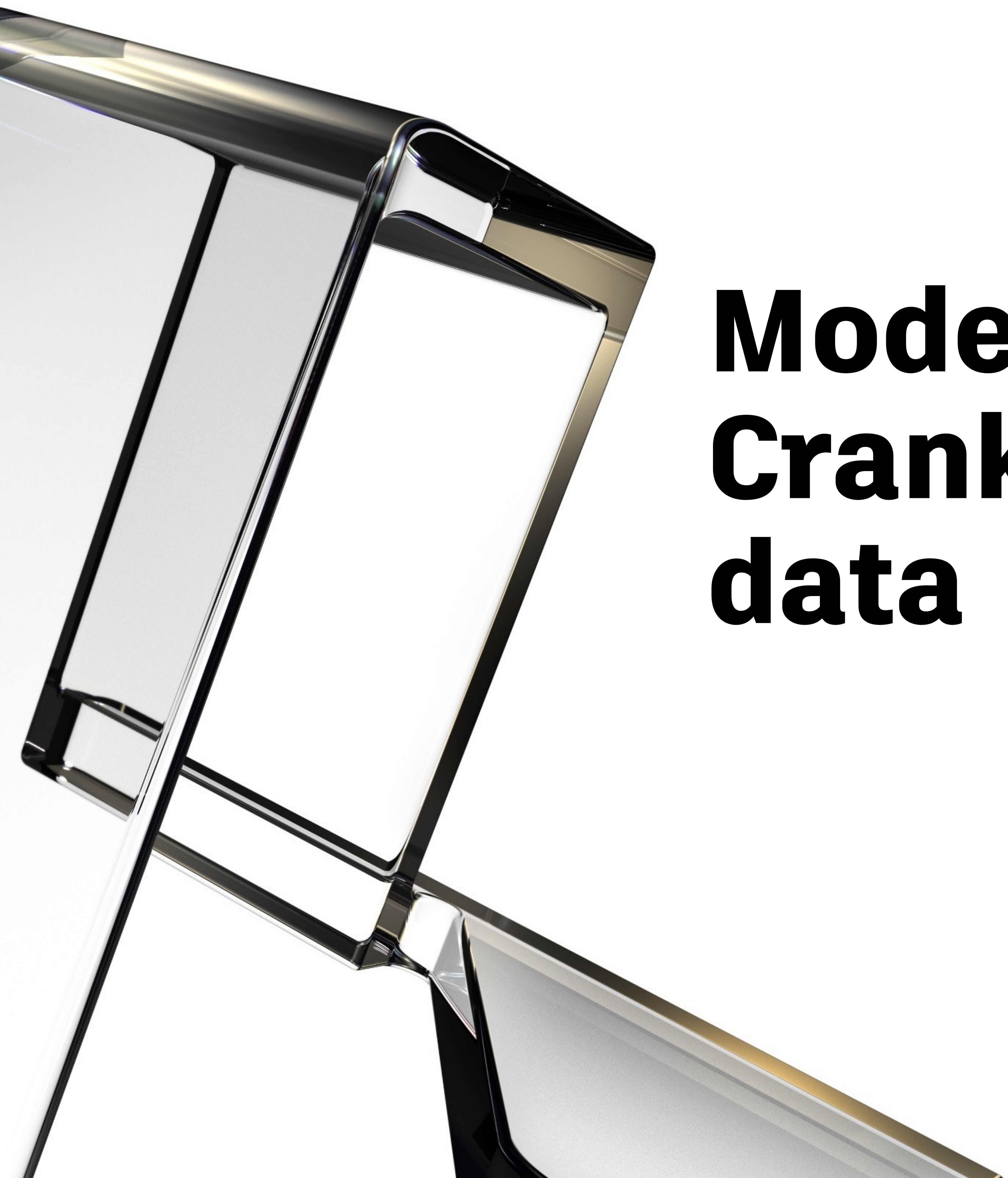


# Model engine: Crank soft jaw cutting data

# Learning objectives

After completing this module, you'll be able to:

- Identify the tools used to machine the part
- Identify cutting data for machining applications
- Define common cutting data terms
- Calculate chip thickness and chip load
- Calculate feeds and speeds for a tool
- Define surface footage
- Explain cutting forces and cutter engagement
- Follow general guidelines

# What tools were used to machine the crank soft jaw?

- T1 – 1/4” 3 Flute Flat Carbide Endmill .75” Flute Length

# Common terms

- **Cutting feed rate (in/min)**: The linear speed at which a tool is moving.
- **Cutter force**: The resistance of the material against the intrusion of the cutting tool.
- **Cutter-workpiece engagement (CWE)**: the instantaneous contact geometry between the cutter and the in-process workpiece during machining. It plays an important role in machining process simulation and directly affects the calculation of the predicted cutting forces and torques.
- **Feed per tooth (in)**: Amount of material each tooth on a tool will remove per revolution. Also known as **Chip Load.**

# Common terms continued

- **Plunge/Lead-in/Lead-out/Ramp**: Various types of tool movements that often require a variation to the cutting federate.
- **Spindle Speed (RPM)**: Revolutions per Minute or the speeds at which the tool will rotate
- **Stepdown**: The amount a tool will engage material vertically and is typically a ratio of the tool diameter based on specific cutter movements.
- **Surface footage**: Linear feet a rotating cutter travels in one minute at a given RPM.

# Calculating feeds and speeds

A baseline federate and spindle speed is often acquired from a tool manufacturer. This value will be based on the tool diameter, number of cutting flutes and their geometry, tool coatings, material to be cut, rigidity of the workholding and type of material that is being cut.

- A ½" Diameter 3 flute Endmill might give you a range of values.
  - Depth of cut = 2.0 x Diameter
  - Side cut = 0.1 x Diameter
  - Cutting Speed = 820 – 3250 SFM(surface feet per minute) for Non-ferrous materials
  - Feed Per Tooth for side milling = 0.0045 (IPT)
  - Feed Per Tooth for slotting = 0.0045 * 0.2 = 0.0009 (IPT)
- General formulas
  - SFM: 0.262 x RPM x Tool Diameter
  - RPM: (3.82 / SFM)/ Tool Diameter
  - IPM: # of teeth x Inch Per Tooth x RPM
  - IPT: IPM(RPM x # of teeth)
  - Inch(in) to millimeter (mm): Inch x 25.4
  - Millimeter (mm): to inch(in) : mm/25.4 or mm x 0.03937

# Calculating feeds and speeds (continued)

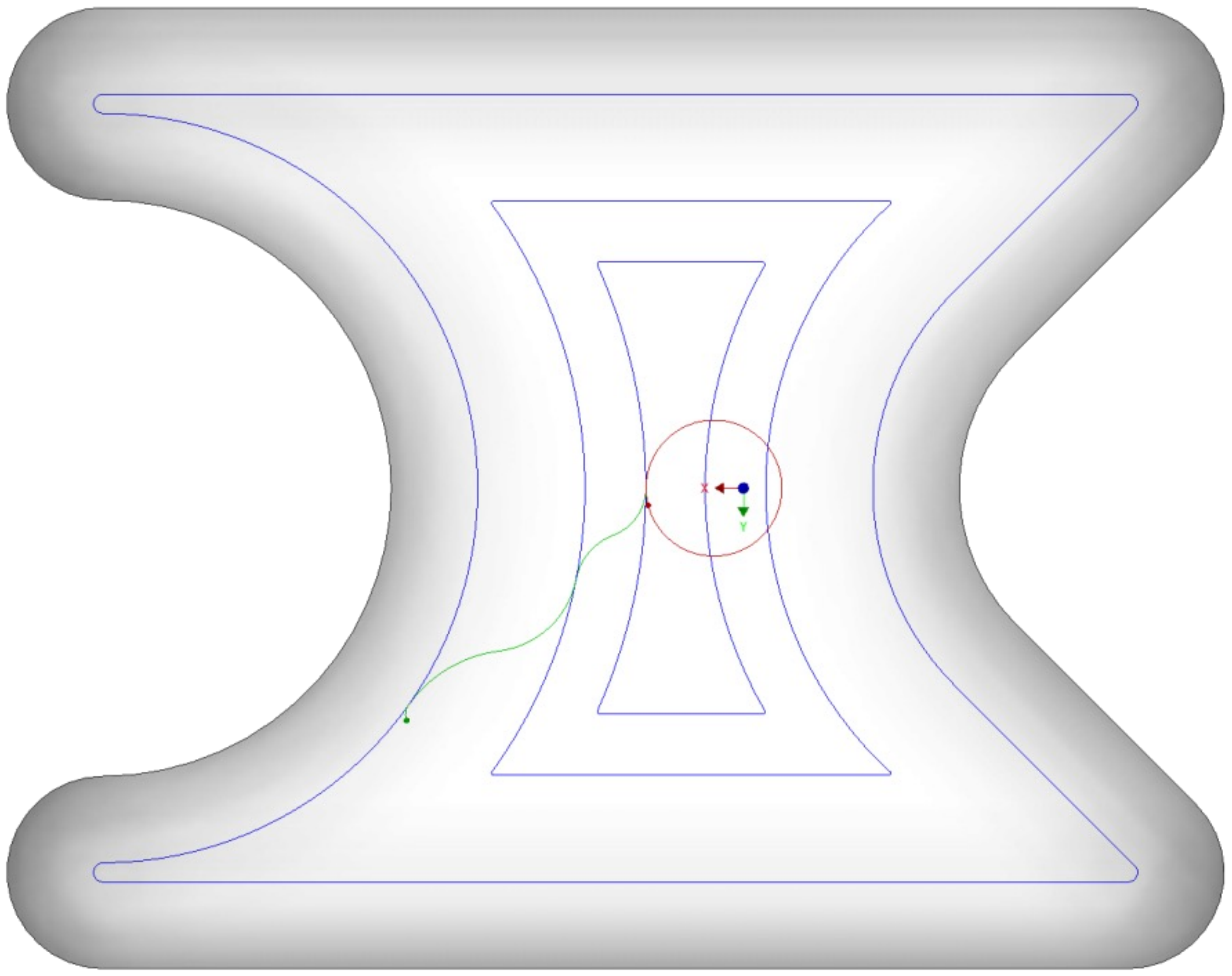
- Baseline values in combination with a specific machine should be used as a starting point and adjusted up/down at the machine control to influence future programs.

- Test cuts in the same material will help ensure efficient cutting practices.
  - Sample ½” Diameter 3 flute Endmill.
    - Depth of cut = 2.0 x Diameter
    - Side cut = 0.1 x Diameter
    - Cutting Speed = 820 – 3250 SFM(surface feet per minute) for Non-ferrous materials
    - Feed Per Tooth for side milling = 0.0045 (IPT)
    - Feed Per Tooth for slotting = 0.0045 * 0.2 = 0.0009 (IPT)

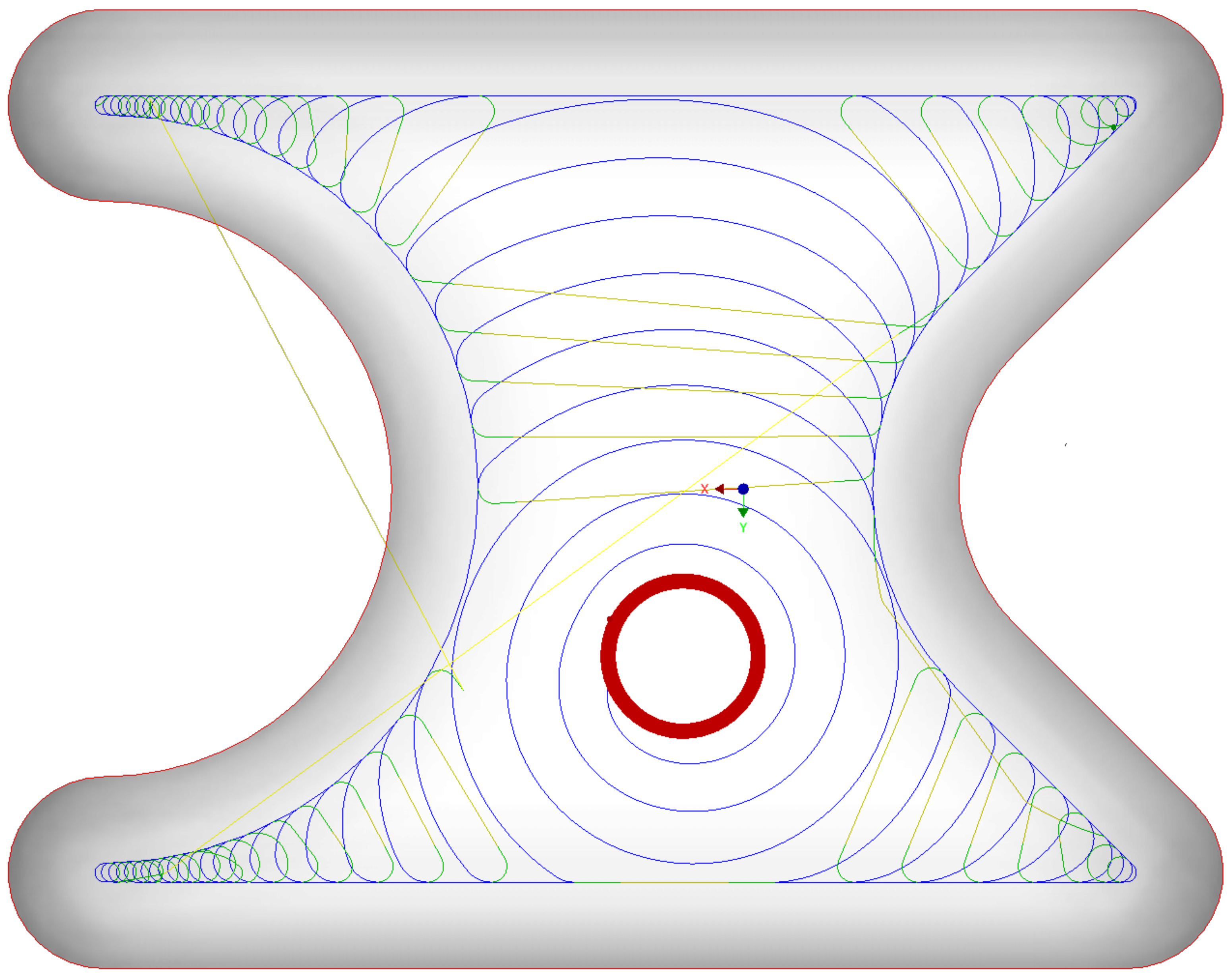
- Calculating Feed rate to be run at 7500 RPMS:
  - IPM = # of teeth x IPT x RPM
  - IPM = 3 x 0.0045 x 7500
  - IPM = 101.25 Inches per minute feed rate

  Note: The feeds and speeds examples and calculations for this guide are designed for aluminum. If other materials such as steels are used, speeds and feeds calculations will need to be adjusted.

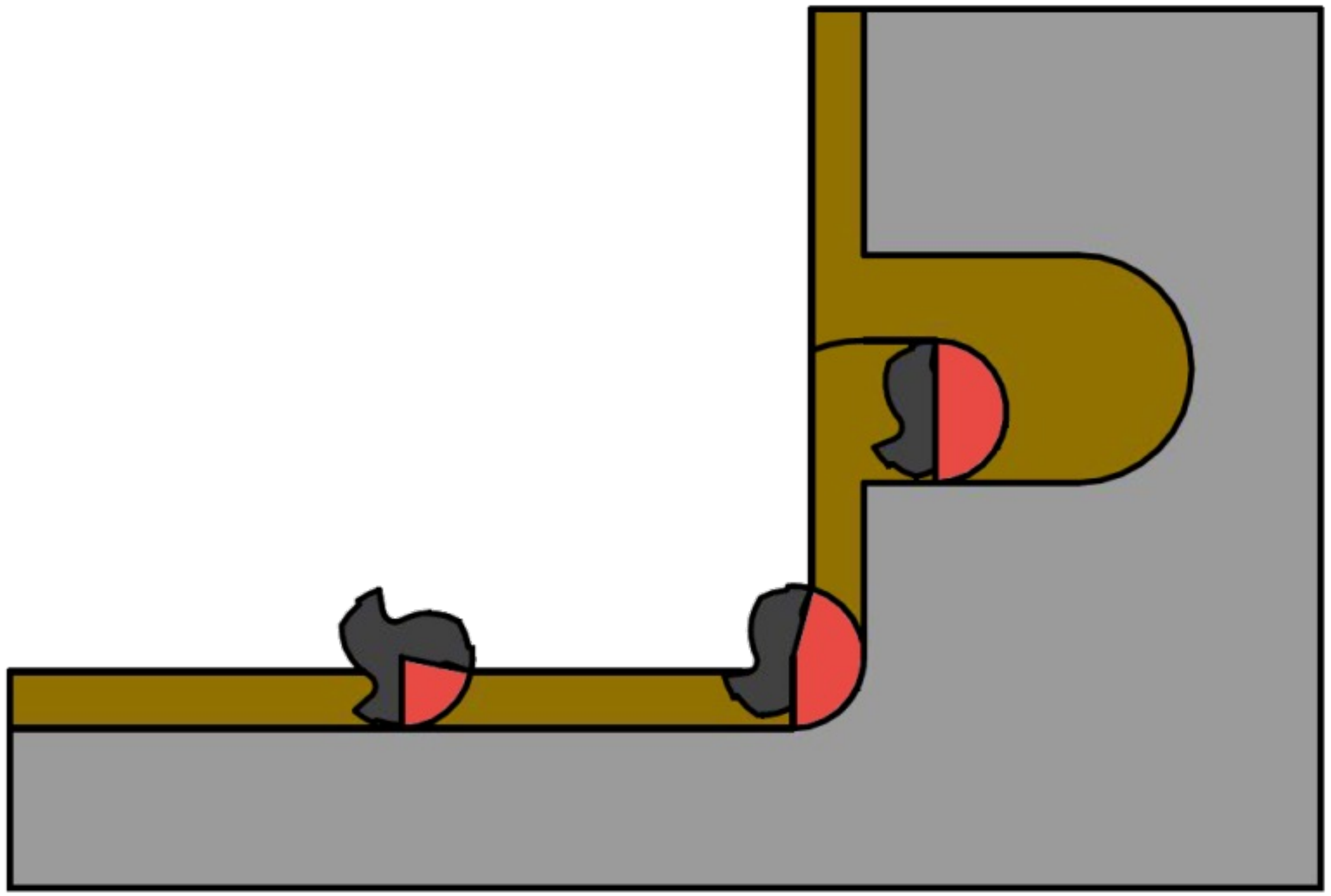
# Traditional Roughing



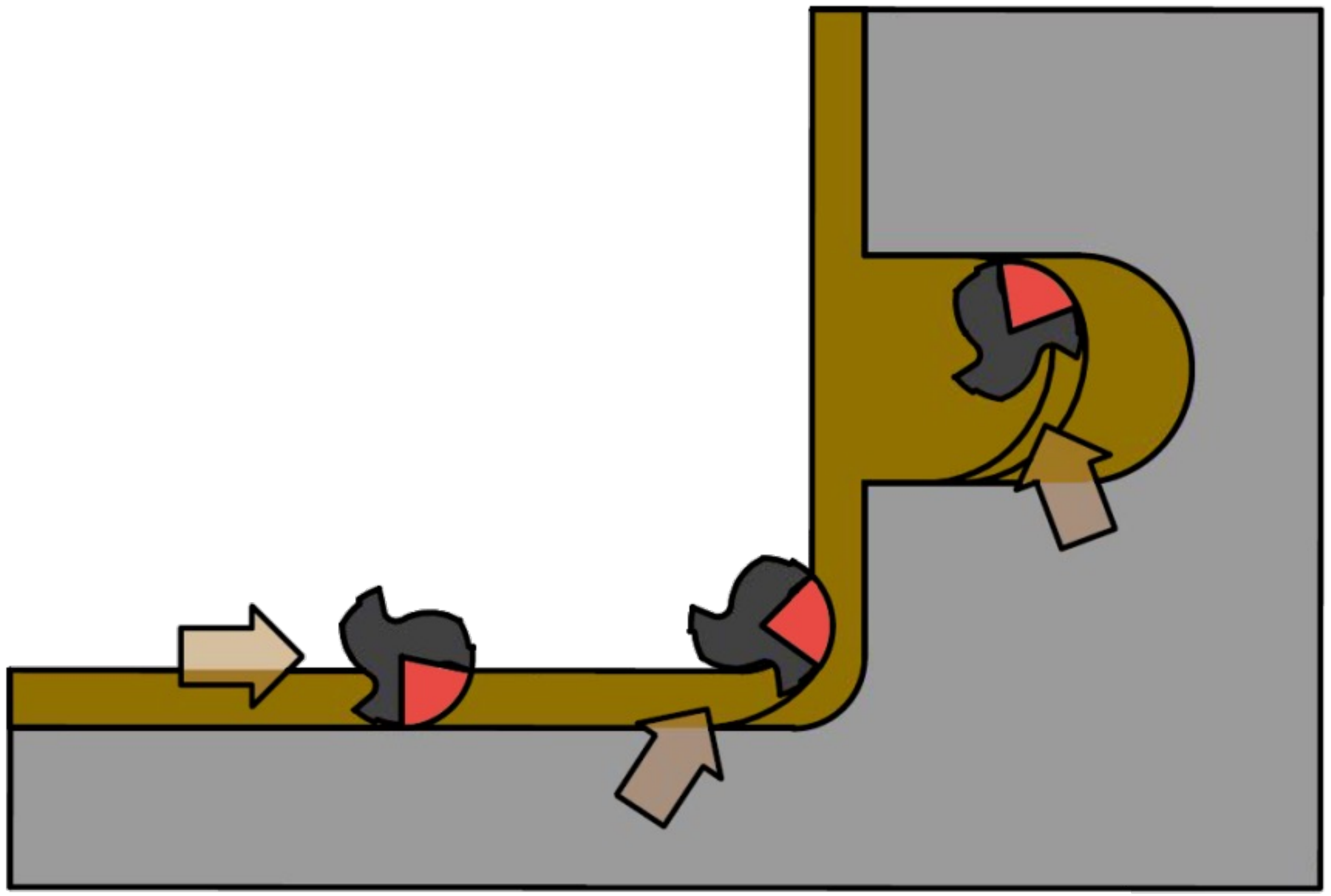
# High Efficiency Roughing



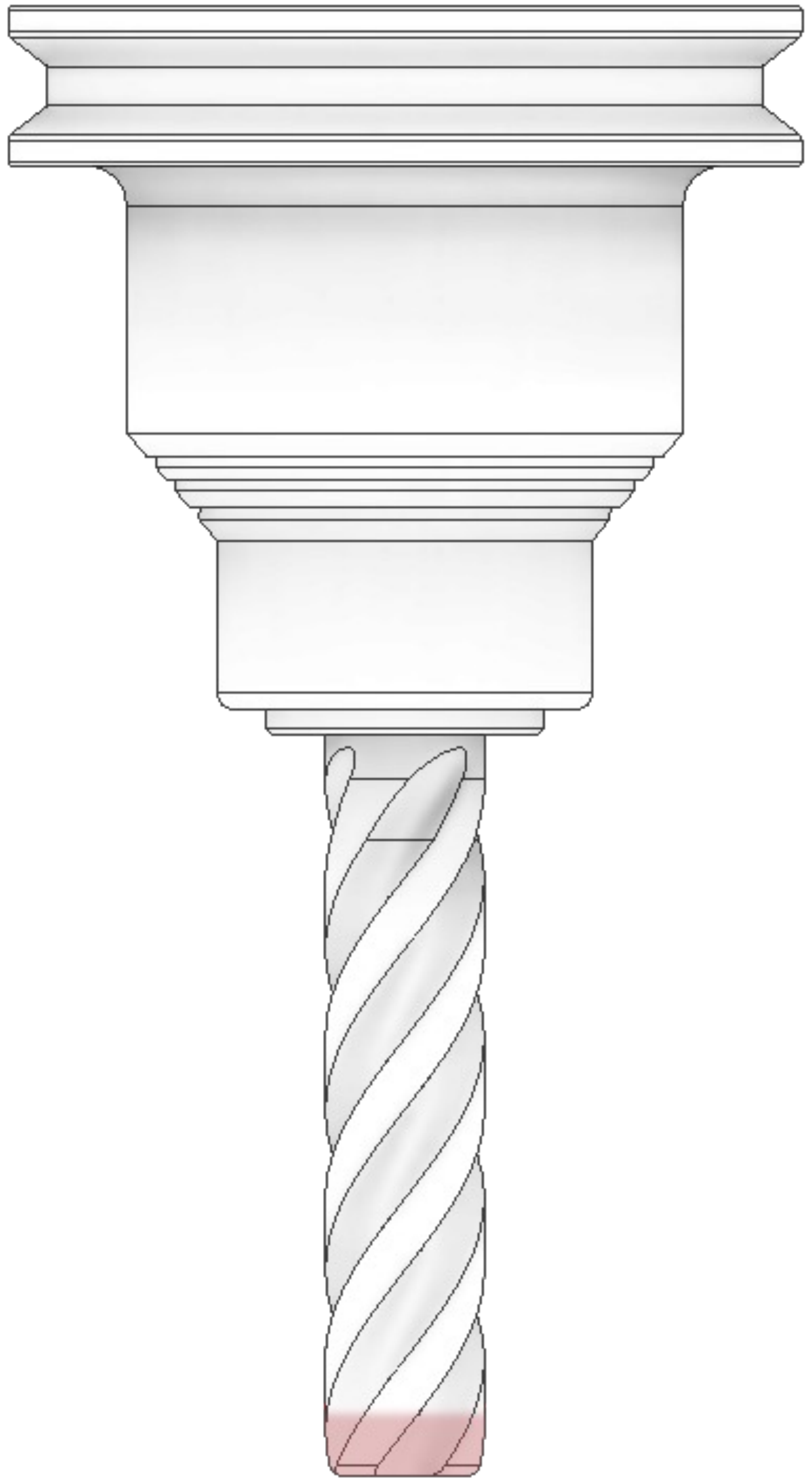
# Traditional Roughing



# High Efficiency Roughing

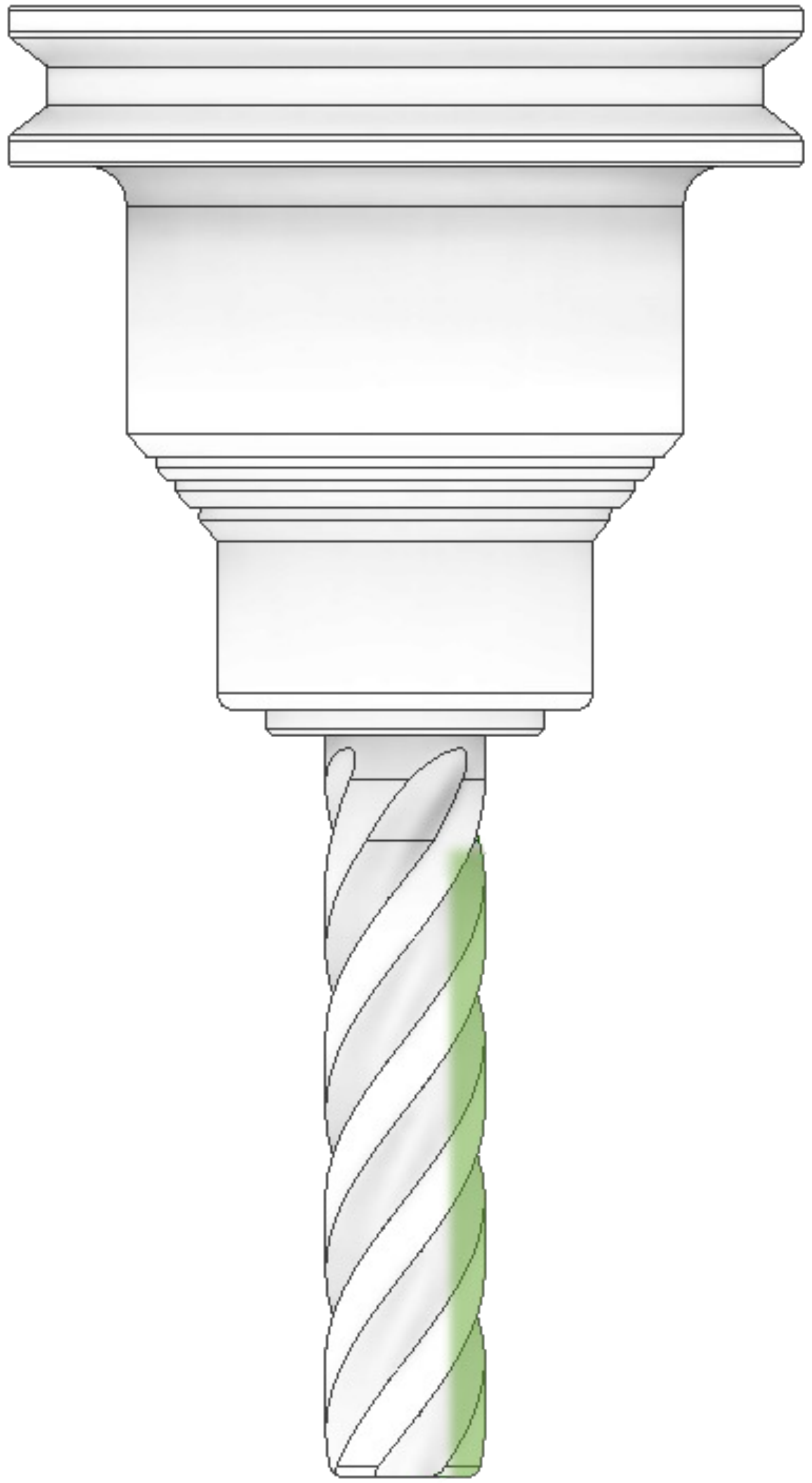


# Traditional Roughing



- Load spikes in internal corners
- Axial depth is of cut is the only constantly controlled parameter

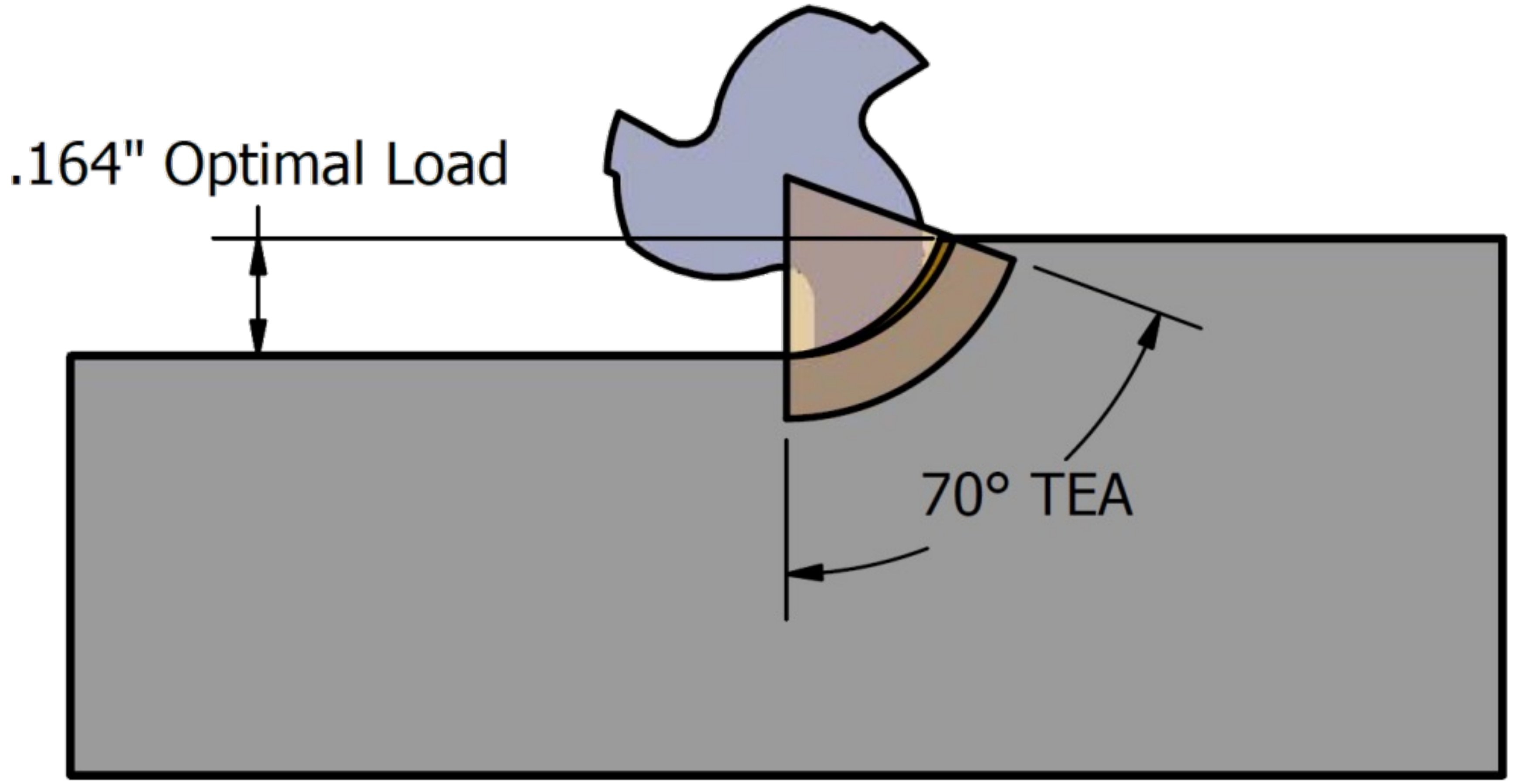
# High Efficiency Roughing

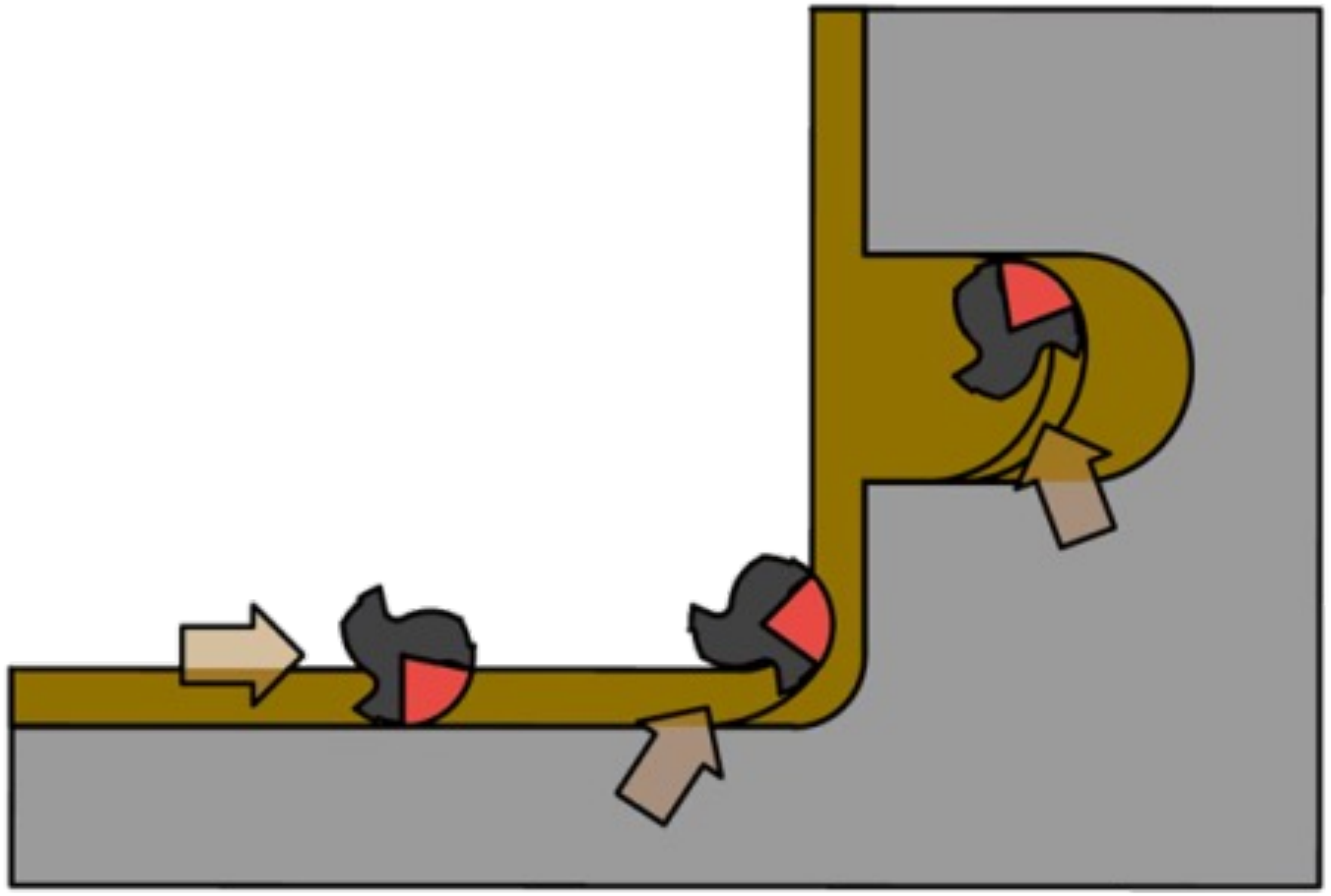


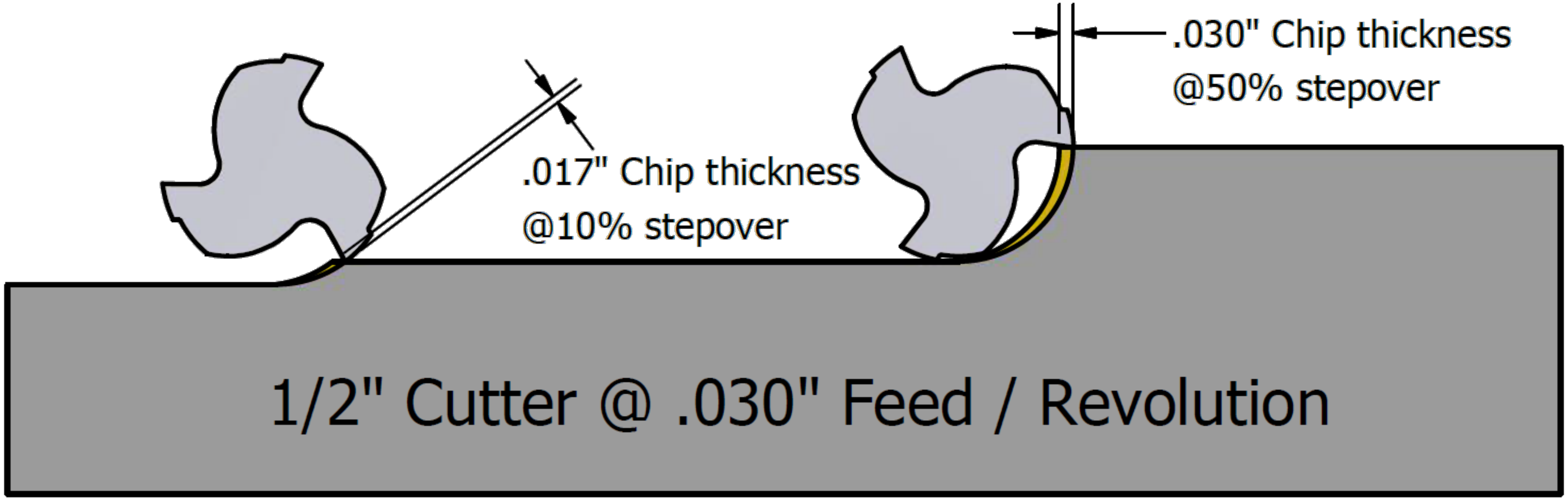
- Use the entire tool!
- Full control over Radial and Axial depth of cut

# TEA vs Optimal Load

## Tool Engagement Angle






# Radial Chip Thinning

- Light Radial depth of cut reduces the effective chip thickness

# Safety rules

- Never use any equipment which you have not been trained to operate by a qualified person.
- Wear ANSI approved safety glasses or ANSI safety approved glasses with side shields. You must wear safety glasses at all times in the shop, not just when at the machine.
- Do not wear long sleeve shirts because these could get caught in equipment. Wear short sleeves or T-shirts. Remove rings and watches when at the machine.
- Never wear gloves as they can be caught in the machine. Latex gloves are acceptable.
- Long hair should be tied back or under a hat to prevent it being caught in the machine spindle.
- Know where your hands are at all times.
- Always be aware of what could happen if your hand slips. For example, when tightening a bolt, think about what would happen if the wrench slipped. Would your hand or arm contact a tool? A pile of sharp chips?
- No horseplay or practical jokes are allowed in the shop.