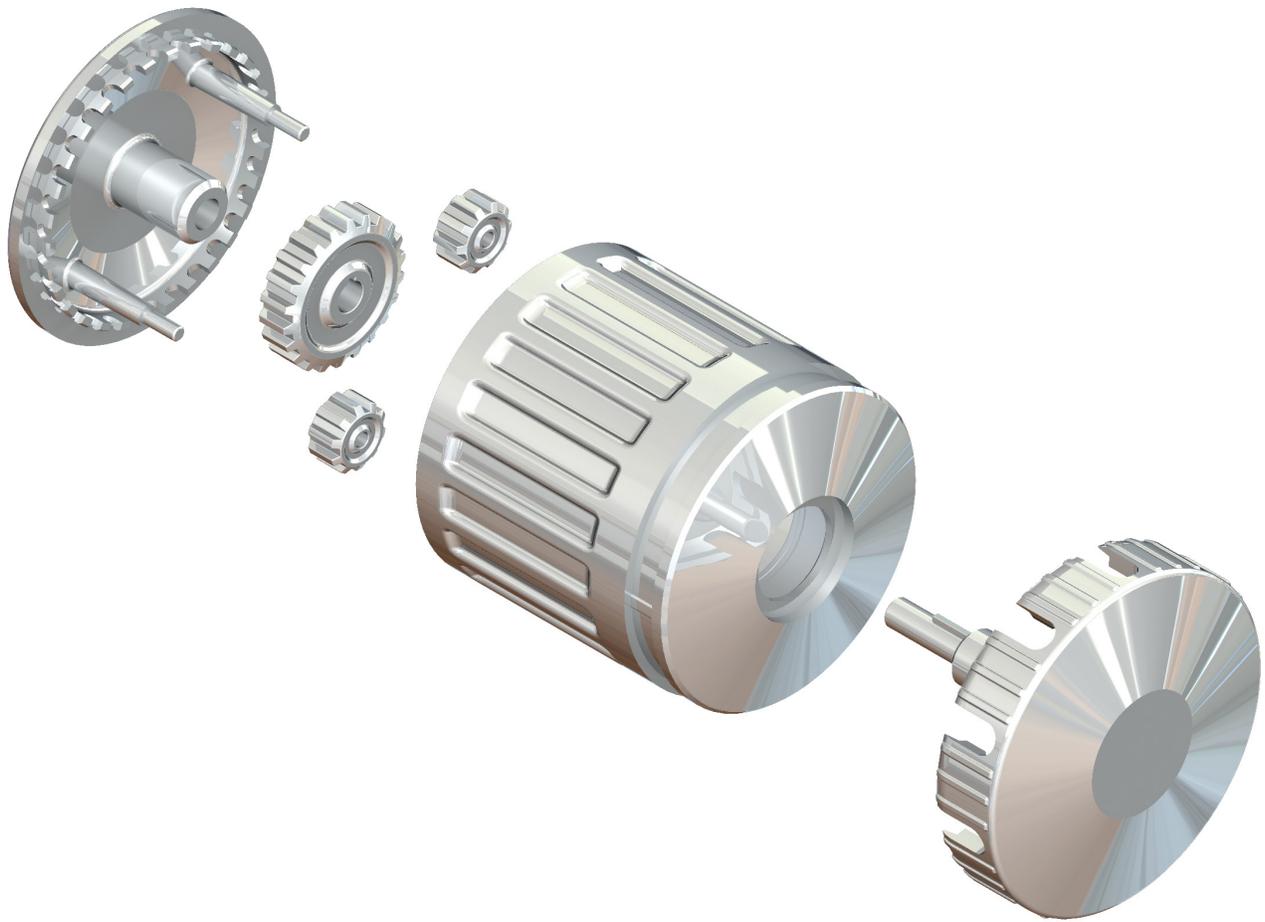


# SOLIDWORKS® 2020

## Basic Tools

Getting Started with Parts,  
Assemblies and Drawings



Paul Tran CSWE, CSWI

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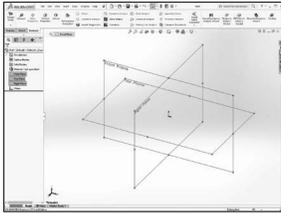
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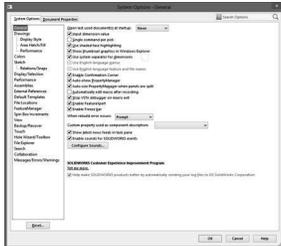
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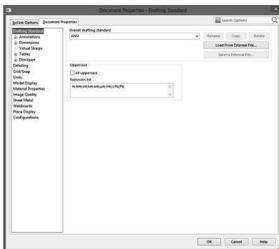
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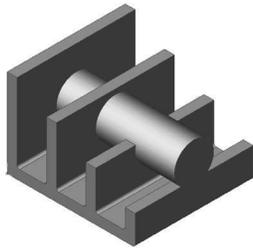
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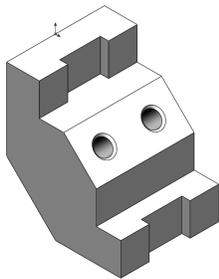
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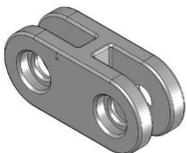
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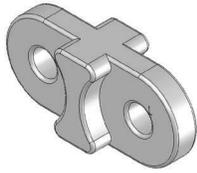
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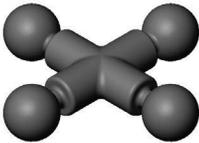
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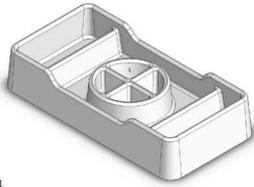


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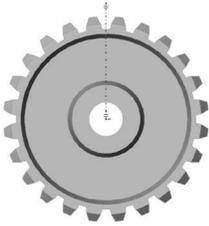


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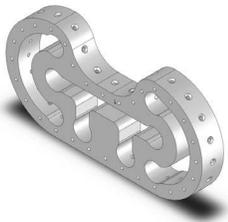
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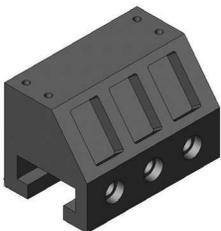


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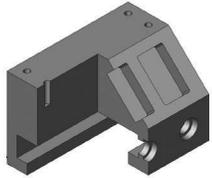
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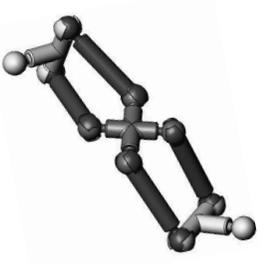
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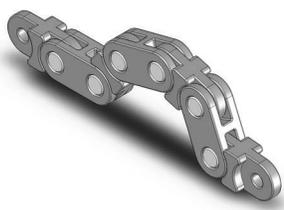
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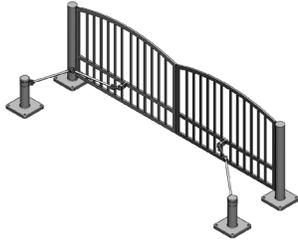
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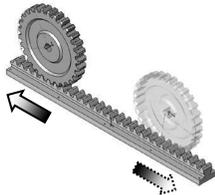


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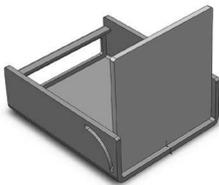
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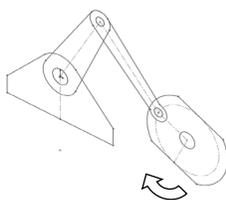
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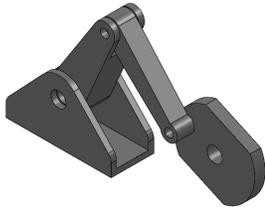
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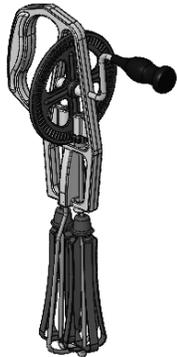
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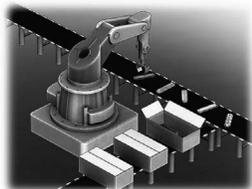
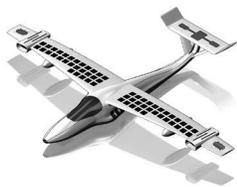


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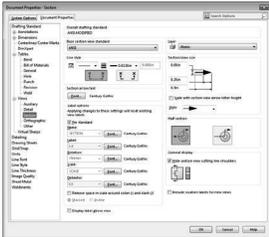
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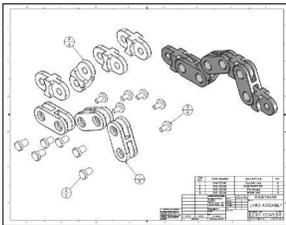
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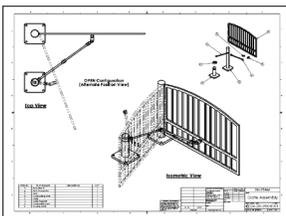
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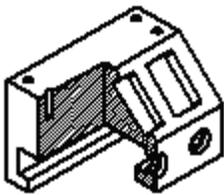
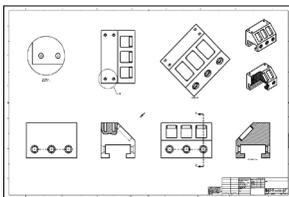
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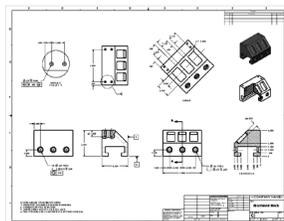
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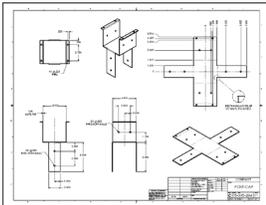
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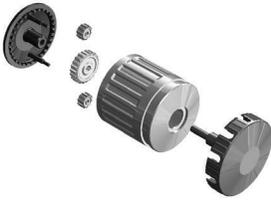
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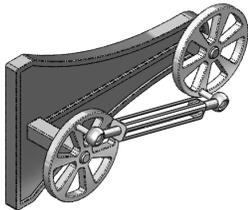
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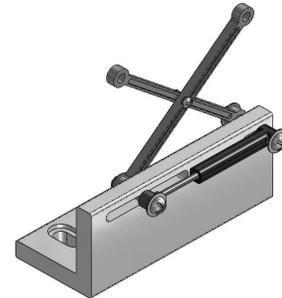
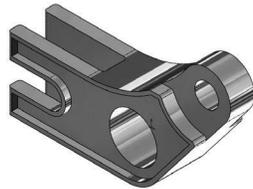
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