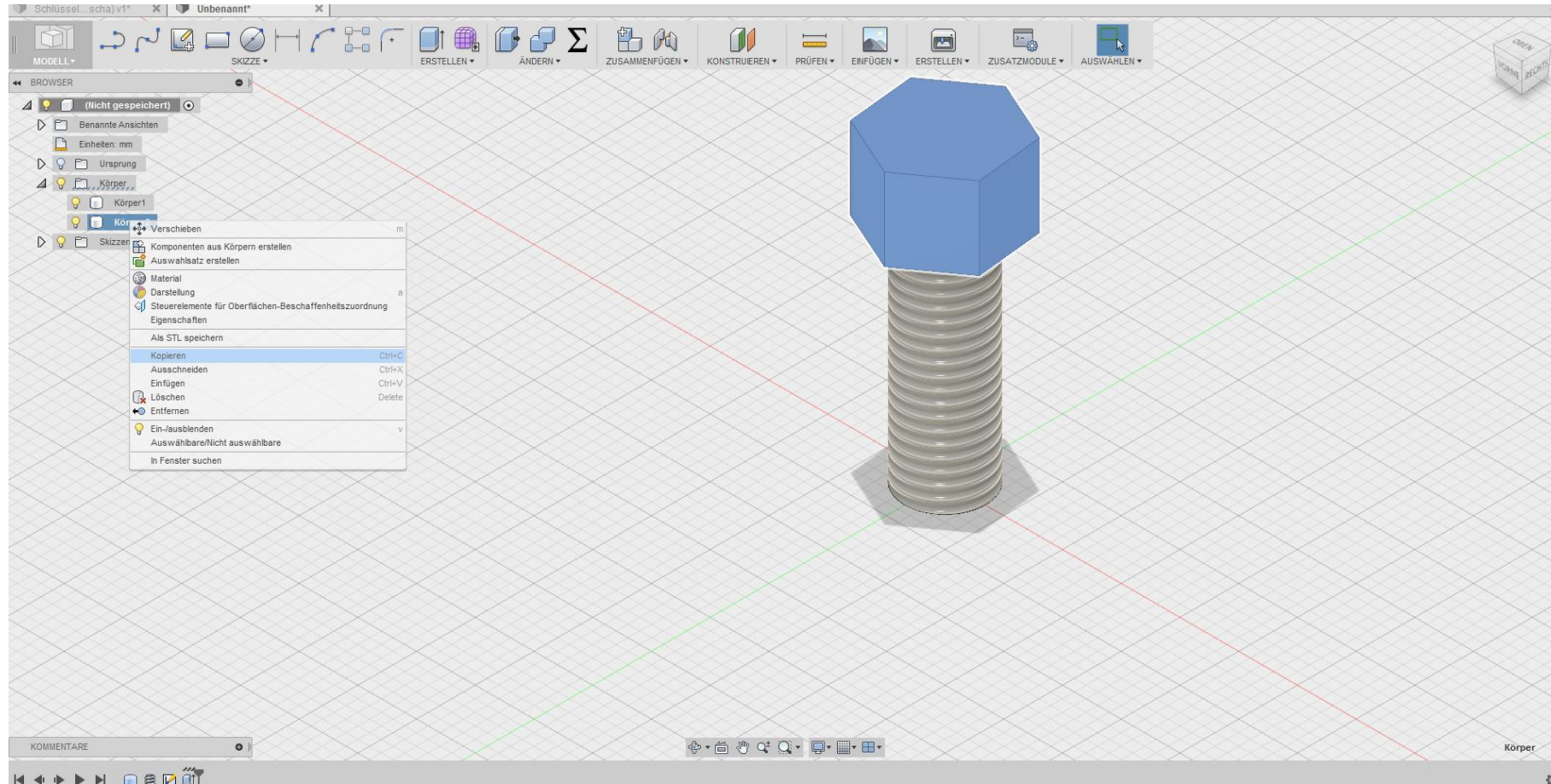
3. Schraubenkopf erstellen (Polygon)



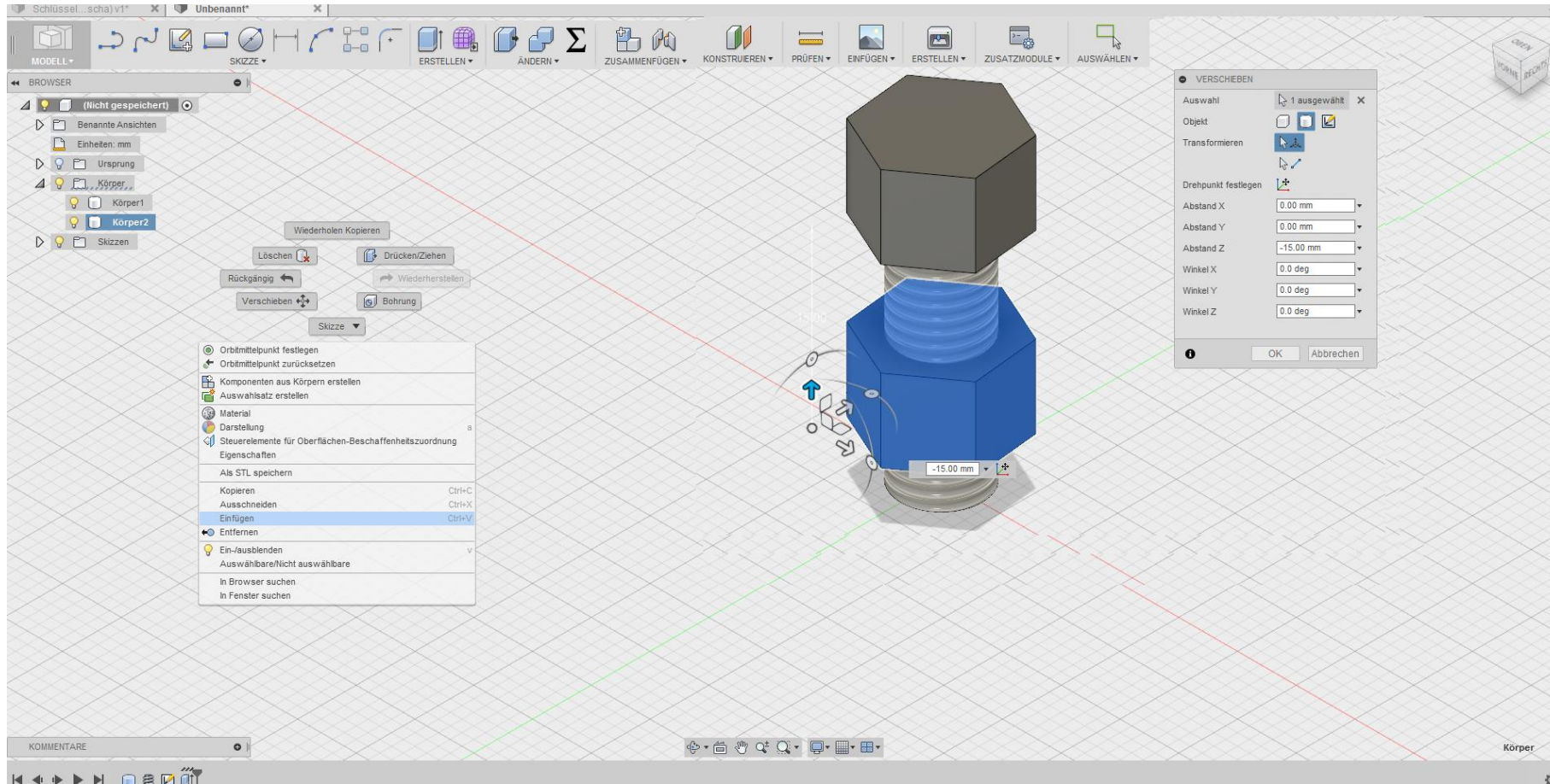
3. Schraubenkopf erstellen (Polygon)



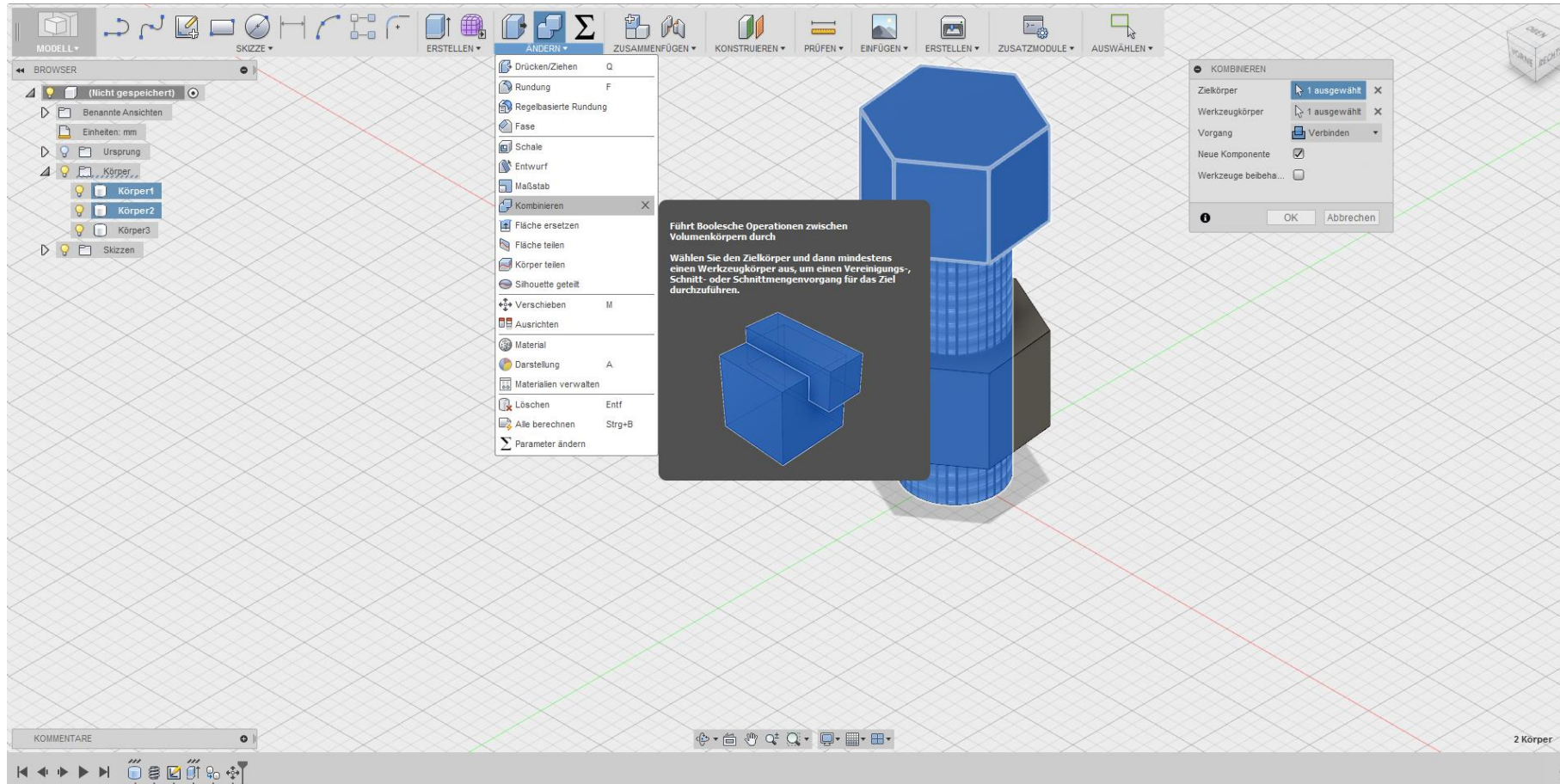
4. Schraubenkopf duplizieren



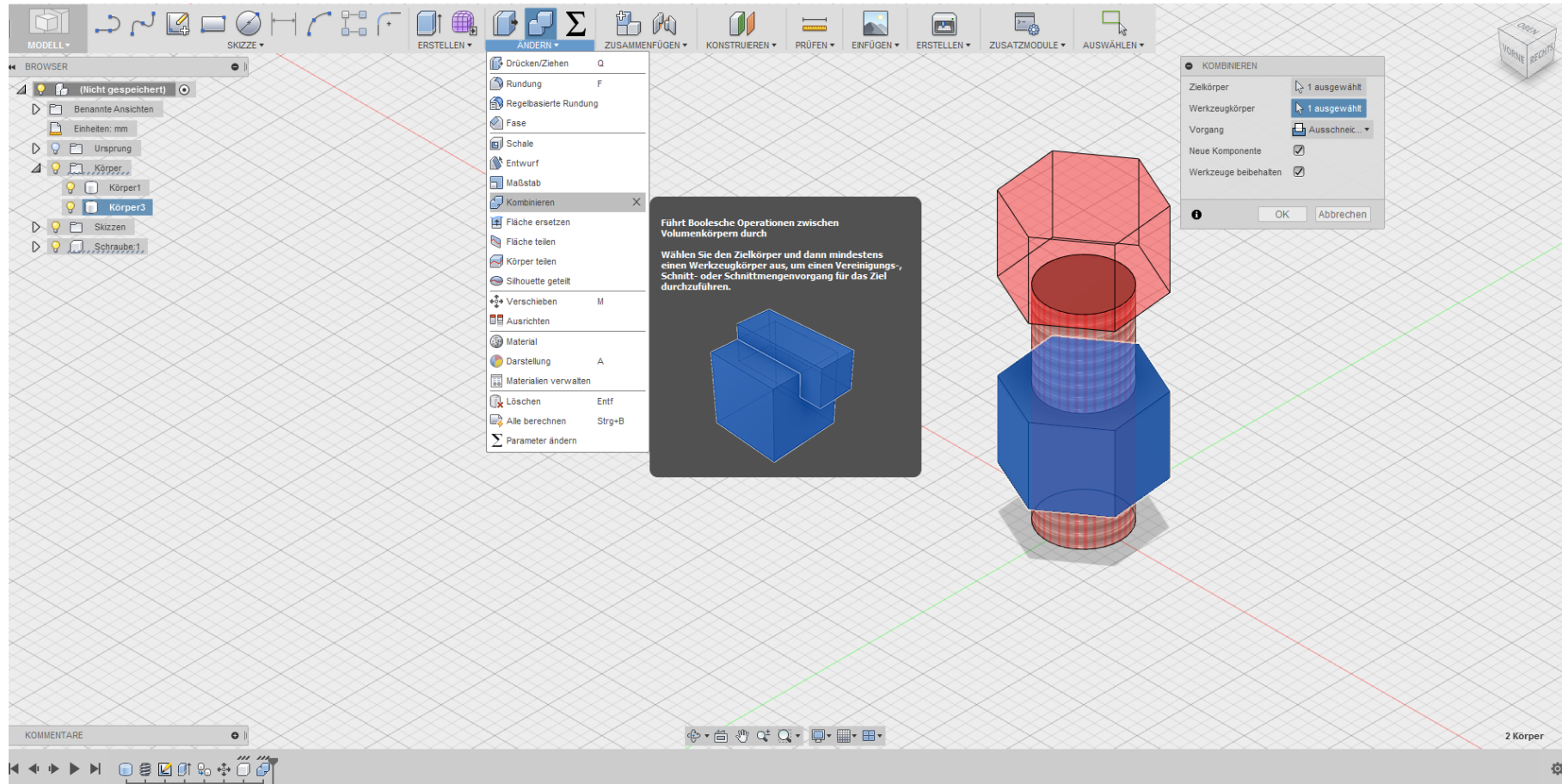
4. Schraubenkopf duplizieren



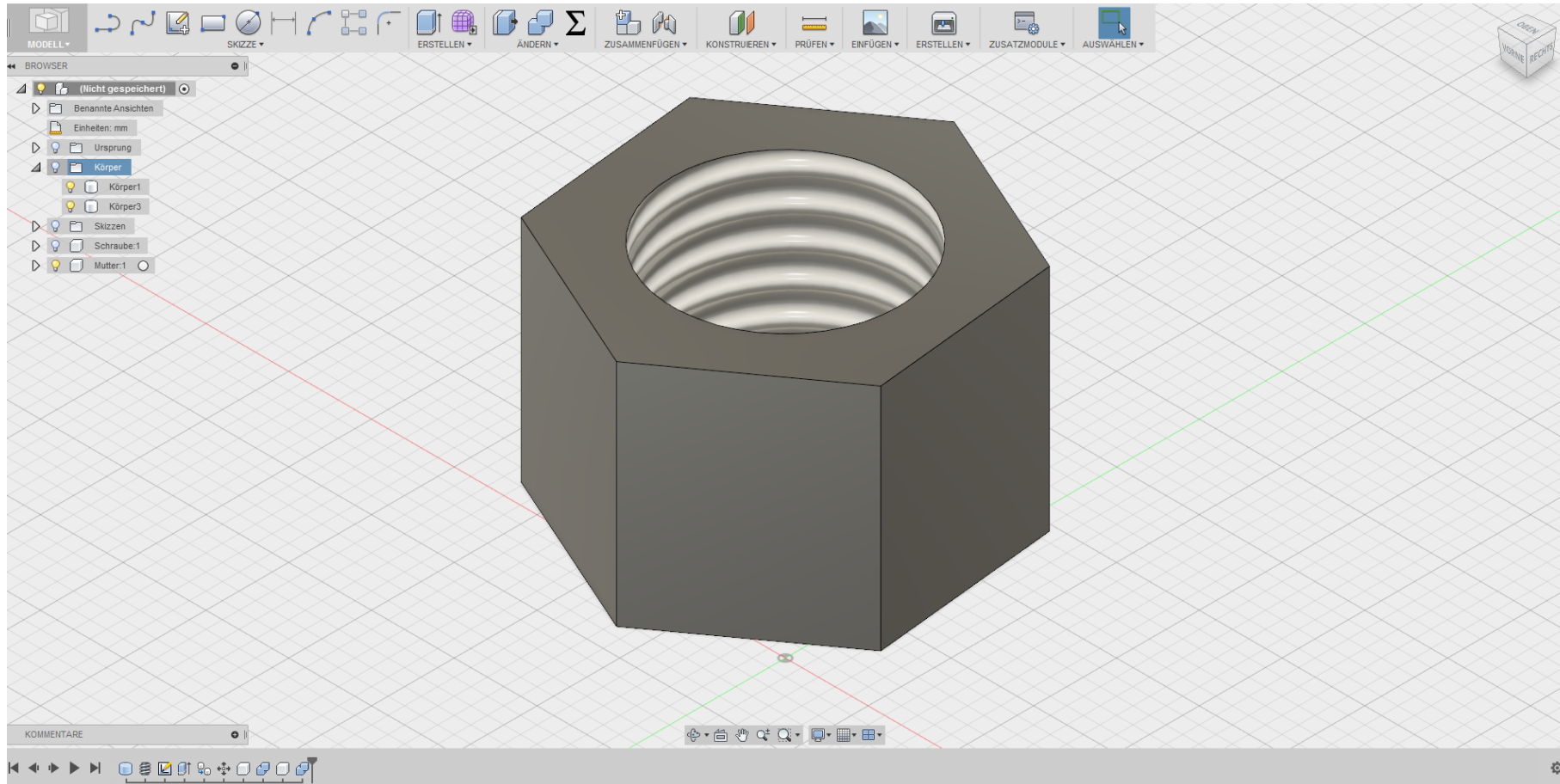
5. Schraubenelemente Joinen



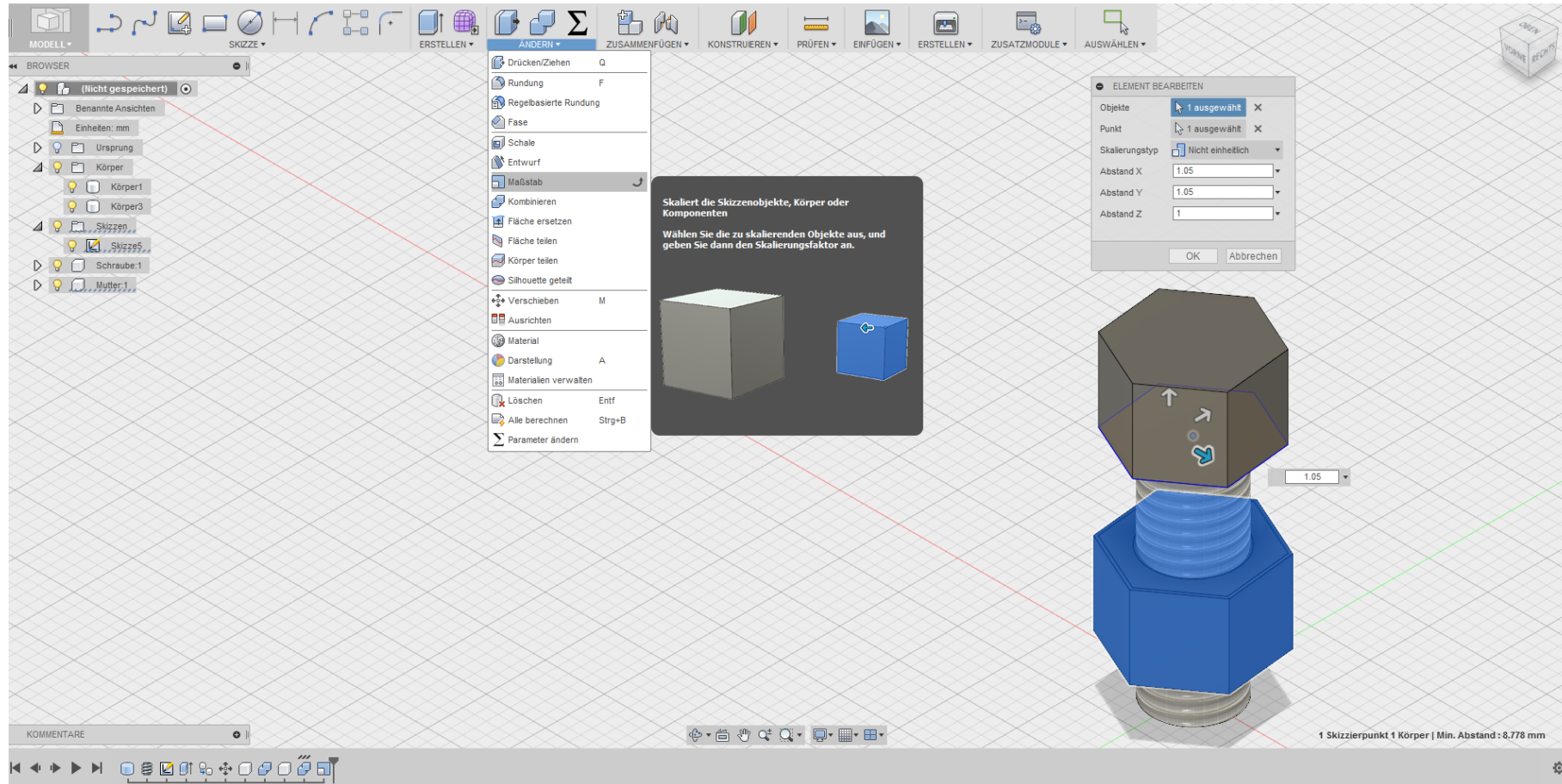
6. Mutter erstellen



6. Mutter erstellen


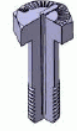





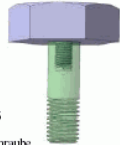

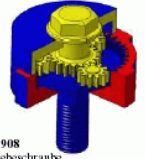



7. Mutter skalieren



Sonderschrauben

 <p>DIN 875 Für versetzte Löcher</p>	 <p>DIN 876 Bei einseitig fehlendem Platz für Schraubenkopf</p>	 <p>DIN 877 Für zu groß gesenkte Löcher</p>
 <p>DIN 878 Für schräg gebohrte Löcher</p>	 <p>DIN 879 Für Löcher die auf der falschen Seite angesenkt wurden</p>	 <p>DIN 880 Schrauben in Feldsicher Form für doppelt gebohrte Löcher</p>
 <p>DIN 881 Montageschraube für zu große Löcher</p>	 <p>DIN 882 Montageschraube für zu tief gesenkte Bohrungen</p>	 <p>DIN 883 Sonderschraube zur Verringerung der Montagezeit</p>
 <p>DIN 884 Teleskopschraube wenn Zweifel über die Länge bestehen</p>	 <p>DIN 885 Für wechselnde Winkelfehler</p>	 <p>DIN 886 Für Schlüsselweite 13, 17 und 19</p>

 <p>DIN 899 Flügelkorkschraube</p>	 <p>DIN 900 Drehmomentschraube</p>	 <p>DIN 901 Rechtsgewindeschraube für Linksgewinde</p>
 <p>DIN 902 Noppenkernungs - schraube (M10) für blinde Mitarbeiter</p>	 <p>DIN 903 Bohrsenkgewindeschneidschraube</p>	
 <p>DIN 904 Rohrzangen - kopschraube</p>	 <p>DIN 905 Zwillingschraube</p>	<p>Schraubenkopf auswechselbar</p>  <p>DIN 906 Vario- mogelschraube zum Vortäuschen stabiler mechanischer Verbindungen</p>
 <p>DIN 907 Ausweichschraube</p>	 <p>DIN 908 Getriebeschraube nur in Verwendung mit Getriebeschraubenschlüssel</p>	 <p>DIN 909 Sonderschraube mit Passfeder als Ausdrehsicherung</p>

 <p>DIN 887 Sonderschraube für M5 - M10</p>	 <p>DIN 888 Wie DIN 887 M5 - M10 jedoch Auch für Zoll und seltene Zwischengößen z.B. M8.7</p>	 <p>DIN 889 Für Gabel und Ringschlüssel von SW12 - 17</p>
 <p>DIN 890 Flügel - 6Kt - Schlitz - Inbus - Torx Kreuzschlitz - Schraube</p>	 <p>DIN 891 Magnet - Schraubenkopf Zum schnellen Vortäuschen einer Verschraubung</p>	 <p>DIN 892 Magnet - Schraubenkopf Zum schnellen Vortäuschen einer Verschraubung auch von hinten</p>
 <p>DIN 893 Sonderschraube mit Vorbereitung für eine Notsprennung</p>	 <p>DIN UNF Einwegschraube (Ratscheneffektschraube)</p>	 <p>DIN 895 Räumschraube Zum Säubern von Bohrungen</p>
<p>Abziehhilfe f. doppels. Klebeband</p>  <p>DIN 896 Klebe - Schraubenkopf Zum schnellen Vortäuschen einer Verschraubung für alle nichtmagnetischen Werkstoffe</p>	<p>Abziehhilfe f. doppels. Klebeband</p>  <p>DIN 897 Wie DIN 896 jedoch auch von hinten</p>	 <p>DIN 898 Schlagschraube Bei Bohrungen ohne Gewinde</p>