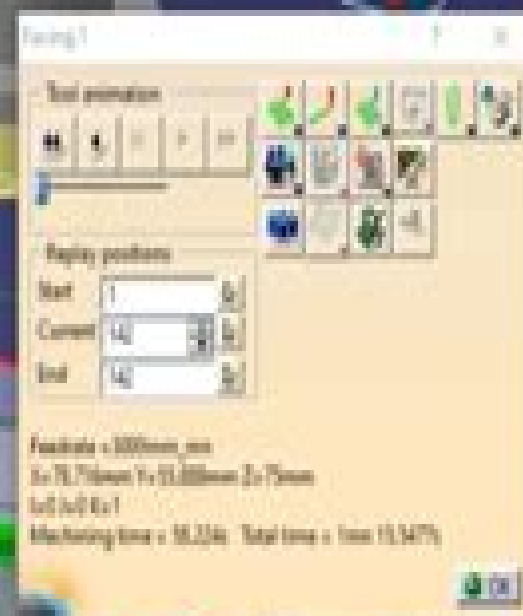




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# Macro In Catia Tutorial

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**Advanced Manufacturing and Information Engineering, Intelligent Instrumentation and Industry Development** J.Z. Ma, Fang Shao, L.P. Hu, J. Liu, D.M. Chen, 2014-08-11 Selected peer-reviewed papers from the 2014 2nd International Conference on Precision Mechanical Instruments and Measurement Technology ICPMIMT 2014, May 30-31, 2014, Chongqing, China.

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*AIAA Aerospace Sciences Meeting and Exhibit, 42nd*, 2004 **Proceedings of the ASME Fluids Engineering Division Summer Conference, 2006: Forums** American Society of Mechanical Engineers. Fluids Engineering Division, 2006 *Catia V5-6r2017 ASCENT - Center for Technical Knowledge*, 2020-01-06 The CATIA V5 6R2017 Visual Basic Automation learning guide provides you a good understanding of the different ways to automate tasks using CATIA macros and Visual Basic programming Using hands on practices you will use VB programming to work with parts assemblies drawings selections parameters and formulas graphic properties and to exchange data with Microsoft Excel This guide was written against CATIA V5 6R2017 Service Pack 1 Topics Covered CATIA V5 Object Model Creating Part Design and Shape Design features Working with Product Structure and Assembly Design Scripting Drawing Views Frames and Title Blocks Deleting Cutting Copying Pasting CATIA objects Interactive Selections Communication with MS Office Prerequisites Visual Basic programming and working knowledge of CATIA

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