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*Parametric 2.0 2. Hit Select*

*Working Directory on the top*

*bar and select whatever*

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*folder you want your new part to go into. 3. Next hit the New Button, make sure the type is set to part. Change the name to*

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*2 - 2 Creating a Simple  
Object (Part I) Figure 1  
Part at the end of this*

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*lesson Figure 2 Creating a  
new part Start Creo  
Parametric as usual. If it  
is already up, close all  
windows (except the base  
window) and erase all  
objects in session using  
File ' Manage Session '*

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*Erase Current and/or File '  
Manage Session ' Erase Not  
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Navigator and*

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Parametric1.0. PTC released  
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Spring of 2012. This book is  
fully compatible with Creo  
Parametric 2.0 except for  
the changes shown in this*

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*insert. Chapter 1 1-23 The  
Reorient command is located  
on the Named Views drop-down  
list.*

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video tutorial you will  
learn to setup Creo  
Parametric so that when  
creating an extrude feature,  
the direction of the feature*

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*determines if the extrude  
will add or remove material  
from the model.*

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2d04](#)

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