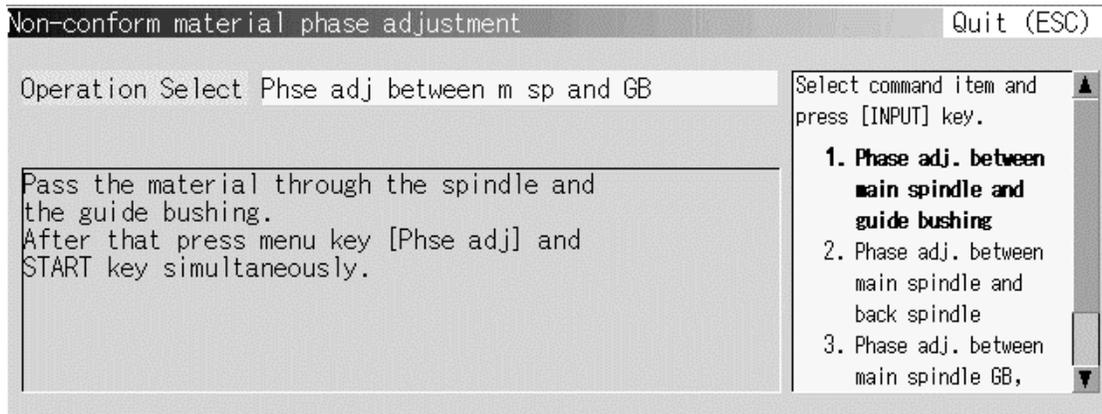


Did You Know? Non-conform Material Phase Adjustment

Looking to run or pick off non-round bar stock? Setting the non-conform material phase adjustment function on any Mitsubishi-controlled Citizen-Cincom Swiss is quick and easy. The adjustment orients the main spindle C-axis to the sub spindle C-axis with the push of a button.

The phasing between collets and jobs can change, so orienting and storing shifts automatically will save you time and worry. While in Preparation Mode:

- Pass your material through the main guide bushing and sub spindle
- Press the Non-Conform soft key **for the menu shown below**
- Press Cycle Start
 - The sub spindle C-axis will orient to the main spindle C-axis



Miss our Last Tech Tips? Click Below!

McCLAIN *Committed to 100% Customer Satisfaction*
MACHINE TOOL SOLUTIONS *Call (314) 432-3220*
Zeman Technologies

Did You Know?
Standard Control Features on Enshu Horizontals

ENSHU pallet changing Horizontal Machining Centers are built to last, with 5,000 Hours Mean Time Between Failure. The following control features standard on ENSHU contribute to speed, reliability, and long term accuracy.

Manual Operation of Automatic Pallet Changer

This feature allows an operator to rotate the pallet shuttle by hand to easily clean, maintain, and check fixture clearance safety while pallets exchange.

1. Select Manual Mode
2. Select desired pallet using the APC Drive Switch
3. Press 'APC Drive' button to stop through pallet change
4. Release the 'APC Drive' button at any time to stop that operation, and press again to resume operation.

Interested in learning more about Enshu?

Tool Change Arm Recovery

If the tool change arm is interrupted, there are two recovery methods:

McCLAIN *Committed to 100% Customer Satisfaction*
MACHINE TOOL SOLUTIONS *Call (314) 432-3220*
Zeman Technologies

Did You Know?
Macro Programs Using G & M Codes

On Fanuc & Mitsubishi controlled machines, users can create custom macro programs that are called with simple G (Geometric) or M (Machine) codes that the user defines in parameters. Any redundant action - returning axes home, drilling cycles, custom boring cycle, etc. - can be stored as a macro program that is called with a G or M code...

Example Macro Program: Move to Home Position

1. Set Parameter 6050 to 25 (See Fanuc/Mitsubishi Manual)
2. Hit Camera G26 to Call Program 9010
3. Write Program 9010 Below.

```
G9010  
G00G53 Z0  
G53X0Y0  
M99
```

Now G25 and Cycle Start saves 3+ button pushes and a mode selection.

Watch our video tutorial on Special G Codes here

Explanation

Website

Products

News

Support

Contact Us

Tricky Workpiece? Let our Expert Engineers Take a Look...

